Parametric Study of Vertical Ground Loop Heat Exchangers for Ground Source Heat Pump Systems

Abdelrahman S. Ramadan
The University of Western Ontario

Supervisor
Dr. Karman Siddiqui
The University of Western Ontario

Graduate Program in Mechanical and Materials Engineering

A thesis submitted in partial fulfillment of the requirements for the degree in Master of Engineering Science

© Abdelrahman S. Ramadan 2016

Follow this and additional works at: https://ir.lib.uwo.ca/etd

Recommended Citation
https://ir.lib.uwo.ca/etd/3521
Abstract

We report on a numerical study conducted to investigate the effect of various parameters on the heat exchange inside a vertical ground loop heat exchanger (VGLHE) for a ground-source heat pump (GSHP) system. The simulations were conducted for three piping configurations of the ground loop which were U-Tube, Concentric pipes and Spiral. The results show a linear temperature rise along the pipe length for the U-Tube configuration. The concentric pipes configuration shows two distinct linear trends for the temperature rise; a slow temperature rise during the downward flow through the inner pipe and a higher temperature rise during the upflow through the annulus. The spiral configuration shows a steeper slope for the temperature rise in the spiral section and almost a flat slope for the temperature rise in the straight vertical section of the pipe. The research also examines a simulation case of integrating a VGLHE inside a micro-pile foundation system.

Keywords

Ground source heat pumps, vertical ground loop heat exchanger, numerical modelling, computational fluid dynamics, ANSYS FLUENT, micro-pile, geothermal, parametric study.
Statement of Originality

This is to certify that, to the best of my knowledge, the content of this thesis is my own work. This thesis has not been submitted for any degree or other purposes.

I certify that the intellectual content of this thesis is the product of my own work and that all the assistance received in preparing this thesis and sources have been acknowledged.

Abdelrahman Saied Ramadan
Acknowledgments

I would like to express my gratitude and appreciation for both my supervisor Dr. Kamran Siddiqui and co-supervisor Dr. Hesham El-Naggar for their guidance, support and, above all, patience to help me complete my research and thesis. I’m grateful to their efforts in enhancing my knowledge and research skills as well as giving me the chance to participate in various technical conferences which added to my graduate experience.

I’m extremely thankful to my whole family who supported and motivated me in every step of the way of this long journey, especially my father.

Finally, and most importantly, I’m indebted to my wife Dima for her encouragement, support, quiet patience and unending love. I dedicate all my work to you and to our new family addition, our baby daughter “Yara”.
Table of Contents

Abstract ........................................................................................................................................ ii
Statement of Originality .................................................................................................................. iii
Acknowledgments .......................................................................................................................... iv
Table of Contents .......................................................................................................................... v
List of Tables .................................................................................................................................. viii
List of Figures ................................................................................................................................. x
List of Symbols ............................................................................................................................... xiv
List of Abbreviations ...................................................................................................................... xvi

Chapter 1: INTRODUCTION ....................................................................................................... 1

1 Introduction ................................................................................................................................. 1

1.1 Background .............................................................................................................................. 1

1.2 Ground Source Heat Pump (GSHP) Systems ........................................................................ 4

1.2.1 Components of a GSHP System ....................................................................................... 5

1.2.2 Vertical Ground Loop Heat Exchanger (VGLHE) .......................................................... 8

1.2.3 Relevant Parameters and Factors: ................................................................................... 9

1.2.4 Heat transfer modes in boreholes .................................................................................... 11

1.3 Energy Piles – Fundamentals ............................................................................................... 13

1.3.1 Pile foundations - Background ..................................................................................... 14

1.3.2 Energy Piles .................................................................................................................... 18

1.4 Literature Review ................................................................................................................... 20

1.5 Motivation and Objectives .................................................................................................... 23

1.6 Thesis Format and Layout ..................................................................................................... 24

Chapter 2: NUMERICAL MODEL ............................................................................................ 25

2 Numerical Model ....................................................................................................................... 25
2.1 Modelling Process

2.2 Exclusion of Soil Modelling

2.3 Numerical Model Development and Formulation

2.3.1 Continuity and Momentum Equations

2.3.2 Turbulence Model

2.3.3 Energy and Convective Heat Transfer Modelling

2.3.4 Energy Modelling in Solid Regions

2.3.5 Conjugate Heat Transfer

2.3.6 Boundary Conditions

2.3.7 Solution Methods and Initialization

2.4 Geometry

2.5 Mesh Generation and Dependency Test

2.6 Model Validation

2.6.1 Description of the experimental data

2.6.2 Simulation and results

Chapter 3: PARAMETRIC STUDY

3 Parametric Study

3.1 Geometrical Parameters

3.1.1 U-Tube Piping Configuration

3.1.2 Concentric Piping Configuration

3.1.3 Spiral (Helical) Piping Configuration

3.1.4 Comparison and Discussion

3.2 Parametric analysis of Thermophysical Properties and Operational Parameters

3.2.1 Thermal Conductivity of the pipe

3.2.2 Grout Thermal Conductivity
3.2.3 Flow Rate of Heat Transfer Fluid .......................................................... 80
3.2.4 Concentric VGLHE in an “Energy Micro-Pile” .............................. 83

Chapter 4 : CONCLUSIONS .............................................................................. 91

4 Conclusions ........................................................................................................ 91

4.1 Recommendations for future work ............................................................ 95

References: ............................................................................................................ 97

Appendix A ........................................................................................................... 101

Appendix B .......................................................................................................... 103

Appendix C .......................................................................................................... 104

Curriculum Vitae ................................................................................................. 105
List of Tables

Table 2-1: Geometrical dimensions of 3D model.......................................................... 39
Table 2-2: Properties of different mesh sizes used for mesh dependency test .............. 41
Table 2-3: Boundary conditions used in mesh dependency test.................................... 41
Table 2-4: Thermophysical properties of material used in simulation.............................. 41
Table 2-5: Outlet temperature for different mesh sizes ................................................. 42
Table 2-6: Physical dimensions of experimental borehole............................................. 45
Table 2-7: Thermophysical properties of materials used in the experiment..................... 45
Table 2-8: Operating conditions and properties for thermal response test........................ 46
Table 3-1: Thermophysical properties of material used in simulation............................ 51
Table 3-2: Boundary conditions used [Adopted from Esen et al. 2009] ......................... 51
Table 3-3: Area-weighted average surface heat flux values for different pipe sections... 55
Table 3-4: Calculated thermal resistances of individual VGLHE components................. 59
Table 3-5: Boundary conditions used [Adopted from Esen et al. 2009] ......................... 62
Table 3-6: Boundary conditions used [Adopted from Esen et al. 2009] ......................... 69
Table 3-7: Summary of simulation results for three different piping configurations ...... 74
Table 3-8: Summary of simulation results for different pipe materials in concentric pipe configuration .................................................................................................. 76
Table 3-9: Summary of simulation results for concentric piping configurations with changing grouting thermal conductivity values................................................. 79
Table 3-10: Summary of simulation results for concentric piping configurations with varying volume flow rate of heat transfer fluid ............................................................... 81

Table 3-11: Parameters and dimensions of micro-pile used in simulation .................. 86

Table 3-12: Simulation results of energy micro-pile ................................................. 90
List of Figures

Figure 1.1: World energy consumption in quadrillion Btu, 1990-2040 [EIA 2013] .......... 2

Figure 1.2: World energy consumption by fuel type, 1990-2040 (quadrillion Btu) [EIA 2013] ........................................................................................................................................ 2

Figure 1.3: Total US energy consumption by sector in 2013 [US DOE 2013] .............. 3

Figure 1.4: Percentage distribution of solar energy [Omer 2008] ............................. 4

Figure 1.5: Depth dependence of annual range of ground temperatures in Ottawa, Canada [Williams and Gold 1976] ........................................................................................................ 5

Figure 1.6: Work and heat flow balance for a heat pump [NRC 2005] ...................... 6

Figure 1.7: Refrigeration cycle of a typical heat pump unit [NRC 2005] .................... 6

Figure 1.8: Typical GHE loop configurations for GSHP systems. a) Vertical closed GHE configuration, b) Horizontal closed GHE configuration and c) Open loop groundwater configuration. [NRC 2005] .................................................................................................................................. 7

Figure 1.9: Top view and side view of a single pipe U-Tube piping configuration in a VGLHE. .................................................................................................................................................. 9

Figure 1.10: Cross section of a U-tube borehole and corresponding thermal circuit ...... 11

Figure 1.11: Typical side views of a) End Bearing Pile and b) Friction Pile [Beardmore 2012] .................................................................................................................................................. 15

Figure 1.12: Typical hollow bar micro-pile (Drbe et al. 2013) ................................. 17

Figure 1.13: Photo of a typical reinforcement cage for a foundation pile integrated with high density polyethylene piping [BINE IS, 2010] ......................................................................................... 19

Figure 1.14: Photo of the integrated reinforcement cage inserted inside the pile borehole [Skanska, 2011] ........................................................................................................................................ 20
Figure 2.1: A schematic diagram showing horizontal and vertical cross sections of a vertical U-tube GHE (Development of a numerical model for the simulation of vertical U-tube ground heat exchangers) ................................................................. 26

Figure 2.2: Section view of simulated components and type of heat transfer ................. 33

Figure 2.3: Algorithm illustrating steps of a Pressure-Based solution ............................. 36

Figure 2.4: Top view and side view of U-Tube piping geometry domain used in simulation.................................................................................................................................................. 37

Figure 2.5: Top isometric view of the vertical ground loop 3D geometry ..................... 37

Figure 2.6: Bottom isometric view of the vertical ground loop 3D geometry ............... 38

Figure 2.7: Top isometric view of generated mesh....................................................... 40

Figure 2.8: Plot of wall y+ along depth of pipe (top 0.5m) ............................................. 42

Figure 2.9: Entrance length for fluid flow for mesh #4 ................................................. 43

Figure 2.10: Schematic of an in-situ test setup (adapted from Esen et al [2009]) .......... 44

Figure 2.11: Comparison between experimentally measured and numerically simulated fluid outlet temperatures ........................................................................................................................................................................ 47

Figure 3.1: Top view and side view of U-Tube piping geometry domain used in simulation.................................................................................................................................................. 50

Figure 3.2: Temperature of fluid along the pipe ............................................................. 52

Figure 3.3: Temperature distribution in the borehole horizontal plane at depths of (a) 0 m, (b) 10 m, (c) 20 m, (d) 30 m. The colour bar represents the temperature in degree Celsius.......................................................... 54

Figure 3.4: Temperature distribution in the fluid horizontal plane at depths of (a) 0 m, (b) 10 m, (c) 20 m, (d) 30 m. The colour bar represents the temperature in degree Celsius.. 56
Figure 3.5: Temperature profile of fluid at different depths of down-flow pipe ............... 57
Figure 3.6: Sketch indicating the location of temperature data points .......................... 58
Figure 3.7: Normalized temperature profile of fluid at different depths of down-flow pipe .................................................................................................................. 58
Figure 3.8: (a) Temperature and (b) velocity contours of the bottom bend of pipe ......... 60
Figure 3.9: Top view and side view of concentric piping geometry domain used in simulation .................................................................................................................. 62
Figure 3.10: Area-averaged temperature of fluid along the pipe ................................. 63
Figure 3.11: Temperature distribution in the borehole horizontal plane at depths of (a) 0 m, (b) 10 m, (c) 20 m, (d) 29 m. The colour bar represents the temperature in degree Celsius ........................................................................................................ 65
Figure 3.12: Temperature distribution in the fluid horizontal plane at depths of (a) 0 m, (b) 10 m, (c) 20 m, (d) 29 m. The colour bar represents the temperature in degree Celsius. ........................................................................................................ 66
Figure 3.13: (a) Temperature and (b) velocity contours in the bottom section of the concentric pipe ........................................................................................................... 67
Figure 3.14: Isometric view of 3D model of spiral piping configuration simulated ......... 69
Figure 3.15: Average temperature of fluid along the length of loop ............................. 70
Figure 3.16: Temperature distribution in the borehole horizontal plane at depths of (a) 0 m, (b) 2 m, (c) 4 m, (d) 6 m, (e) 8 m, (f) 10 m. The colour bar represents the temperature in degree Celsius ........................................................................................................ 71
Figure 3.17: Temperature distribution in the fluid horizontal plane at depths of (a) 0 m, (b) 2 m, (c) 4 m, (d) 6 m, (e) 8 m, (f) 10 m. The colour bar represents the temperature in degree Celsius ........................................................................................................ 72
Figure 3.18: Temperature of fluid at different positions of the pipe for three different piping configurations ................................................................. 74

Figure 3.19: Average fluid temperature along full length of concentric piping for four cases with different piping materials. ................................................................. 77

Figure 3.20: Temperature of fluid along full length for three concentric piping configurations with changing grout thermal conductivity values................................. 80

Figure 3.21: Temperature of fluid along full length for three concentric piping configurations with changing fluid volume flow rates values ........................................ 82

Figure 3.22: Typical hollow bar micro-pile (Drbe et al. 2013) ........................................ 84

Figure 3.23: Top view and side view of energy micro-pile geometry domain used in simulation.................................................................................................................. 85

Figure 3.24: Ground temperature profile for the Goderich area in south western Ontario (Markle, 2011) ........................................................................................................ 87

Figure 3.25: Temperature of fluid along full length of pipe ........................................ 89
## List of Symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$c$</td>
<td>Specific Heat Capacity, J/kg K</td>
</tr>
<tr>
<td>$^\circ$C</td>
<td>Celsius Degree</td>
</tr>
<tr>
<td>$C_{1e}$</td>
<td>Constant (experimentally determined)</td>
</tr>
<tr>
<td>$C_2$</td>
<td>Constant (experimentally determined)</td>
</tr>
<tr>
<td>$COP_{h/c}$</td>
<td>Design Heating/Cooling Coefficient of Performance</td>
</tr>
<tr>
<td>$D_H$</td>
<td>Hydraulic Diameter, m</td>
</tr>
<tr>
<td>$E$</td>
<td>Total Energy Transported, J</td>
</tr>
<tr>
<td>$F_{h/c}$</td>
<td>Part load factor (full load hours to total number of hours in design month)</td>
</tr>
<tr>
<td>$\vec{g}$</td>
<td>Gravitational Acceleration Constant, m/s$^2$</td>
</tr>
<tr>
<td>$G_b$</td>
<td>Generation of turbulence kinetic energy due to buoyancy</td>
</tr>
<tr>
<td>$G_k$</td>
<td>Generation of turbulence kinetic energy due to the mean velocity gradient</td>
</tr>
<tr>
<td>$k$</td>
<td>Turbulence Kinetic Energy</td>
</tr>
<tr>
<td>$K_{eff}$</td>
<td>Effective Thermal Conductivity, W/(m*K)</td>
</tr>
<tr>
<td>$L_e$</td>
<td>Entrance Length, m</td>
</tr>
<tr>
<td>$\dot{m}$</td>
<td>Mass Flow Rate, kg/s</td>
</tr>
<tr>
<td>$p$</td>
<td>Pressure, Pa</td>
</tr>
<tr>
<td>Pr</td>
<td>Prandtl number</td>
</tr>
<tr>
<td>$Q_C$</td>
<td>Heat extracted from the cold reservoir, kW</td>
</tr>
<tr>
<td>$Q_H$</td>
<td>Heat added to the hot reservoir, kW</td>
</tr>
<tr>
<td>Re</td>
<td>Reynolds Number</td>
</tr>
<tr>
<td>$R_p$</td>
<td>Pipe Thermal Resistance, °K/W</td>
</tr>
<tr>
<td>$R_s$</td>
<td>Soil Thermal Resistance, °K/W</td>
</tr>
<tr>
<td>$S$</td>
<td>Modulus of the mean rate-of-strain tensor</td>
</tr>
<tr>
<td>$t$</td>
<td>Time, s</td>
</tr>
</tbody>
</table>
\( T_{\text{ewt, min/max}} \)  Minimum/Maximum design entering water temperature, °C

\( T_{\text{g, min/max}} \)  Minimum/Maximum undisturbed ground temperature, °C

\( TI \)  Turbulent Intensity

\( T_i \)  Temperature data point in the mid vertical plane, °C

\( T_{\text{inlet}} \)  Fluid Inlet Temperature, °C

\( T_{\text{outlet}} \)  Fluid Outlet Temperature, °C

\( \bar{v} \)  Velocity, m/s

\( W \)  Work input, kW

\( x \)  Cartesian x coordinate

\( y \)  Cartesian y coordinate

\( YM \)  Contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate

\( z \)  Cartesian z coordinate

Greek Symbols

\( \mu_t \)  Turbulent Viscosity

\( \epsilon \)  Dissipation Rate

\( \mu \)  Dynamic Viscosity, kg/m·s

\( \rho \)  Density, kg/m³

\( \sigma_k \)  Turbulent Prandtl numbers for \( k \) (Turbulence Kinetic Energy)

\( \sigma_\epsilon \)  Turbulent Prandtl numbers for \( \epsilon \) (Dissipation Rate)

\( \bar{\tau} \)  Stress Tensor

\( \bar{\Omega}_{ij} \)  Mean rate-of-rotation tensor in a rotating reference frame
## List of Abbreviations

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>COP</td>
<td>Coefficient of Performance</td>
</tr>
<tr>
<td>EIA</td>
<td>Energy Information Administration</td>
</tr>
<tr>
<td>GHE</td>
<td>Ground Heat Exchanger</td>
</tr>
<tr>
<td>GSHP</td>
<td>Ground Source Heat Pump</td>
</tr>
<tr>
<td>HDPE</td>
<td>High Density Poly Ethylene</td>
</tr>
<tr>
<td>HT Fluid</td>
<td>Heat Transfer Fluid</td>
</tr>
<tr>
<td>SIMPLE</td>
<td>Semi-Implicit Method for Pressure-Linked Equations</td>
</tr>
<tr>
<td>TRT</td>
<td>Thermal response Test</td>
</tr>
<tr>
<td>UDF</td>
<td>User Defined Function</td>
</tr>
<tr>
<td>USDOE</td>
<td>US Department of Energy</td>
</tr>
<tr>
<td>VGLHE</td>
<td>Vertical Ground Loop Heat Exchanger</td>
</tr>
</tbody>
</table>
Chapter 1: INTRODUCTION

1 Introduction

1.1 Background

The 2013 International Energy Outlook published by the U.S. Energy Information Administration (EIA) projected a continuous increase in the world energy consumption levels over the next few decades [EIA 2013]. As shown in Figure 1.1, it is estimated that the world energy consumption will grow by approximately 56% between 2010 and 2040. Different energy sources will be required to meet this increasing demand and as can be seen by Figure 1.2, fossil fuels (oil, coal and gas) are predicted to account for approximately 75% of the world energy consumption by 2040 [EIA 2013]. This fossil fuel dependency outlook is most certainly reinforced by the recent drop in crude oil prices globally. The heavy reliance on fossil fuels combined with the fact that burning fossil fuels produces greenhouse gases such as nitric oxide and carbon dioxide, has generated growing concerns related to the environmental impacts of these fuels. It is estimated that the world energy-related carbon dioxide emissions will increase by approximately 36% by 2040 (due mainly to non-OECD countries) [EIA 2013]. This massive global energy consumption has always demanded engineers, scientists and designers to explore other renewable resources and more efficient systems.

The largest energy consuming sector in the U.S. is the building sector (consisting of residential and commercial buildings). As shown in Figure 1.3, it is estimated that approximately 40% of total U.S. energy consumption in 2013 (97.4 Quadrillion Btu) was utilized by buildings, which was higher than each of the other two remaining sectors, transportation and industrial [US DOE 2013]. Approximately 50% of the total energy consumed by buildings is used for space heating and air conditioning [EIA 2013]. This presents a major potential opportunity for utilizing renewable energy sources for space heating and cooling, which reduces the use and current dependency on fossil fuels and consequently the associated negative environmental effects.
Figure 1.1: World energy consumption in quadrillion Btu, 1990-2040 [EIA 2013]

Figure 1.2: World energy consumption by fuel type, 1990-2040 (quadrillion Btu) [EIA 2013]
One of these potential renewable energy sources is the earth’s ground energy, which is utilized via a Ground Source Heat Pump (GSHP) system with considerable economic advantages and cost savings. The GSHP system operates on the basis of using ground as a heat source for heating purposes in winter and as a heat sink for cooling purposes in summer. It is argued that geothermal heating systems can be more efficient than electric resistance heating, gas or oil-fired heating systems and air-source heat pumps [Omer 2008].

Although GSHP systems could be viewed as attractive and feasible from the energy cost point of view, the installation costs associated with the drilling of the boreholes containing the ground heat exchangers could be prohibitive and could act as a barrier for increasing the use and installation of beneficial GSHPs. Nonetheless, relatively novel methods have been explored in order to reduce this premium installation cost and one of these methods is the “Energy Pile” system where it utilizes the building structural foundation piles as ground heat exchangers.

In the next sections a background for GSHP and energy pile systems will be presented as well as the objective and motivation for this research.
1.2 Ground Source Heat Pump (GSHP) Systems

It has been well established that the main source of energy on earth is solar radiation [Phillips 1995], which also serves as the source for many global energy sources available to us including solar energy, wind energy, petrochemical and more importantly geothermal and earth-energy systems. Considering that almost half of the sun’s solar energy gets absorbed by earth as shown in Figure 1.4, it seems logical to utilize this abundant storage of renewable energy that is readily available on site all year long.

![Figure 1.4: Percentage distribution of solar energy [Omer 2008]](image)

With that solar energy being stored underground as thermal energy, the soil also acts as an insulator between the ambient air above ground and the earth below. This insulation produces a constant ground temperature below a certain depth all year round that is independent of the above ground air temperatures that fluctuate due to seasonal variations as shown in Figure 1.5. The temperature of the ground at shallower depths may not be constant; however, their fluctuations are greatly reduced when compared to the ambient air temperatures above.
Therefore, the temperature of the ground below a certain depth is warmer than the ambient air in winter and is cooler than the ambient air in summer. This is the basis of operation for ground source heat pump systems (GSHP) where heat can be absorbed from the relatively warm ground in winter and rejected into the relatively cool ground in the summer through the use of ground source heat pumps.

1.2.1 Components of a GSHP System

Ground source heat pump systems consist of three main components:

1- Heat pump machine:

The ideal heat pump “pumps” heat from a cold source to a hot reservoir through the application of work as shown in Figure 1.6 below (opposed to the natural flow of heat from hot to cold). This cycle is known as the vapour compression refrigeration cycle and is typical for any heat pump whether ground-source, water-source, or air-source as shown in Figure 1.7.
The function of the heat pump in a GSHP system is to transfer the heat between the earth connection and the heating/cooling distribution system. The most common kind of GSHPs is the “water-to-air” where the heat carried by the fluid from and to the earth connection is ultimately transferred to and from the air distribution system within the building. If the distribution system is hydronic (heating/cooling water loop) then the GSHP would be a “water-to-water” type.
All of the heat pump components shown are usually contained in one enclosure and the enclosure itself is placed indoors in a furnace room or a mechanical room. Typical capacities for GSHPs range from 3.5kW to 35kW [NRC 2005].

2- Heating/Cooling distribution system:

This is the distribution system which delivers heating or cooling to the building of interest. The distribution system is usually the conventional air duct or hydronic (hot water) piping distribution system.

3- Earth connection:

The earth connection is the ground loop piping system that acts as the heat exchanger between the ground and the GSHP system. There are two ground heat exchanger (GHE) systems; closed loop and open loop systems. Within each system there are different configurations of the underground piping loop layouts: vertical, horizontal, coiled, surface water and open system wells. Some examples are shown in Figure 1.8 below.

![Figure 1.8: Typical GHE loop configurations for GSHP systems. a) Vertical closed GHE configuration, b) Horizontal closed GHE configuration and c) Open loop groundwater configuration. [NRC 2005]](image)

In the closed loop systems, the GSHP system circulates a heat transfer fluid, usually water or an antifreeze glycol mixture from the heat pump machine to the ground loop and back into the heat pump. In the open loop system, the GSHP system continuously pumps
water from the designated well or aquifer to the heat pump and then returns it back to the environment through injection wells.

In the present research, the type of GHE of interest is the vertical closed loop type, hereinafter referred to as “Vertical Ground Loop Heat Exchanger” or “VGLHE”, and therefore more related information will be presented here.

1.2.2 Vertical Ground Loop Heat Exchanger (VGLHE)

A typical Vertical Ground Loop Heat Exchanger (VGLHE) consists of three key components, as shown in the single pipe U-tube configuration in Figure 1.9:

1- Heat Transfer Fluid  
2- Piping  
3- Grout material

The depth and diameter of the boreholes containing the VGLHE vary from case to case depending on many specific factors, such as the peak heating load, peak cooling load, number of VGLHEs in a system, thermal properties of the soil and others. However, the diameter and depth of the borehole generally range from 100 mm to 150 mm and from 15 m to 120 m, respectively [GSHPA 2007].

The piping material commonly used in GSHP installations is High Density Polyethylene (HDPE) and the pipe nominal sizes typically range from 20 mm 40 mm. The function of the piping is to convey the heat transfer fluid, typically water or an anti-freeze mixture, to circulate into and out of the VGLHE from and back to the heat pump equipment. There are different configurations of the piping other than the common single pipe U-tube configuration, such as the concentric piping, spiral piping and the double pipe U-tube configurations.
The third component surrounding the piping is the grout. The grout prevents the ground water from the risk of contamination and provides a thermal connection to the surrounding soil. It is for the latter reason that the grout material is preferable to a have a high thermal conductivity value. Typical geothermal grout material is a bentonite grout mixture.

1.2.3 Relevant Parameters and Factors:

The efficiency of any heat pump including GSHPs is measured by a parameter called the coefficient of performance (COP). The COP is the ratio of change in heat at the reservoir of interest to work input in the process. Generally ground source heat pumps have heating COPs ranging from 2.4 to 5.0 and cooling COPs ranging from 3.1 to 8.8 [Rafferty et al 1997].
Heating and cooling COPs are calculated using the following equations:

\[ Q_H = W + Q_C \]  
\( Eq. \ 1-1 \)

\[ COP_{Heating} = \frac{Useful \ Heating \ energy}{Compressor \ and \ Fan \ and \ Pump \ Energy} = \frac{W + Q_C}{W} \]  
\( Eq. \ 1-2 \)

\[ COP_{Cooling} = \frac{Useful \ Cooling \ energy}{Compressor \ and \ Fan \ and \ Pump \ Energy} = \frac{Q_C}{W} \]  
\( Eq. \ 1-3 \)

Where:

\( Q_C \) is the amount of heat extracted from the cold reservoir  
\( Q_H \) is the amount of heat added to the hot reservoir  
\( W \) is the work input

Another important objective when designing for a GSHP system is the proper sizing of the GHE length. This is a critical consideration since the capital costs of GSHP system are higher than conventional systems. Below are the relevant equations from a simplified method from the International Ground Source Heat Pump Association (IGSHPA) [IGSHPA 1988]:

Sizing based on the heating load:  
\[ L_h = q_{d,heat} \left\{ \frac{(COP_h - 1)}{COP_h} \left( R_p + R_s F_h \right) \right\} \frac{T_{g,\min} - T_{ewt,\min}}{T_{ewt,\min} - T_{ewt,\max}} \]  
\( Eq. \ 1-4 \)

Sizing based on the cooling load:  
\[ L_c = q_{d,cool} \left\{ \frac{(COP_c - 1)}{COP_c} \left( R_p + R_s F_c \right) \right\} \frac{T_{ewt,\max} - T_{g,\max}}{T_{ewt,\min} - T_{ewt,\max}} \]  
\( Eq. \ 1-5 \)

Where:

\( COP_{h/c} \) is the design heating/cooling coefficient, respectively  
\( T_{g,\min/\max} \) is the minimum/maximum undisturbed ground temperature  
\( T_{ewt,\min/\max} \) is the minimum/maximum design entering water temperature  
\( R_p \) is the pipe thermal resistance
$R_s$ is the soil thermal resistance

$F_{h/c}$ is the part load factor (full load hours to total number of hours in design month)

The design entering water temperatures are estimated using the following equations [Kavanaugh et al 1997]:

\[
T_{ewt,min} = T_{g,min} - 15^\circ C
\]  
Eq. 1-6

\[
T_{ewt,max} = \min(T_{g,max} + 15^\circ C, 43^\circ C)
\]  
Eq. 1-7

1.2.4 Heat transfer modes in boreholes

In a borehole, the heat exchange takes place between the soil and the heat transfer fluid which serve as the temperature nodes. The heat flows between these two nodes from the higher temperature node to the lower temperature node. During its passage from one node to the other, the heat flow experiences resistances, which influence the magnitude of the heat transfer rate. These resistances which are also termed as “thermal resistances” depend on the material properties as well as the flow behaviour (when present).

The thermal resistance of a borehole at a 2D section can be represented through a thermal circuit as shown in Figure 1.10.

![Figure 1.10: Cross section of a U-tube borehole and corresponding thermal circuit](image-url)
$T_{f1}$ and $T_{f2}$ are the temperatures of the heat transfer fluid in pipe 1 and pipe 2, respectively.

$R_{f1}$ and $R_{f2}$ are the convective thermal resistances between the flowing fluid and the pipe within pipe 1 and pipe 2, respectively.

$T_{p1,i}$ and $T_{p2,i}$ are the temperatures of the inner surface of pipe 1 and pipe 2, respectively.

$R_{p1}$ and $R_{p2}$ are the wall conductive thermal resistances of pipe 1 and pipe 2, respectively.

$T_{p1,o}$ and $T_{p2,o}$ are the temperatures of the outer surface of pipe 1 and pipe 2, respectively.

$R_g$ is the conductive thermal resistance within the grout body.

$T_g$ is the average wall temperature of borehole’s grout wall.

$R_s$ is the conductive thermal resistance of the surrounding soil.

$T_s$ is the average temperature of the surrounding soil.

The figure shows a 2D heat transfer process and two heat fluxes, $q_1$ and $q_2$. $q_1$ is the heat transferred between the soil and the fluid in pipe 1, and $q_2$ is the heat transferred between the soil and the fluid in pipe 2.

For the purposes of this research the simulation domain has been selected to include only the fluid, piping, and grout. The research excluded the soil effect or behaviour as described later in section 2.2. Based on that, the soil temperature, $T_s$, and soil thermal resistance, $R_s$, will be excluded from this analysis.

As shown in Figure 1.10, the total thermal resistance between the fluid nodes and the grout wall consists of conductive and convective resistances. These thermal resistances are defined by the following equations.

The resistance of the fluid is calculated using the following equation [Drake et al 1972]:

$$R_{f1} = \frac{1}{2\pi r_{1i} h_1}, \quad R_{f2} = \frac{1}{2\pi r_{2i} h_2} \quad \text{Eq. 1-8}$$

Where,

$h_1$ and $h_2$ are the convective heat transfer coefficients of the fluid inside pipe 1 and pipe 2, respectively

$r_{1i}$ and $r_{2i}$ are the inside radii of pipe 1 and pipe 2, respectively
The resistance of the pipe is calculated using the following equation [Drake et al 1972]:

\[ R_p = \frac{\ln \left( \frac{r_{1o}}{r_{1i}} \right)}{2\pi k}, \quad R_p = \frac{\ln \left( \frac{r_{2o}}{r_{2i}} \right)}{2\pi k}, \quad \text{Eq. 1-9} \]

Where,

- \( k \) is the thermal conductivity of the pipe
- \( r_{1o} \) and \( r_{2o} \) are the outside radii of pipe 1 and pipe 2, respectively

The thermal resistance of the grout involves 2D heat transfer consideration since the source/sink is embedded within the grout. This calculation requires the computation of the conduction shape factor, which is based on the configuration and dimensions of the heat source/sink (Incropera 2007). \( R_g \) could also be calculated from the average temperature profile at the wall of the borehole and the surface of the U-tube pipes using the following equations [Hellström, 1991]:

\[ R_g = \frac{T_g - T_{p1,o}}{q_1} = \frac{T_g - T_{p2,o}}{q_2}, \quad \text{Eq. 1-10} \]

The overall heat transfer rates \( q_1 \) and \( q_2 \) can then be described using the following equations:

\[ q_1 = \frac{T_g - T_{f1}}{R_g + R_{p1} + R_{f1}} \quad \text{Eq. 1-11} \]
\[ q_2 = \frac{T_g - T_{f2}}{R_g + R_{p2} + R_{f2}} \quad \text{Eq. 1-12} \]

### 1.3 Energy Piles – Fundamentals

As mentioned earlier, Energy Piles systems are basically thermal foundation piles which serve both as a structural foundation for the building and as a ground energy heat exchanger for GSHP systems. This section provides the background description on Energy Piles.
1.3.1 Pile foundations - Background

Pile foundations are deep building foundations that transfer the structural load from the building to the soil layers below, which can provide the required load bearing capacity. They generally consist of long, slim and columnar elements installed into the ground. A pile foundation is different from a shallow foundation and typically has a depth that is more than three times its width [Atkinson 2007]. Piles are commonly constructed from reinforced concrete, steel or timber and are usually used for large building structures or in situations where the shallow soil is just not suitable for a shallow foundation structure.

Based on the functions of the pile foundations, they could be classified into two types as shown in Figure 1.5 below:

1- Friction pile foundation: The load bearing capacity of each pile is provided by the shear stresses developed by the contact between the sides of the piles and the soil.

2- End bearing pile foundation: The majority of the load bearing capacity is developed at the bottom “toe” of the pile.

The ultimate load-carrying capacity of a pile, $Q_u$, is given by the equation [Atkinson 2007]:

$$Q_u = Q_p + Q_s$$  \hspace{1cm} \text{Eq. 1-13}

Where:

$Q_p$ is the load-carrying capacity of the pile’s bottom “toe”

$Q_s$ is the frictional resistance derived from soil-pile interface

The end-bearing capacity, $Q_p$, is given by the following equation [Atkinson 2007]:

$$Q_p = A_p q_p = A_p (c^* N_c^* + q' N_q^*)$$  \hspace{1cm} \text{Eq. 1-14}
Where:

\( A_p \) is the area of pile tip

\( c' \) is the cohesion of the soil supporting the pile tip

\( q_p \) is the unit point resistance

\( q' \) is the effective vertical stress at the level of the pile tip

\( N^*_c, N^*_q \) are the bearing capacity factors

\[
Q_s = \sum p \Delta L f
\]

Eq. 1-15

Where:

\( p \) is the perimeter of the pile section
\( \Delta L \) is the incremental pile length over which \( p \) and \( f \) are taken to be constant

\( f \) is the unit friction resistance at any depth \( z \)

The installation and construction of foundation piles can be broken down to two methods:

1. Driven piles: The steel or concrete pile structure is delivered to site and then vertically driven into the soil through the action of a driving hammer machine (falling weight impact on the top of the pile).

2. Bored piles: In this installation method, the soil is extracted (bored) out of the ground first and then the concrete mixture is poured into the hole. Other variants include extraction of the soil out of the ground and pouring and pumping of the concrete mixture simultaneously [O'Sullivan, 2001].

The selection of the pile installation method depends on many site specific factors, such as the soil conditions, type of building structure, cost and available area of construction.

1.3.1.1 Micro-Pile System

One specific pile foundation type of system, the Micro-Pile, will be given more attention in this section as it will be used in a computational fluid dynamics (CFD) case simulation in later chapters.

Micro-Piles in general are deep foundation piles that typically have small diameters (less than 300 mm) [Wynne 1988]. They were initially conceived to underpin historic buildings and monuments but they have evolved ever since to become a common foundation option for new construction projects as well. They have a main advantage over other deep foundation systems when installation sites have limited access or low headroom due to the smaller installation and drilling equipment required.

As shown in Figure 1.12, a typical hollow bar micro-pile consists mainly of a sacrificial drill bit at the bottom, a threaded steel hollow bar (the pile body itself), couplers to extend the overall length of the micro-pile and the grouting around the steel pile to provide the grout to ground bond.
As the ground hole or pile shaft is being drilled by the sacrificial drill bit, the hollow core bar is advanced into the required depth of the drilled hole. High strength grouting material is then pumped inside the hollow bar and after passing through one or more nozzles in the sacrificial drill bit, it diffuses and fills up the gap between the pile and the ground [Liew et al, 2003]. Although the function of the grouting around the pile is to bind the outside surface of the threaded steel hollow bar to the adjacent soil, the typical end product usually has the grouting inside of the hollow bar steel pile.

Figure 1.12: Typical hollow bar micro-pile (Drbe et al. 2013)
1.3.2 Energy Piles

Energy piles are basically foundation piles that incorporate a closed loop VGLHE system within it to act as a heat sink in the heating season and a heat source in the cooling season. The energy pile is ultimately connected to the GSHP system serving the building. The main advantage of an energy pile system over a conventional GSHP system is that the foundation structure that houses the ground heat exchanging system is already required for structural purposes and the VGLHE does not need to be drilled or constructed separately [Suryatriyastuti et al, 2012].

This structural and thermal multi-purpose feature of the energy pile system can eliminate the prohibitive high installation cost associated with the drilling of dedicated boreholes for the GHSP system and thus reducing the premium installation cost. Another advantage of the energy pile system is the reduction of the land use which originally would have been required for a conventional GSHP system.

1.3.2.1 Construction of Energy Piles

Historically, the structural components of a building to be utilized for ground heat transfer applications were foundation slabs. Overtime, other types of structural components such as bored piles, precast driven piles and diaphragm walls were effectively developed to be used for heating and cooling applications [Thompson III, 2013]. The use of bored piles with large diameters has been increasing over the past decade and is believed to have overtaken the use of prefabricated driven piles in these heat transfer applications [Brandl, 2006].

Typical foundation piles are transformed to energy piles by retrofitting the inside of each pile with one or more loops of high density polyethylene (HDPE) piping along its depth. Regardless of the geothermal requirements, the length and diameter of the pile should be sized and designed based on the applied structural load and the skin friction required to resist.

The drilling and excavation of the soil in a bored energy pile is usually performed by lowering a rotating hollow core continuous flight auger into the ground until the required
depth of the energy pile is achieved. The liquid concrete or grout is then pressurized and pumped into the freshly drilled borehole through an outlet port on the tip of auger while it is slowly being withdrawn and risen. This is done in order to ensure the structural capacity of the pile is not compromised by vacancies in the pile.

The piping loops are typically attached and strapped onto the welded pile reinforcement cage, shown in Figure 1.13, which is then inserted and lowered into the borehole following the drilling and the removal of the soil, as shown in Figure 1.14. The reinforcement cage can be inserted before or after the liquid concrete has been poured into the borehole. The piping loops are usually internally pressurized before being inserted into the liquid concrete filled borehole to protect the piping from damage and to ensure the piping inner cross section is not crimped to allow for unrestricted flow for the heat transfer fluid once put in operation.

Figure 1.13: Photo of a typical reinforcement cage for a foundation pile integrated with high density polyethylene piping [BINE IS, 2010]
1.4 Literature Review

There are relatively few studies reported in the literature that investigated the heat transfer in VGLHEs and due to the cost and logistics, mostly comprised of numerical approaches. Furthermore, these studies were focused on a particular geometry or condition and hence, there is a lack of comprehensive parametric studies investigating the effects of main parameters (such as geometry, operational and thermo-physical properties) on the performance of VGLHEs. Lenhard et al. [2013] performed a Computational Fluid Dynamics (CFD) simulation on a double U-tube VGLHE utilizing the k-ε turbulence model. The only parameter of comparison used in this study was the depth of the VGLHE where the effects on the circulating fluid were analysed at different depths ranging from 50 m to 147 m (mesh dependency test was not demonstrated in the study). The simulation results showed that the total heat transfer rate inside the VGLHE increased linearly with depth and at a higher rate than that of the soil temperature. He et al. [2012] presented a numerical 3D modelling study of a single U-tube VGLHE in both transient and steady-state stages. The study tested and evaluated the effect of the fluid flow rates on the fluid outlet temperature and the total heat flux. They observed that the fluid temperature profile and the corresponding total heat flux, along the borehole’s
depth, were linear at relatively high fluid flow rates and noticeably non-linear at mid-range and low fluid flow rates. The study also looked at the inter-tube heat flux, also called “short-circuit” heat flux, between the upward and downward fluids flowing in the adjacent U-tube pipes and estimated it to be inversely proportional to the flow velocity. The simulation, however, assumed a constant ground temperature along the depth of the borehole.

Esen et al. [2009] conducted an experimental study analysing the effect of the borehole depth on the overall COP of the GSHP system. The experimental work included an in-situ Thermal Response Test (TRT) for a ground source heat pump system in Elazig, Turkey. It used an above ground pump with a heater circulating heat transfer fluid through the borehole piping while continuously measuring the fluid temperatures at the inlet and outlet of the borehole. The experiment looked at both summer and winter modes of operations. The study produced well documented data and readings applicable for CFD simulations. The authors have also conducted other studies validating the results of the presented experiments using Artificial Neural Network (ANN) and Adaptive Neuro-fuzzy Inference System (ANFIS) models [Esen et al 2010].

Gustafsson et al. [2010] conducted numerical investigation of two different VGLHE geometries, U-tube and concentric, using the ANSYS FLUENT software. The results of the numerical simulations were compared with published laboratory experiments. Unlike conventional U-tube borehole configuration, they modelled the U-tube immersed in the groundwater (instead of the conventional grout) followed by the surrounding soil. The study was primarily focused on the influence of the induced velocity flow in the surrounding groundwater due to the temperature gradient and the resulting density differences. They found that the induced natural convection in the groundwater significantly decreased the thermal resistance of the borehole. The VGLHE models in this study were only 3 metres deep and no direct comparison of the performances between the two different geometries was performed.

Bidarmaghz et al. [2013] studied the effects on the heat extraction rate of a VGLHE system by (i) varying the volume flow rate of the heat transfer fluid and (ii) changing the
piping configuration and geometry. The piping configurations that were simulated were the single U-tube, double U-tube and double cross U-tube. The results showed that the magnitude of the heat extraction rate increased at a high rate as the flow rate of the fluid was increased within the laminar regime (low Reynolds numbers). However, above a certain flow rate and when the flow became turbulent, the magnitude of the heat extraction rate increased at a slower rate compared to that in the laminar regime. As for the effect of the piping configuration, the results showed that the double U-tube piping configuration achieved between 40% to 90% higher extraction rate when compared to the single U-tube piping configuration of the same depth. The double and the double cross U-tube piping configurations showed very similar heat extraction rates when the fluid flow was in the laminar and transitional flow regimes, however, in the turbulent flow regime, the double U-tube piping configuration resulted in a 23% increase in the heat extraction rate.

Recently, Gashti et al. [2014] performed a 3D numerical simulation for heating/cooling operations of a ground heat exchanger incorporated within a steel pile foundation (energy pile). The results of the simulation were compared with those of a 20 m deep experimental energy pile with two different types of piping configurations (single U-tube and double U-tube) under different fluid flow rates. The study showed good agreement between the simulated and experimental performance of the energy pile which validated the simulation model. Analysis of the results indicated that an increase in the number of piping loops inside of the energy pile is more efficient than increasing the diameter of the pipes themselves (double U-tube systems performed better than single U-tube systems). This improved performance ranged from 10% to 60% depending of the fluid flow rate. The study also revealed that systems with small differences between tube inlet and ground temperatures had little difference in their power output and implied that higher temperature differences between the inlet fluid and ground temperature are required to achieve tangible differences in the power output.

Zarrella et al. [2014] used the equivalent thermal resistance and capacitance circuit approach to model heat transfer in an energy pile. The model was validated with field measurements carried out on two energy pile installations and a comparative analysis
between helical and a triple U-tube configuration inside the energy pile was conducted. The results showed that helical-pipe energy piles performed better thermally than the conventional U-tube configuration. In addition, the performance of a standard double U-tube borehole heat exchanger was compared with the modelled two energy pile configurations. As expected, the thermal performance of the double U-tube heat exchanger was lower than both energy pile configurations (30% lower than the helical-pipe energy pile and 13% lower than the triple U-tube one).

Cvetkovski [2014] conducted a detailed numerical study and simulation on the fluid flow and heat transfer behaviour at the bottom 180° bend of a U-tube VGLHE. The study investigated the effect of Reynolds and Dean Numbers on the fluid flow and heat transfer. It utilized the ANSYS FLUENT software package and the realizable $k-\varepsilon$ turbulence model to solve the associated flow and heat transfer equations. The results were validated with the values provided from experimental testing. The study concluded that in additional to redirecting the fluid flow back up, the 180° bend generated Dean’s vortices which enhanced the heat transfer significantly overall and particularly in that location. Decreasing the fluid flow velocity was found to decrease the resident time for heat transfer at the bend and hence a reduction in the outlet temperature.

1.5 Motivation and Objectives

The motivation for this research is to better understand and ultimately optimize the performance of Ground Source Heat Pump systems especially when these systems are integrated in an energy pile structure. There are many geometrical, thermo-physical, and operational parameters that could highly affect the heat exchange process between the VGLHE and the soil and hence, the COP of the whole system. For instance, there are several piping loop configurations that could be installed inside a VGLHE or an energy pile such as the U-tube, concentric and the spiral piping configuration with each configuration producing different fluid outlet temperatures and total heat transfer rates. As the above literature review shows, there is a lack of detailed parametric study to investigate the effects of these parameters on the heat transfer process. The understanding of these effects is vital in order to improve the design and selection process for GSHPs and energy piles.
Objectives

The main objectives of this proposed study are:

1. To develop a numerical 3D CFD model simulating the heat transfer process inside a Vertical Ground Loop Heat Exchanger (VGLHE) for a Ground Source Heat Pump (GSHP) system.

2. To conduct a parametric study using the developed numerical 3D model to further understand the heat transfer process and draw comparisons for different piping configurations, materials of construction and fluid flow rates.

1.6 Thesis Format and Layout

This thesis consists of four chapters. Chapter 1 provides an introduction and background information on the Ground Source Heat Pump (GSHP) system and Energy Piles, literature review, and motivation and objectives for this research. Chapter 2 describes the 3D numerical model that was developed including the modelling process, geometry, mesh generation and validation. Chapter 3 presents the detailed parametric study along with the comparison and discussion of its results. Finally, Chapter 4 presents the conclusion sections, final thoughts, comments and recommendations for future work.
Chapter 2 : NUMERICAL MODEL

2 Numerical Model

This chapter describes the 3D numerical model that was developed in order to simulate the proposed heat exchange processes in a Vertical Ground Loop Heat Exchanger (VGLHE) system. The specifics and geometry for the model are based on typical VGHLEs. This chapter begins with a description of the geometry of interest followed by the description of governing mathematical equations and models utilized for simulations. The chapter will then present the mesh dependency test results, followed by the model validation.

2.1 Modelling Process

As mentioned in the previous chapter, the main focus of this research is the vertical ground loop heat exchanger (VGLHE) component of the ground source heat pump system. This is where the heat transfer occurs between the ground and the GSHP system and has the direct impact on the heat pump system performance. The ground heat transfer process occurs within four main components that make up a typical VGLHE (shown in Figure 2.1):

1- Surrounding soil
2- Grout
3- Piping
4- Heat Transfer Fluid
2.2 Exclusion of Soil Modelling

A complete soil model analysis would need to include the following effects:

1- Impact of thermal cycling on the bond strength between the grout and the surrounding soil. As the operational mode of the system switches between cooling and heating every year and due to thermal expansion and contraction, it’s expected that the bond strength between the grout and the soil will be impacted. This is more relevant for energy pile applications where the pile’s structural strength is dependent on its “skin” friction.

2- Impact of water content or ground water in the soil on the thermal conductivity of the grout material. As the overall size of the borehole may change (due to thermal expansion) there’s the possibility of the grout absorbing some of the water content in the surrounding soil over time. This would affect the thermal conductivity of the grout and the overall thermal conductivity of the borehole.
For the purposes of this research the simulation domain has been selected to include only the fluid, piping, and grout. The research excluded the soil effect or behaviour on the performance of the VGLHE. This assumption was used to focus on the effects of varying internal VGLHE parameters, such as geometric, thermophysical and operational parameters on the overall performance while maintaining the exterior ground conditions unchanged. In addition, the work in this thesis focuses on the individual performance of the VGLHE and not the group effect; hence, the soil model could be neglected since it’s more relevant when evaluating the interactions between multiple VGLHEs. This exclusion also reduces the large computation time and resources required.

A wall temperature boundary condition was set for the model on the outer grout wall to simulate the temperature of the surrounding soil.

2.3 Numerical Model Development and Formulation

The physical modelling and meshing were constructed using the default ANSYS modeller and mesher while the simulations were solved using FLUENT 14.0. FLUENT is a numerical solver with modelling capabilities for incompressible and compressible, transient and steady-state, laminar and turbulent fluid flow problems [FLUENT 2010]. It’s very versatile in the way it allows the user to easily change boundary conditions and parameters while producing accurate simulation results.

In this section the governing mathematical equations, models and numerical assumptions implemented in the developed model are described.

2.3.1 Continuity and Momentum Equations

All CFD simulations are founded on the solution of governing equations which describe the behaviour of the flow. The CFD solver numerically solves the mass (Continuity) and momentum conservation (Navier-Stokes) equations along with other additional transport equations depending on the complexity of the flow (e.g. energy conservation, species mixing or reactions, turbulent flow). The turbulence and heat transfer governing equations will be described in the following sections. The governing and transport
equations and other related terms presented in this Chapter that provide the model
description are obtained from the FLUENT user manual [FLUENT 2010].

The conservation of mass equation, or continuity equation, has the following general
form:

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{v}) = S_m
\]

Eq. 2-1

For incompressible ($\rho \sim \text{constant}$) and steady state flow as in the present case, the first
term on the left in Eq. (2.1) can be neglected. Likewise, the source term on the right side
of the equation, $S_m$, can be neglected since no mass is being added from a dispersed phase
to another continuous phase. Thus, in the present case, this equation reduces to:

\[
\nabla \cdot (\mathbf{v}) = \frac{\partial v_x}{\partial x} + \frac{\partial v_y}{\partial y} + \frac{\partial v_z}{\partial z} = 0
\]

Eq. 2-2

Where $v_x$, $v_y$, and $v_z$ are the velocity components of the fluid in $x$, $y$ and $z$, directions,
respectively.

The conservation of momentum equation, in an inertial (non-accelerating) reference
frame, is described as follows:

\[
\frac{\partial}{\partial t} (\rho \mathbf{v}) + \nabla \cdot (\rho \mathbf{v} \mathbf{v}) = -\nabla p + \nabla \cdot (\mathbf{t}) \cdot \rho \mathbf{g}
\]

Eq. 2-3

Where $p$ is the static pressure, $\rho \mathbf{g}$ is the gravitational body force and $\mathbf{t}$ is the stress tensor
defined as:

\[
\mathbf{t} = \mu \left[ (\nabla \mathbf{v} + \nabla \mathbf{v}^T) - \frac{2}{3} \nabla \cdot \mathbf{v} \mathbf{I} \right]
\]

Eq. 2-4

Where $\mu$ is the dynamic viscosity and $\mathbf{I}$ is the unit tensors.
2.3.2 Turbulence Model

The fluid flow regime inside the piping of ground source heat pump systems is turbulent in nature due to the high Reynolds number, $Re$, which is defined as

$$Re = \frac{\rho v (D_H)}{\mu}$$

where $D_H$ is the hydraulic diameter which is equal to the physical pipe diameter for a circular pipe, $v$ is the mean velocity of the fluid, $\rho$ is the density of the fluid and $\mu$ is the dynamic viscosity of the fluid.

The Reynolds number for a given fluid flow is used to classify the flow regime. The flow is considered to be laminar if the Reynolds Number is lower than 2,000 and turbulent if it is higher than 5,000. Between these two limits, the flow regime would be considered in a transitional phase [White 2002]. In the base case for which we are initially applying the geometrical and operational parameters from an experimental VGLHE testing apparatus [Esen et al 2009], the Reynolds Number was calculated to be 11,270. This calculation considered a density of 1017 kg/m$^3$, an inlet velocity of 0.591 m/s, a hydraulic diameter (pipe inner diameter) of 30 mm and a dynamic viscosity of 0.0016 kg/m·s.

FLUENT offers three main categories for turbulent flow simulation methods. These are:

- DNS – Direct Numerical Simulation
- SRS – Scale Resolving Simulations
- RANS – Reynolds Averaged Navier-Stokes Simulations

The first two categories, DNS and SRS, are usually best fit for unsteady flow conditions and complex flow patterns. The RANS turbulence models are the only meddling approach for steady stage simulation of turbulent flows and they provide the required accuracy.
Within the RANS category, FLUENT further provides an array of models for the steady state calculations. These models are generally divided between “one-equation” and “two-equations” models. Of these steady state models, the realizable $k$-$\epsilon$ model is selected. The $k$-$\epsilon$ model is considered to be one of the simplest “complete models” of turbulence where the solution of two separate transport equations allows the turbulent velocity and length scales to be independently determined. Due to its robustness and reasonable accuracy, it has become commonly used in industrial flow and heat transfer simulations since it was proposed by Launder et al. (1972).

The realizable $k$-$\epsilon$ turbulence model has also been utilized for very similar VGLHE simulations by other researchers [Cvetkovski 2014, Congedo et al. 2014] with accurate results. It is a recent development from the standard $k$-$\epsilon$ model which is a semi-empirical model based on the solution of two separate transport equations for the turbulence kinetic energy ($k$) and the energy dissipation rate ($\epsilon$). The realizable model differs from the standard one in the way it formulates the turbulent viscosity ($\mu_t$) and it also has a new transport equation for the energy dissipation rate ($\epsilon$).

The main two transport equations used to obtain the turbulence kinetic energy ($k$) and its rate of dissipation ($\epsilon$) are:

$$
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_j} (\rho k u_j) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \epsilon - Y_M \quad \text{Eq. 2-6}
$$

$$
\frac{\partial}{\partial t} (\rho \epsilon) + \frac{\partial}{\partial x_j} (\rho \epsilon u_j) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial \epsilon}{\partial x_j} \right] + \rho C_1 \frac{S \epsilon}{k} - \rho C_2 \frac{e^2}{k + \sqrt{\nu \epsilon}} + C_1 \frac{\epsilon}{k} C_3 \epsilon G_b \quad \text{Eq. 2-7}
$$

Where,

$$
C_1 = \max \left[ 0.43, \frac{\eta}{\eta + 5} \right], \quad \eta = \frac{5}{\epsilon}, \quad S = \sqrt{2S_{ij}S_{ij}} \quad \text{Eq. 2-8}
$$

In the above transport equations:

- $G_k$ is the generation of turbulence kinetic energy due to the mean velocity gradient
- $G_b$ is the generation of turbulence kinetic energy due to buoyancy
• $Y_M$ is the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate.

• $C_{1\epsilon}$ and $C_2$ are constants that are experimentally determined and have the following values: $C_{1\epsilon}=1.44$, $C_2=1.9$

• $S$ is the modulus of the mean rate-of-strain tensor

• $\sigma_k$ and $\sigma_\epsilon$ are the turbulent Prandtl numbers for $k$ and $\epsilon$, respectively. They are experimentally determined and have the following values: $\sigma_k=1.0$, $\sigma_\epsilon=1.2$

• $\mu_t$ is the turbulent viscosity, defined as $\mu_t = \rho C_{\mu} k^2 \overline{\epsilon}$ where $C_{\mu} = \frac{1}{A_0 + \frac{A_s k U^*}{\overline{\epsilon}}}$

where $U^* = \sqrt{S_{ij}S_{ij} + \overline{\Omega_{ij}} \overline{\Omega_{ij}}}$

and $\overline{\Omega_{ij}} = \Omega_{ij} - 2\epsilon_{ijk} \omega_k$

$\Omega_{ij} = \overline{\Omega_{ij}} - \epsilon_{ijk} \omega_k$

$\overline{\Omega_{ij}}$ is the mean rate-of-rotation tensor in a rotating reference frame with angular velocity $\omega_k$.

$A_0$ and $A_s$ are constants: $A_0 = 4.04$ and $A_s = \sqrt{6} \cos \phi$

where

$\phi = \frac{1}{3} \cos^{-1}(\sqrt{6} W)$, $W = \frac{S_{ij} S_{jk} S_{kl}}{S^3}$, $\overline{S} = \sqrt{S_{ij} S_{ij}}$, $S_{ij} = \frac{1}{2} \left( \frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right)$

2.3.3 Energy and Convective Heat Transfer Modelling

Due to the presence of heat transfer in our simulations, the turbulent model selected in FLUENT also models the turbulent heat transport using the notion of Reynolds’ similarity to the turbulent momentum transfer. The energy equation used is given as [FLUENT 2010]:

$$\frac{\partial}{\partial t}(\rho E) + \frac{\partial}{\partial x_i} [u_i (\rho E + p)] = \frac{\partial}{\partial x_j} \left( k_{eff} \frac{\partial T}{\partial x_j} + u_i (\tau_{ij})_{eff} \right) + S_h \quad \text{Eq. 2-9}$$

Where,
• $E$ is the total energy transported,
• $S_h$ is the defined volumetric heat source,
• $k_{eff}$ is the effective thermal conductivity and
• $(\tau_{ij})_{eff}$ is the deviatoric stress tensor defined as:

$$(\tau_{ij})_{eff} = \mu_{eff} \left( \frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right) - \frac{2}{3} \mu_{eff} \frac{\partial u_k}{\partial x_k} \delta_{ij}$$

Eq. 2-10

• $k_{eff}$, the effective thermal conductivity in the above equation, is defined as

$$k_{eff} = k + \frac{c_p \mu_t}{Pr_t}$$

Eq. 2-11

Where, $k$ represents the thermal conductivity of the material and $Pr_t$ is Prandtl number.

### 2.3.4 Energy Modelling in Solid Regions

As the present case consists of two solid components, pipe and grout, the proposed model will need to solve the energy equations in these solid regions. The energy transport equation used by FLUENT is the following:

$$\frac{\partial}{\partial t} (\rho h) + \nabla \cdot (\vec{v} \rho h) = \nabla \cdot (k \nabla T) + S_h$$

Eq. 2-12

Where $\rho = \text{Density}$

$h = \text{Sensible Enthalpy}, \int_{T_{ref}}^{T} c_p dT$

$k = \text{conductivity}$

$T = \text{Temperature}$

$S_h = \text{Volumetric Heat Source}$

Since the flow being simulated is steady and incompressible ($\rho \sim \text{constant}$), the first term on the left in Eq. (2.9) can be neglected. The second term on the left, which represents the convective energy transfer due to rotational or translational motion of the solids, can also
be neglected since the pipe and grout solid components are motionless. This equation simulates the conductive heat transfer in the solid regions.

2.3.5 Conjugate Heat Transfer

As the present modelling domain contains a fluid/solid interface involving heat transfer, the solver will need to simulate it as a conjugate heat transfer problem. The FLUENT solver computes the conduction of heat through solids, coupled with convective heat transfer in the fluid. Generally, the Navier Stokes and the convective energy equations in the fluid region (Heat Transfer Fluid) are solved first followed by the conductive heat transfer equations in the solid regions (pipe and grout). Figure 2.2 outlines the locations of each solid and fluid region as well as the fluid/solid interface wall.

![Diagram of simulated components and type of heat transfer]

**Figure 2.2: Section view of simulated components and type of heat transfer**

The pipe/fluid wall is considered a “two-sided-wall” since it forms the interface between the two regions. It is there where a “shadow” zone is created so that each side of the wall is a distinct wall zone. Then a “Coupled Thermal Condition” is applied and the solver calculates the heat transfer directly from the solution in the adjacent cells.
At the boundary condition definition stage, the walls are selected to be “coupled” and any resistance parameter set for one side of the wall will automatically be assigned to its shadow wall zone.

2.3.6 Boundary Conditions

A critical component of any CFD simulation is the setting up of appropriate thermal and physical parameters on the physical boundaries of the model. FLUENT provides a range of boundary condition types and this section describes those that were selected for this model.

2.3.6.1 Inlet and outlet boundary conditions

A velocity inlet boundary condition was selected at the pipe inlet. The magnitude and direction of velocity, fluid temperature, hydraulic diameter and turbulent intensity are the variables required to fully define the inlet velocity boundary condition. All of these variables are provided by the model’s physical properties except for the turbulent intensity, $TI$, which is related to the Reynolds number, $Re$, in the following manner [FLUENT 2010]:

$$TI = 0.16 \, Re^{-1/8}$$

The boundary type at the pipe outlet in the model was selected to be a pressure outlet boundary condition with turbulent intensity factor and hydraulic diameter variables identical to those considered for the inlet boundary conditions.

2.3.6.2 Walls

The fluid adjacent walls were set as stationary with no-slip conditions. The external grout walls were set with a constant temperature boundary condition representing the temperature of the soil.

Also, it must be noted that all of the simulations in this research were considered to be conjugate heat transfer problems due to the interface between the fluid region (heat transfer fluid) and the solid region (piping and grout bodies). This was achieved by
selecting the “Coupled” option for the two-sided walls in the thermal conditions setting within the software.

### 2.3.7 Solution Methods and Initialization

Based on previous research and simulations conducted by other researches on similar types of flow problems, the FLUENT solver selected for this model was the pressure-based solver, which is intended for low-speed incompressible flows [FLUENT 2010]. The segregated pressure-based scheme SIMPLE, Semi-Implicit Method for Pressure-Linked Equations, was used which utilizes the relationship between velocity and pressure corrections to impose the mass conservation (continuity) and find the pressure field. The steps used in the algorithm are illustrated in Figure 2.3. The convergence criterion was set at $10^{-3}$ for the continuity equation, $10^{-4}$ for the axial velocity, $10^{-4}$ for $k$ and $\varepsilon$, and $10^{-5}$ for the energy (temperature) equation. The criterion for each parameter has been selected and evaluated based on the steady behaviour of the resultant residual plot.

The gradients were computed according to the “Least Squares Cell Based” method. The spatial discretization scheme used to evaluate pressure, momentum, turbulent kinetic energy, turbulent dissipation rate, and energy quantities was the “Second Order Upwind” scheme. This scheme produces higher order accuracy through a Taylor series expansion [FLUENT 2010]. Standard initialization was used with the steady state flow and temperature values provided by the physical model.
The full scale 3D geometry was created using the ANSYS Design Modeller software. The 2D top and section view sketches with dimensions are shown in Figure 2.4. Due to the symmetrical behaviour of the heat transfer process about the centre vertical plane of the geometry, only one half of the full geometry was created as shown in Figure 2.5 and Figure 2.6. This gives the advantage of reducing time and resources required for computation. Note that due to significantly longer length of the domain compared to its cross-section, the upper and lower regions of the computations domain are shown in Figures 2.3 and 2.4, respectively. As for the boundary conditions for these symmetry walls, a “symmetry” boundary type was selected, which assumes a zero flux for all of the simulated quantities across them.

Figure 2.3: Algorithm illustrating steps of a Pressure-Based solution
Figure 2.4: Top view and side view of U-Tube piping geometry domain used in simulation.

Figure 2.5: Top isometric view of the vertical ground loop 3D geometry
Three segments were created in this geometry, which were the heat transfer fluid, the pipe and the surrounding grout. Initially, the two-dimensional top profile for each body was created, and then an extrusion step for each 2D profile was created by using the sweep feature along a path line down the geometry then back up again representing the U-tube pipe path. This created a three-dimensional sweepable geometry which made the meshing procedure easier later on. The bodies were set as solid for the grout and the pipe and as fluid for the inner heat transfer fluid.

The geometrical dimensions of the 3D model were based on an experimental VGLHE installation in Elazig, Turkey reported by Esen et al. [2009]. Table 2-1 shows these geometrical dimensions.
Table 2-1: Geometrical dimensions of 3D model

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Borehole Depth</td>
<td>30 m</td>
</tr>
<tr>
<td>Borehole Diameter</td>
<td>150 mm</td>
</tr>
<tr>
<td>Pipe Inner diameter</td>
<td>30 mm</td>
</tr>
<tr>
<td>Pipe Outer diameter</td>
<td>40 mm</td>
</tr>
<tr>
<td>Pipe Spacing (centre to centre)</td>
<td>60 mm</td>
</tr>
</tbody>
</table>

2.5 Mesh Generation and Dependency Test

The mesh was generated by using the ANSYS Mesher Software. The volume mesh for the heat transfer fluid and the surrounding pipe were created by sweeping a 2D Quad/Tri surface mesh along the path line representing the U-tube pipe path. The volume mesh for the grouting region has also been meshed using Tri-Quad elements. It must be noted that since the temperature gradient itself undergoes a gradual change from large to small, further away from the U-tube pipes; the size of the meshing cell also undergoes a similar course of change [Li et al 2009]. This is reflected by having the mesh size of the grout set to be larger, relative to the mesh size of the heat transfer fluid. This resulted in less computation time and required less computer memory. Once the mesh was created, the main boundaries were labelled accordingly (inlet, outlet and grout wall). A generated mesh for the model is shown in Figure 2.7.

It must be noted that one of the limitations encountered during the mesh generation stage was the creation of the interface mesh cells between the grout and the pipe. This limitation was mainly due to the limited computational capacity available during the meshing process. As seen in Figure 2.7, the cell size of the grout mesh at the interface with the pipe increases abruptly and is considerably larger than the average size of the mesh cell in the rest of the grout volume. This in turn reduced the number of interface nodes between the grout and the pipe. Nevertheless, this has a negligible effect on the simulation for two main reasons:

1. The majority of the thermal resistance inside of the borehole is attributed mainly to the conductive heat transfer mode within the grout body itself. As will be presented in later sections and based on the thermal circuit resistance model, over 80% of the total borehole thermal resistance was computed to be from the grout.
2- With vertical ground loop heat exchangers, the temperature gradient along the borehole’s horizontal cross section is more significant than the temperature gradient along the vertical depth of the borehole. Hence, the mesh resolution is less critical along the depth.

Figure 2.7: Top isometric view of generated mesh

In order to ensure that the proposed simulations were not dependent on the mesh size and in order to avoid unnecessary additional computation time and power, a mesh dependency test was performed. Four meshes of different sizes ranging from very coarse to very fine were created for the same borehole geometry. Also, the fluid mesh has been inflated near the pipe wall in order to resolve the viscous sublayer in that region and produce acceptable $y^*$ values [FLUENT 2010]. Table 2-2 shows different mesh sizes that were used for the mesh dependency test and their properties (See Appendix A for screenshots of different meshes). It must be noted that although Table 2-2 shows that mesh #3 and mesh #4 have the same element dimensions, the nodes count for mesh #4 is higher than that for mesh #3. This is because the fluid inflation layers near the wall have been refined further, while maintaining the general element size for the fluid and grout, thus increasing the total mesh node count.
Table 2-2: Properties of different mesh sizes used for mesh dependency test

<table>
<thead>
<tr>
<th>Mesh name</th>
<th>Fluid element size (mm)</th>
<th>Grout element size (mm)</th>
<th>Sweep element length (mm)</th>
<th>Mesh nodes count</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mesh #1</td>
<td>6</td>
<td>15</td>
<td>15</td>
<td>610,302</td>
</tr>
<tr>
<td>Mesh #2</td>
<td>3</td>
<td>7.5</td>
<td>15</td>
<td>6,140,106</td>
</tr>
<tr>
<td>Mesh #3</td>
<td>2</td>
<td>5</td>
<td>10</td>
<td>8,077,792</td>
</tr>
<tr>
<td>Mesh #4</td>
<td>2</td>
<td>5</td>
<td>10</td>
<td>8,630,390</td>
</tr>
</tbody>
</table>

The operating conditions that were applied as boundary conditions for the simulation runs in the dependency test are presented in Table 2-3.

Table 2-3: Boundary conditions used in mesh dependency test

<table>
<thead>
<tr>
<th></th>
<th>Inlet velocity</th>
<th>Inlet temperature</th>
<th>Grout wall temperature (ground temperature)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0.591 m/s</td>
<td>8.15°C</td>
<td>16°C</td>
</tr>
</tbody>
</table>

As for the materials used in the mesh dependency test simulation runs; the heat transfer fluid was a water-propylene glycol mixture (25% propylene glycol by weight), the pipe material was high density polyethylene (HDPE) and the grout material was bentonite. Table 2-4 shows the thermophysical properties of these materials.

Table 2-4: Thermophysical properties of material used in simulation

<table>
<thead>
<tr>
<th>Material</th>
<th>Density (kg/m³)</th>
<th>Thermal conductivity (W/mK)</th>
<th>Specific heat (J/kgK)</th>
</tr>
</thead>
<tbody>
<tr>
<td>HT Fluid</td>
<td>1017</td>
<td>0.475</td>
<td>3947</td>
</tr>
<tr>
<td>HDPE Piping</td>
<td>960</td>
<td>0.4</td>
<td>2170</td>
</tr>
<tr>
<td>Grout (Bentonite)</td>
<td>1540</td>
<td>1.7</td>
<td>2030</td>
</tr>
</tbody>
</table>

The comparison parameter considered in the mesh dependency test was the outlet temperature of the heat transfer fluid which was obtained as the average temperature of the outlet cross-section. Table 2-5 shows the simulated outlet temperature for each mesh size.
Table 2-5: Outlet temperature for different mesh sizes

<table>
<thead>
<tr>
<th>Mesh name</th>
<th>Inlet Temp (°C)</th>
<th>Outlet Temp (°C)</th>
<th>Temperature Difference (°C)</th>
<th>% Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mesh #1</td>
<td>8.15</td>
<td>9.136</td>
<td>0.986</td>
<td>-</td>
</tr>
<tr>
<td>Mesh #2</td>
<td>8.15</td>
<td>9.171</td>
<td>1.021</td>
<td>3.52</td>
</tr>
<tr>
<td>Mesh #3</td>
<td>8.15</td>
<td>9.199</td>
<td>1.049</td>
<td>2.65</td>
</tr>
<tr>
<td>Mesh #4</td>
<td>8.15</td>
<td>9.194</td>
<td>1.044</td>
<td>0.44</td>
</tr>
</tbody>
</table>

As Table 2-5 shows, slight variations in the outlet temperature are observed by varying the mesh size as evidenced by the small percentage difference. The percentage difference ranged from 3.52% at the most coarse mesh to 0.44% at the finest mesh. Since the percentage difference between mesh #4 and mesh #3 is less than 2%, the results were hence considered to be independent of the mesh size and mesh #4 was therefore selected for the planned simulations. Furthermore, the value of wall \( y^+ \) for mesh #4 has been examined and a plot of the \( y^+ \) along the top 0.5m depth of the pipe is shown in Figure 2.8. As can be seen in the figure, the \( y^+ \) at the wall-adjacent cell is in the order of \( y^+ =1 \), which is well within the acceptable range to accurately model the near-wall region.

Figure 2.8: Plot of wall \( y^+ \) along depth of pipe (top 0.5m)
The entrance length, where flow is fully developed, has also been investigated for mesh #4 and compared with the theoretical estimate. The equation used for calculating the entrance length ($L_e$) of the developed turbulent flow in a circular pipe is:

$$\frac{L_e}{d} = 4.4 \frac{Re^{1/6}}{d}$$

Eq. 2-13

Where $Re$ is Reynolds number and $d$ is the pipe diameter [FLUENT 2010]. Based on the pipe dimensions and flow velocity the calculated entrance length was found to be 0.6m which is in an acceptable agreement of the simulated 0.56 m shown in Figure 2.9.

![Entrance length for fluid flow for mesh #4](image)

Figure 2.9: Entrance length for fluid flow for mesh #4

2.6 Model Validation

In order to ensure that the developed model correctly simulates the physical process, it must be validated against experimental results. Although there were some published
experimental papers about vertical ground loop heat exchanger installations in Canada, most were either missing key parameters that would be required for numerical simulations (such as ground/grout temperature or geometric dimensions) or they considered very deep boreholes (deeper than 100 metres), which were too large domains to be simulated given the computation power and resources available. However, a published experimental work for a VGLHE installation in Elazig, Turkey by Esen et al. [2009] and further communications with the author provided all of the parameters required for the numerical simulations. The borehole size considered in their study was also in the typical range of ground-source heat pump installations.

2.6.1 Description of the experimental data

The experiment consisted of an in-situ ground thermal test installation where an above ground pump with heater circulated the heat transfer fluid through the borehole piping while measuring the fluid temperatures at the inlet and outlet of the borehole. The set up was tested in both summer and winter months. Figure 2.10 shows a schematic for the setup used. The dimensions of the borehole and piping are shown in Table 2-6 and the specifications of the material used and their thermophysical properties are listed in Table 2-7.

![Schematic of an in-situ test setup](image)

**Figure 2.10: Schematic of an in-situ test setup (adapted from Esen et al [2009])**
Table 2-6: Physical dimensions of experimental borehole

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Borehole Depth</td>
<td>30 m</td>
</tr>
<tr>
<td>Borehole Diameter</td>
<td>150 mm</td>
</tr>
<tr>
<td>Pipe Inner diameter</td>
<td>30 mm</td>
</tr>
<tr>
<td>Pipe Outer diameter</td>
<td>40 mm</td>
</tr>
<tr>
<td>Pipe Spacing (centre to centre)</td>
<td>60 mm</td>
</tr>
</tbody>
</table>

Table 2-7: Thermophysical properties of materials used in the experiment

<table>
<thead>
<tr>
<th></th>
<th>Density (kg/m³)</th>
<th>Thermal conductivity (W/mK)</th>
<th>Specific heat (J/kgK)</th>
</tr>
</thead>
<tbody>
<tr>
<td>HT Fluid</td>
<td>1017</td>
<td>0.475</td>
<td>3947</td>
</tr>
<tr>
<td>HDPE Piping</td>
<td>960</td>
<td>0.4</td>
<td>2170</td>
</tr>
<tr>
<td>Grout (Bentonite)</td>
<td>1540</td>
<td>1.7</td>
<td>2030</td>
</tr>
</tbody>
</table>

For validation purposes, the seven cases reported in the experimental study were numerically simulated. Three cases were in the winter (heating) season and four cases were in the summer (cooling) season. All boundary conditions were the same for all cases except for the inlet fluid temperature. The average ground temperature and the inlet fluid velocity as provided in the experimental study were 16°C and 0.591 m/s, respectively. The fluid inlet temperatures of the selected experimental cases for simulations are shown in Table 2-8.
### Table 2-8: Operating conditions and properties for thermal response test

<table>
<thead>
<tr>
<th>Case</th>
<th>Grout wall temperature (ground temperature)</th>
<th>Inlet Temperature (°C)</th>
<th>Inlet velocity</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case 1 (Winter)</td>
<td></td>
<td>6.00</td>
<td></td>
</tr>
<tr>
<td>Case 2 (Winter)</td>
<td></td>
<td>4.77</td>
<td></td>
</tr>
<tr>
<td>Case 3 (Winter)</td>
<td></td>
<td>8.00</td>
<td></td>
</tr>
<tr>
<td>Case 1 (Summer)</td>
<td>16 °C</td>
<td>39.33</td>
<td>0.591 m/s</td>
</tr>
<tr>
<td>Case 2 (Summer)</td>
<td></td>
<td>41.50</td>
<td></td>
</tr>
<tr>
<td>Case 3 (Summer)</td>
<td></td>
<td>44.00</td>
<td></td>
</tr>
<tr>
<td>Case 4 (Summer)</td>
<td></td>
<td>34.80</td>
<td></td>
</tr>
</tbody>
</table>

#### 2.6.2 Simulation and results

In order to simulate these cases in FLUENT, an identical 3D geometry was built using the ANSYS Design Modeller software as outlined in section 2.4, and mesh #4, containing more than 8,500,000 nodes, was selected from the mesh dependency test. The materials and boundary conditions cited in section 2.6.1 were applied to the model. The parameter of comparison used for validation was the fluid outlet temperature. Appendix B shows the experimental tabulated inlet and outlet temperatures as received from the author (Esen et al 2009) and Figure 2.11 shows the simulation and experimental results for all seven cases for comparison.
Figure 2.11: Comparison between experimentally measured and numerically simulated fluid outlet temperatures

The simulated outlet temperatures for all cases showed a trend similar to that of the experimental ones. As predicted, they both have shown an increase in the fluid temperature inside the ground loop in winter cases where heat is being absorbed from ground and a decrease in the fluid temperature in the summer cases where heat is being rejected into ground. The comparisons show relatively higher percentage error for the winter cases compared to that for the summer cases. For winter, the percentage error ranged from 10.5% to 18.5% with an average of 16.7%, while for summer, the percentage error ranged from 0.61% to 4.14% with an average of 1.88%. The high percentage error in winter could possibly be attributed to experimental uncertainties in the experimental apparatus itself. These include the placement position of the temperature sensor within the inlet and outlet flow streams; where a temperature sensor placed too close to the pipe wall would give higher readings than those given if it was placed near the centre of the pipe.
Other sources of error would include the impact of the bond strength between the soil and grout. In the given model, the exterior wall of the grout was set to a temperature that emulates the temperature of the surrounding soil and that temperature was uniform along the full surface of grout wall. This supposition may have contributed to the error as the interface between the grout and the soil in the validation experiment may not have been as uniform. Causes of this grout/soil interface issue in the experiment are related to the issues discussed earlier in section 2.2.

Also, the recording interval of the outlet temperature is very critical when recording steady-state measurements. It must be noted that the temperature gradient between the inlet and the grout wall (ground temperature) is higher in summer than in winter; that temperature difference reached 25°C in summer, while for winter it only reached 11°C. Hence, it is concluded that the present model correctly simulates the thermo-fluid process inside the ground loop.
3 Parametric Study

As discussed earlier in the Introduction Chapter, the performance of a Ground Source Heat Pump (GSHP) system is evaluated by the coefficient of performance of the system (COP). The COP is the ratio of the desired heat transfer rate to the electrical power consumed by the heat pump system. This heat transfer rate is in part dependent on the heat exchanging efficiency between the ground loop piping and the soil. As mentioned earlier, the focus of the present work is on this heat exchange between the vertical ground loop heat exchanger (VGLHE) and the surrounding soil. There are many geometrical, thermophysical, and operational factors that could highly affect this heat exchange process and hence, the COP of the whole system. In the following, a detailed parametric study is conducted to investigate the effect of these parameters on the heat transfer process in the VGLHE. These factors, if understood well, can optimize the performance of GSHP systems.

The parameters used in this study were divided into three categories; geometrical, thermo-physical and operational, which cover all of the important parameters related to the heat transfer process in a VGLHE. Furthermore, as discussed in Chapter 1, a heat pump operates in both summer and winter seasons with reverse operating modes. The heat transfer process is expected to be similar in both modes except for the direction of heat transfer. Thus, to save computational time, only one operating mode (i.e. winter mode) was considered in the present parametric study. The validated numerical model described in Chapter 2 was used to simulate the process for all parametric cases.

3.1 Geometrical Parameters

The geometry of the piping inside the VGLHE is an important factor that could impact the overall effectiveness of the heat exchange process in the ground loop. Hence, three different loop configurations were chosen and simulated in this section, which are:

1- U-Tube Piping Configuration
2- Concentric Piping Configuration

3- Spiral (Helical) Piping configuration

Detailed simulation and analyses of each of these configurations are presented in the following sub-sections.

3.1.1 U-Tube Piping Configuration

The U-tube piping configuration is the one most commonly used in vertical ground loop installations. This configuration consists of two vertically parallel straight pipes joined at the bottom of the borehole by an 180° elbow fitting. As described earlier in Chapter 2, this configuration consists of the heat transfer fluid, piping and the grout, all surrounded by the soil. Figure 3.1 shows the top and side views of this configuration along with the dimensions. The two sections of the pipe carrying the heat transfer fluid downwards and upwards will be referred to herein as the down-flow and up-flow pipes, respectively.

![Figure 3.1: Top view and side view of U-Tube piping geometry domain used in simulation.](image-url)
The materials selected for the simulations in the present case were the ones commonly used in the field. The piping was selected as High Density Polyethylene (HDPE), the grout was a bentonite mixture and the heat transfer fluid was a propylene glycol mixture with a dynamic viscosity of 0.0016 \( \frac{kg}{m \cdot s} \). Table 3-1 shows the thermophysical properties of the materials used.

Table 3-1: Thermophysical properties of material used in simulation

<table>
<thead>
<tr>
<th>Material</th>
<th>Density (kg/m(^3))</th>
<th>Thermal conductivity (W/mK)</th>
<th>Specific heat (J/kgK)</th>
</tr>
</thead>
<tbody>
<tr>
<td>HT Fluid</td>
<td>1017</td>
<td>0.475</td>
<td>3947</td>
</tr>
<tr>
<td>HDPE Piping</td>
<td>960</td>
<td>0.4</td>
<td>2170</td>
</tr>
<tr>
<td>Grout (Bentonite)</td>
<td>1540</td>
<td>1.7</td>
<td>2030</td>
</tr>
</tbody>
</table>

Constant inlet fluid velocity and constant inlet fluid temperature were applied as boundary conditions. The typical values of the velocity and fluid temperature at the inlet of a VGLHE for winter operations range from 0.129 m/s to 1.029 m/s and from -8°C to 18°C, respectively. In the present study, we considered the inlet values to be the same as those used in the experimental site (Esen et al. 2009) for the winter operating mode, unless the inlet conditions were changed as a part of the parametric study. These operating conditions are presented in Table 3-2, which are within the range of typical inlet values in the field. A constant temperature was applied at the grout wall along the depth to simulate the temperature of the soil.

Table 3-2: Boundary conditions used [Adopted from Esen et al. 2009]

<table>
<thead>
<tr>
<th></th>
<th>Inlet velocity</th>
<th>Inlet temperature</th>
<th>Grout wall temperature (soil temperature)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0.591 m/s</td>
<td>8.15°C</td>
<td>16°C</td>
</tr>
</tbody>
</table>

3.1.1.1 Results

Figure 3.2 shows the variation in the fluid temperature (averaged over the pipe cross-section) along the pipe at 5 m increments. The result shows that the bulk fluid
temperature varied linearly with the pipe length. The outlet fluid temperature was found to be 9.19 °C which is an increase of 1.04 °C from the inlet fluid temperature of 8.15 °C. It is observed that the bulk fluid temperature increased by approximately 0.017 °C per metre of the pipe length. The Reynolds number was calculated using equation (Eq. 2-5) and for the present case, it was 11,270.

Figure 3.2: Temperature of fluid along the pipe

In the steady operating mode in winter, when the fluid enters the pipe, its temperature is lower than the surrounding soil. This temperature difference between the soil and the fluid serves as the temperature potential to drive the heat transfer from the soil to the fluid via the grout and pipe wall. The heat is continuously transferred to the fluid throughout its passage through the pipe. The simulations in the given domain provided detailed information about various parameters such as velocity, temperature, etc. which can be used to obtain a deeper insight into the heat transfer process in the VGLHE. In the following, these parameters are presented in various forms to highlight the key aspects of the heat transfer process.

The temperature distribution within the borehole horizontal plane is presented in Figure 3.3 at different depths. As observed, the perimeter of the grout wall shows the highest temperature representing the constant soil temperature. The temperature distribution
shows a typical gradual decrease in the inward radial direction as expected. The plots also show that the fluid temperature continues to increase from the inlet to the outlet, as expected. Consequently, the difference between the fluid temperatures in the up-flow and down-flow pipes is largest near the top and gradually decreased towards the bottom. An interesting observation is the temperature distribution in the middle region of the borehole cross-sectional plane. The plots show an almost constant temperature of the grout in this region, which takes an ellipsoidal shape encapsulating the up-flow and down-flow pipes. The grout temperature in this region is relatively low compared to that in the other sections of the borehole. These results are important as they indicate that the heat transfer rate from the grout to the fluid is not uniform along the pipe circumference due to the variation in the grout temperature surrounding the pipes. That is, the heat transfer rate is higher through the pipe surfaces facing the outer periphery (hereinafter referred to as the peripheral side) than the pipe surfaces facing the centre of the borehole (hereinafter referred to as the inner side). In other words, a fraction of the pipe surface does not play an active role in the heat transfer from the soil in this configuration. To quantify this difference, the area-weighted average heat flux through the peripheral side and the inner side are computed for both down-flow and up-flow sections of the pipe. These results are presented in Table 3-3. The results show that the heat flux through the peripheral pipe surface is approximately 1.5 times of the heat flux through the inner pipe surface for both up-flow and down-flow pipe sections. These results also show that the heat flux in the up-flow section is less than the heat flux in the down-flow section. This difference in heat flux between the up-flow and down-flow pipes is due to the temperature difference between the grout and the fluid in the up-flow pipe being smaller than that of the fluid in the down-flow pipe. The results in Figure 3.3 and Table 3-3 indicate that the distance between the up-flow and down-flow pipes has an impact on the overall heat transfer rate to the fluid. Increasing the distance between the pipes is expected to increase the grout temperature in the middle region but the overall borehole diameter has to increase, which adds to the cost of the VGLHE.
Figure 3.3: Temperature distribution in the borehole horizontal plane at depths of (a) 0 m, (b) 10 m, (c) 20 m, (d) 30 m. The colour bar represents the temperature in degree Celsius.
Table 3-3: Area-weighted average surface heat flux values for different pipe sections

<table>
<thead>
<tr>
<th>Area Weighted Average Surface Heat Flux (W/m²)</th>
<th>Periphery facing pipe surface (Down-flow pipe)</th>
<th>Centre facing pipe surface (Down-flow pipe)</th>
<th>Periphery facing pipe surface (Up-flow pipe)</th>
<th>Centre facing pipe surface (Up-flow pipe)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Periphery facing pipe surface (Down-flow pipe)</td>
<td>379</td>
<td>261.4</td>
<td>350.8</td>
<td>228.9</td>
</tr>
</tbody>
</table>

Due to the color axis used in Figure 3.3 which captured the entire range of grout and fluid temperatures, the temperature variation within the fluid was not clearly represented. To obtain a better insight into the fluid temperature distribution in a cross-sectional plane, the temperature distribution within the fluid cross-sectional plane at different depths is shown in Figure 3.4. The temperature contours of the fluid are more visible and noticeable at shallow depths than near the bottom of the borehole. The contours of fluid in the up-flow and down-flow pipes near the bottom are similar because of their close proximity to each other along the length of the pipe. It was also observed that the temperature distribution within the fluid horizontal plane is asymmetric with larger temperature variations in the fluid side closer to the borehole edge than the fluid side closer to the other pipe. This is due to the difference in the heat flux at the pipe wall as discussed earlier.
Figure 3.4: Temperature distribution in the fluid horizontal plane at depths of (a) 0 m, (b) 10 m, (c) 20 m, (d) 30 m. The colour bar represents the temperature in degree Celsius.
The fluid temperature profiles in the mid vertical plane of the pipe loop at different depths are shown in Figure 3.5. The temperature profiles of the fluid in the regions that are very close to the pipe wall were all much higher than those in the rest of the pipe cross section. This is due to the presence of the thermal boundary layer adjacent to the pipe wall. It is also observed that once the flow is fully developed and the entrance length has been achieved, the temperature profiles beyond that point were all similar in pattern with just a vertical shift to reflect the increase in temperature.

Data normalization was performed in order to fit the data within unity (1) using the following equation:

\[ T_{i,0 \text{ to } 1} = \frac{T_i - T_{\text{inlet}}}{T_{\text{outlet}} - T_{\text{inlet}}} \]

Where:

\( T_i \) is each temperature data point in the mid vertical plane (°C)
\( T_i \) is the fluid inlet temperature (°C)
\( T_{\text{outlet}} \) is the fluid outlet temperature (°C)
$T_{i,0 \text{ to } 1}$ is the data temperature point between 0 and 1

The fluid temperature data point, $T_i$, is taken at distance, $X$, along the diameter, $D$, of the cross section of the fluid at a specific depth. Figure 3.6 below shows a sketch identifying these parameters.

![Figure 3.6: Sketch indicating the location of temperature data points](image)

The normalized temperature profiles at different depths are shown in Figure 3.7. The normalized temperature has a pattern that is very similar to the temperature profiles plot (Figure 3.5) with just a vertical shift to reflect the increase in temperature.

![Figure 3.7: Normalized temperature profile of fluid at different depths of down-flow pipe](image)
The average convective heat transfer coefficient, \( h \), for the fluid along the full length of the U-tube pipe was computed as a part of simulation and was found to be 1779 W/m\(^2\)K. This value is consistent with typical heat transfer coefficient values for turbulent flows in similar conditions [e.g. see Schwencke 2013 and Young 2004]. The magnitudes of individual thermal resistances were computed to quantify the contribution of each thermal resistance to the heat flow and to determine which one is the controlling thermal resistance. As mentioned earlier in Chapter 1, there are two fluid nodes in the given case and also the conductive thermal resistance in the grout is 2D, which makes it difficult to accurately quantify each thermal resistance. As seen earlier in the results, the difference in fluid temperatures between the two pipes is very small as compared to that in the grout, hence, to simplify this analysis, a single fluid node is assumed. Based on computed heat transfer coefficient and other physical and thermal properties of the model the individual thermal resistance of each component was calculated using the equations outlined in section 1.2.4. The values of these thermal resistances are presented in Table 3-4.

**Table 3-4: Calculated thermal resistances of individual VGLHE components**

<table>
<thead>
<tr>
<th>Component</th>
<th>Calculated Thermal Resistance (mK/W)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Heat Transfer Fluid</td>
<td>0.006</td>
</tr>
<tr>
<td>HDPE Pipe</td>
<td>0.115</td>
</tr>
<tr>
<td>Grout</td>
<td>0.50</td>
</tr>
</tbody>
</table>

This shows that the largest thermal resistance in the borehole was contributed by the grout which consisted of almost 80% of the total borehole thermal resistance. The thermal resistance of the piping (HDPE material in this case) consisted of about 18% of the total borehole thermal resistance. The thermal resistance in the fluid is approximately 1% of the overall borehole thermal resistance, which is negligible in comparison to other
thermal resistances. The lower convective thermal resistance in the fluid could be attributed to the turbulent nature of the flow.

The analysis of the velocity data in the mid-vertical plane of the straight sections of the pipe loop (not shown here) indicates the classical parabolic velocity behaviour. The fluid velocity in the 180°-bend showed some variations which influenced the corresponding temperature distribution. To illustrate this behaviour, the temperature distribution and the velocity contours in the 180° pipe bend at the bottom of the borehole are shown in Figure 3.8. The results show that as the fluid approached the bend, it accelerated near the inner pipe wall and then decelerated immediately after negotiating the bend on the same wall side, which is likely due to the flow separation. This causes an acceleration in the fluid on the opposite side of the wall. This velocity variation influenced the corresponding fluid temperature, shown in Figure 3.8 a). It can be seen that the fluid temperature locally decreased in the region of accelerating fluid and locally increased in the region of decelerating fluid. Although, such temperature variation has a relatively insignificant effect on the overall heat transfer rate in the pipe loop.

![Figure 3.8: (a) Temperature and (b) velocity contours of the bottom bend of pipe](image)

Figure 3.8: (a) Temperature and (b) velocity contours of the bottom bend of pipe
As Figure 3.8 shows, the pipe mesh discretization through the 180°-bend is not very fine as the straight sections of mesh cells could be seen. This has a negligible effect on the overall heat transfer process as the length of the 180°-bend is very small relative the fully length of the pipe (less than 0.3% of the overall pipe length).

3.1.2 Concentric Piping Configuration

The second common geometry of piping configuration used in vertical ground loop heat exchangers is the concentric piping geometry. This configuration consists of two vertically concentric pipes: inner and outer, and the grout, all surrounded by the soil. The heat transfer fluid flows down through the inner pipe and returns upwards through the annulus region between the two pipes. The diameters of the inner and outer pipes were selected to be 0.035 m and 0.054 m, respectively, such that the flow areas inside the inner pipe and inside the annulus region are approximately equal. Figure 3.9 shows top and side views of this configuration along with the dimensions.

Similar to the U-tube piping configuration, the materials selected for the simulations of the concentric case were the ones commonly used in the field. Both pipes were selected as HDPE, the grout was a bentonite mixture and the heat transfer fluid was a propylene glycol mixture. Table 3.1 shows the thermophysical properties of the materials used. Constant inlet fluid velocity and constant inlet fluid temperature were applied as boundary conditions at the inner pipe inlet. Similar to the U-tube piping, the inlet values were considered to be the same as those used in the experimental study (Esen et al. 2009) for the winter operating mode, unless the inlet conditions were changed as a part of the parametric study. These operating conditions are presented in Table 3-5, which are within the range of typical inlet values in the field. A constant temperature was applied at the grout wall along the depth to simulate the soil temperature.
Figure 3.9: Top view and side view of concentric piping geometry domain used in simulation.

Table 3-5: Boundary conditions used [Adopted from Esen et al. 2009]

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet velocity</td>
<td>0.591 m/s</td>
</tr>
<tr>
<td>Inlet temperature</td>
<td>8.15°C</td>
</tr>
<tr>
<td>Grout wall temperature (soil temperature)</td>
<td>16°C</td>
</tr>
</tbody>
</table>

3.1.2.1 Results

Figure 3.10 shows the variation in the fluid temperature (averaged over the flow area cross-section) along the concentric pipes at 5 m increments. The bulk fluid temperature showed two linear regions of temperature increase. The first semi-flat region represents the down-flow fluid temperature increase in the inner pipe while the second steeper region represents the up-flow fluid temperature increase in the annulus region. The outlet fluid temperature was found to be 8.75°C which is an increase of 0.75°C from the inlet fluid temperature of 8.00°C. It is observed that the bulk fluid temperature increased
by approximately 0.003°C per metre of pipe length in the inner pipe and by approximately 0.023°C per metre of pipe length in the annulus region. The Reynolds numbers within the inner pipe and the annulus region were calculated to be 6,573 and 5,300 respectively (the hydraulic diameter of the annulus was region was calculated by subtracting the outer diameter of the inner pipe from the inner diameter of the outer pipe).

![Temperature of fluid along length of loop](image)

**Figure 3.10: Area-averaged temperature of fluid along the pipe**

The temperature distribution within the borehole horizontal plane is presented in Figure 3.11 at different depths. As observed, the temperature distribution is symmetrical along the vertical mid-plane of the borehole and the perimeter of the grout wall shows the highest temperature representing the constant soil temperature. The temperature distribution also shows a typical gradual decrease in the inward direction as expected. The difference between the fluid temperatures in the inner pipe and the annulus region is largest near the top of the borehole and gradually decreases towards the bottom. The plots also show that the temperature of the down-flow fluid inside the inner pipe does not change significantly through different depths. However, this temperature change is more noticeable and more significant for the up-flow fluid inside the annulus region at different depths.
As shown earlier, the fluid flowing through the annulus region has a higher rate of temperature increase than that of the fluid flowing through the inner pipe. This is due to the higher temperature gradient that exists between the annular fluid and the adjacent grout resulting in a higher heat transfer rate. To quantify this difference, the total heat transfer rates were computed for each of these two piping sections and were found to be 0.17 kW for the inner pipe (down-flow) and 1.53 kW for the annulus (up-flow) region. The results show that the heat transfer rate through the outer pipe surface into the annulus region is approximately 9 times of the heat flux through the inner pipe surface. These two distinct total heat transfer rates correspond to the two linear regions shown in Figure 3.10 (fluid temperature vs. length of piping loop) indicating that the heat transfer rate through the outer pipe is predominant. Note that the heat gain by the fluid in the inner pipe is in fact a heat loss from the annulus fluid.

Due to the color axis used in Figure 3.11 which captured the entire range of grout, piping and fluid temperatures, the temperature variations within the fluid were not clearly represented. To obtain a better insight into the fluid temperature variations, the temperature distribution within the fluid cross-sectional plane at different depths is shown in Figure 3.12. As the figure shows, the fluid temperature variations are more visible and noticeable at shallow depths than near the bottom of the borehole. This is due to relatively larger temperature difference between the up-flowing and down-flowing fluids. The temperature contours of the up-flow fluid in the inner pipe and the down-flow fluid in the annulus region near the bottom are similar because of their close proximity to each other along the length of the pipe. The plots also show that the temperature distribution is axisymmetric in the cross-sectional planes as expected.
Figure 3.11: Temperature distribution in the borehole horizontal plane at depths of (a) 0 m, (b) 10 m, (c) 20 m, (d) 29 m. The colour bar represents the temperature in degree Celsius.
Figure 3.12: Temperature distribution in the fluid horizontal plane at depths of (a) 0 m, (b) 10 m, (c) 20 m, (d) 29 m. The colour bar represents the temperature in degree Celsius.
As the fluid movement changes in the bottom section of the concentric pipe configuration, it is important to see if it has a significant influence on the flow and temperature fields. For this purpose, the fluid velocity and temperature contours in the bottom of the borehole are shown in Figure 3.13. The results show that the fluid velocity magnitude is high when it exits the inner pipe, which then decreases as the flow is diverged towards the annulus region. The peak velocity magnitude in the annulus section is relatively low compared to that in the inner pipe. This is likely due to the smaller gap width of the annulus region which causes higher flow losses. The plot also shows that the fluid downward velocity magnitude rapidly decreased to zero in the bottom section. This indicates that the fluid remains relatively stagnant in the bottom section resulting in a local rise in pressure. This higher fluid pressure in the bottom causes an early divergence of fluid towards the annulus region soon after it exits from the inner pipe. The corresponding temperature contours show an almost uniform fluid temperature in the bottom section, which is likely due to higher fluid velocities except near the bottom edge of the bore hole where the fluid temperatures are slightly higher. As per velocity plot, this

Figure 3.13: (a) Temperature and (b) velocity contours in the bottom section of the concentric pipe
corresponds to the region of almost stagnant fluid. Thus, this temperature rise is likely due to conduction from the grout.

3.1.3 Spiral (Helical) Piping Configuration

The third geometry of piping configuration used in this study is the spiral “helical” piping geometry. This geometry consists of spiral and straight piping sections embedded in the grout and all are surrounded by soil. The heat transfer fluid enters through the inlet of the spiral piping section and returns upwards through the outlet of the straight piping section located in the centre of the borehole. For the simulation of this piping configuration to be parametrically comparable to the other two other configurations, the total piping volume was maintained by keeping the total length of the piping at 60 metres. However, due to the pitch nature of spiral shapes the depth of this borehole was only 12 metres which is almost one third of the depth of the other two geometries. The diameter of the piping in this configuration is the same as the one used in the U-Tube configuration. Figure 3.14 shows a 3D view of the upper part of the simulated spiral piping configuration model. Same materials as for the previous configurations were selected for the spiral piping simulation. That is, HDPE as the pipe material, the grout was a bentonite mixture and the heat transfer fluid was a propylene glycol mixture. Table 3-1 shows the thermophysical properties of the materials used.

Constant inlet fluid velocity and constant inlet fluid temperature were applied as boundary conditions at the pipe inlet. Similar to the U-tube and the concentric piping configurations, we considered the inlet values to be the same as those used in the experimental study for the winter operating mode (Esen et al. 2009), unless the inlet conditions were changed as a part of the parametric study. These operating conditions are presented in Table 3-6. A constant temperature was applied at the grout wall along the depth to simulate the soil temperature.
Table 3-6: Boundary conditions used [Adopted from Esen et al. 2009]

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet velocity</td>
<td>0.591 m/s</td>
</tr>
<tr>
<td>Inlet temperature</td>
<td>8.15°C</td>
</tr>
<tr>
<td>Grout wall temperature</td>
<td>16°C</td>
</tr>
</tbody>
</table>

3.1.3.1 Results

Figure 3.15 shows the average temperature of the heat transfer fluid in the spiral piping geometry at different locations along the pipe. The bulk fluid temperature showed two linear regions of temperature increase. The first steeper region (0 m to 50 m) represents the temperature of the down-flow heat transfer fluid inside of the spiral piping section while the second semi-flat region (50m to 60m) represents the temperature of the up-flow heat transfer fluid inside of the straight vertical piping section. The outlet fluid
temperature was found to be 8.95°C which is an increase of 0.95°C from the inlet fluid temperature of 8.00°C. It is observed that the bulk fluid temperature increased by approximately 0.0183°C per metre of pipe length in the spiral piping section and by approximately 0.0053°C per metre of pipe length in the straight piping section. The Reynolds number for this case was calculated to be 11,270.

![Temperature of fluid along length of loop](image)

**Figure 3.15: Average temperature of fluid along the length of loop**

The temperature distribution within the borehole horizontal plane at six different depths is shown in Figure 3.16. The temperature distribution is asymmetrical due to the asymmetric geometry of the spiral pipe when taken in a horizontal cross-sectional plane. In Figure 3.16, the area to the right with cooler temperatures (darker blue) represents the heat transfer fluid within the spiral section of the piping while the area in the centre of the borehole represents the temperature of heat transfer fluid in the straight section of the piping. The temperature of the down-flow heat transfer fluid in the spiral piping section increases more significantly than that of the straight piping section. This is more clearly visible in Figure 3.17 where the temperature distribution is shown only for the heat transfer fluid capturing its range of increase. Evidently, the colour of the temperature contours in the spiral section of the piping change entirely while that of the straight section of the piping remains essentially unchanged.
Figure 3.16: Temperature distribution in the borehole horizontal plane at depths of (a) 0 m, (b) 2 m, (c) 4 m, (d) 6 m, (e) 8 m, (f) 10 m. The colour bar represents the temperature in degree Celsius.
Figure 3.17: Temperature distribution in the fluid horizontal plane at depths of (a) 0 m, (b) 2 m, (c) 4 m, (d) 6 m, (e) 8 m, (f) 10 m. The colour bar represents the temperature in degree Celsius.
The heat transfer fluid flowing through the spiral piping section has a higher rate of temperature rise than that of the straight piping section due to the proximity of the spiral section to the grout edge and due to the higher temperature gradient which exists between them. Furthermore, the swirl generated within the fluid due to the pipe geometry also contributed to the increase in the heat transfer rate. To quantify this difference, the total heat flux was computed for each of these two piping sections and was found to be 0.09 kW for the straight piping section (up-flow) and 1.49 kW for the spiral piping section (down-flow) region. The results show that the heat flux through the spiral piping surface is approximately 17 times higher than the heat flux through the straight piping surface.

3.1.4 Comparison and Discussion

In this section the three different piping configurations are compared to each other in order to evaluate the influence of the geometry change on the overall performance of the VGLHE.

A summary of the simulation results of the three piping configurations is shown in Table 3-7 and Figure 3.18 below. It must be noted that all three configurations were simulated maintaining the same inlet temperature, same volume of fluid with the same piping length of 60 metres. The only difference was the total depth for the spiral piping configuration due to its geometry which was much shorter, 10.22 metres, than the other two piping configurations which were 30 metres deep each. This, however, is the practical difference in the real applications for these configurations. As the table shows, U-tube configuration yielded the highest heat transfer rate, which is about 60% and 25% higher than concentric and spiral configurations, respectively. Figure 3.16 provides a comparison of the average fluid temperature along the pipe length for the three configurations. The results show that although the fluid temperature and consequently the heat transfer rate in the spiral section is higher than that in the U-tube, the very low heat transfer in the straight vertical section of the spiral configuration affected its overall performance. The concentric pipe showed low temperatures throughout the length compared to the other two configurations. However, the rise in temperature per unit length in the annular section of the concentric pipe configuration was 35% and 26% higher than U-tube and spiral configurations, respectively. The overall low outlet temperature in the concentric pipe configuration is
due to the low temperature rise per unit length in the middle pipe, which was almost 13% of the temperature rise per unit length in the annulus section.

**Table 3-7: Summary of simulation results for three different piping configurations**

<table>
<thead>
<tr>
<th>Piping Configuration</th>
<th>Inlet Temp (°C)</th>
<th>Outlet Temp (°C)</th>
<th>Delta T (°C)</th>
<th>Heat Transferred to Fluid (kW)</th>
<th>Vertical Depth of Piping Bottom (m)</th>
<th>Diameter of Borehole (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>U-Tube</td>
<td>8.00</td>
<td>9.19</td>
<td>1.19</td>
<td>1.98</td>
<td>30.00</td>
<td>0.15</td>
</tr>
<tr>
<td>Concentric</td>
<td>8.00</td>
<td>8.75</td>
<td>0.75</td>
<td>1.25</td>
<td>30.00</td>
<td>0.15</td>
</tr>
<tr>
<td>Spiral</td>
<td>8.00</td>
<td>8.95</td>
<td>0.95</td>
<td>1.58</td>
<td>10.22</td>
<td>0.41</td>
</tr>
</tbody>
</table>

**Figure 3.18: Temperature of fluid at different positions of the pipe for three different piping configurations**
Although the concentric piping configuration had the least heat transfer and fluid outlet temperature, it would have the least cost of installation of all three piping configurations if it is integrated in the piling foundation of a structure (Energy Piles). With energy piles, the cost of drilling is already anticipated under the foundation work, and the concentric piping is mainly required to be inserted inside of the drilled piles. Furthermore, in the conventional boreholes, HDPE is used as the piping material which has the thermal conductivity of 0.4 W/m.K. Such low thermal conductivity increases the thermal resistance of the circuit and hence reduces the rate of heat transfer. Energy Piles are made of steel with the thermal conductivity of 50 W/m.K. If a concentric pipe configuration is carved into the Energy Pile and hence eliminating the need of HDPE, it is expected to increase the heat conductance and hence the heat transfer rate.

3.2 Parametric analysis of Thermophysical Properties and Operational Parameters

In this section, the impact of thermo-physical and operational parameters on the heat transfer process in the VGLHE is investigated. For a systematic parametric analysis, only one specific parameter is varied at a time in each simulation while maintaining all other parameters unchanged.

Furthermore, the piping configuration (geometrical parameter) in these simulations needs to be unchanged in order to properly evaluate the results of each parameter variation. The piping configuration selected for these simulations is the concentric piping. Although this piping configuration have shown to have the least total heat transfer among all three piping configurations simulated, due to its potential for integration in energy piles compared to the other two configurations and also from constructability and installation cost points of views, concentric piping configuration is chosen.

3.2.1 Thermal Conductivity of the pipe

In the initial simulation for the concentric piping configuration in section 3.1.2, both inner and outer pipes were set as high density polyethylene (HDPE). In this section the piping material is changed to a highly thermal conductive material such as copper. Following different arrangements are considered:
1- Case 1: Inner and outer pipes’ material is HDPE (already presented in section 3.1.2)

2- Case 2: Inner and outer pipes’ material is Copper

3- Case 3: Inner pipe’s material is HDPE and outer pipe’s material is Copper

4- Case 4: Inner pipe’s material is Copper and outer pipe’s material is HDPE

The temperature distribution patterns in all four simulated cases showed similar trends, which have been presented earlier under the concentric configuration section 3.1.2. Therefore, it is not presented here since they do not provide any new insight into the process.

Table 3-8 summarizes the results for all cases and Figure 3.19 shows the average fluid temperature along the full length of the piping for the four simulated cases.

Table 3-8: Summary of simulation results for different pipe materials in concentric pipe configuration.

<table>
<thead>
<tr>
<th>Case #</th>
<th>Inner Pipe Material</th>
<th>Outer Pipe Material</th>
<th>Inlet Temp (°C)</th>
<th>Outlet Temp (°C)</th>
<th>Delta T (°C)</th>
<th>Heat Transferred (kW)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>HDPE</td>
<td>HDPE</td>
<td>8.00</td>
<td>8.75</td>
<td>0.75</td>
<td>1.70</td>
</tr>
<tr>
<td>2</td>
<td>Copper</td>
<td>Copper</td>
<td>8.00</td>
<td>9.00</td>
<td>1.00</td>
<td>2.27</td>
</tr>
<tr>
<td>3</td>
<td>HDPE</td>
<td>Copper</td>
<td>8.00</td>
<td>9.05</td>
<td>1.05</td>
<td>2.38</td>
</tr>
<tr>
<td>4</td>
<td>Copper</td>
<td>HDPE</td>
<td>8.00</td>
<td>8.73</td>
<td>0.73</td>
<td>1.64</td>
</tr>
</tbody>
</table>
As the results show, Case 3 yielded the highest outlet temperature and hence, the highest total heat transferred to the fluid. This configuration considered HDPE inner piping and copper outer piping. This is an interesting observation since it performed better than case #2, where the material of both the inner and the outer pipes was copper which has a much higher thermal conductivity than HDPE piping (387.6 w/m.K vs 0.4 w/m.K).

This could be explained by the fact that when the inner pipe is made out of a highly thermal conductive material, such as copper in this case, the returning up-flow fluid inside the annular region loses its thermal energy to the down-flow fluid inside the inner pipe compared to the case when the inner pipe is made out of low conductivity material, such as HDPE in this case, and hence will have a lower outlet temperature. This is confirmed by fluid’s temperature behaviour for case #2 in Figure 3.19 where the temperature of the fluid peeks at 9.11°C at the 50 m mark before reaching the outlet, which is higher than in other case (including their outlet temperatures), then it drops to 9.00°C at the outlet.
Comparison of results also shows that the material of outer pipe has much higher impact on the overall heat transfer rate than the inner pipe. That is, the use of higher conductivity material for the outer pipe resulted in higher heat transfer rate compared to the higher conductivity material for the inner pipe. This result is expected since the wall conductive resistance of the outer pipe has the primary influence on the overall heat gain into the flow loop. Although the high conductivity of the inner pipe increases the heat transfer from annulus region to the inner region, this heat transfer contributes to the increase in the fluid temperature in the loop. As Figure 3.19 shows, when the inner pipe is made of high conductivity material, most of the heat gain in the fluid occurs within the inner pipe (see Case #2 and Case #4).

In summary, the concentric loop configuration with highly conductive annular piping material and highly insulative inner pipe material performs the best.

### 3.2.2 Grout Thermal Conductivity

In the initial simulation for the concentric piping configuration in section 3.1.2, the grouting material considered was based on an actual thermally enhanced conductive grouting product (CETCO Geothermal Grout) which had a thermal conductivity of 1.7 W/m-K. In this section the thermal conductivity value of the grouting material is changed to analyze its impact on the overall heat transfer in a concentric configured borehole. Three cases are conducted as described below:

1- Case 1: Grouting thermal conductivity is 1.7 w/m-K (already presented in section 3.1.2)
2- Case 2: Grouting thermal conductivity is halved to 0.85 W/m-K
3- Case 3: Grouting thermal conductivity is doubled to 3.40 W/m-K

In case 2, the grout thermal conductivity value of 0.85 W/m-K is based on high solids bentonite grout (Allan et al. 2000). In case 3, the thermal conductivity value of 3.4 W/m-K is based on an actual thermally enhanced grouting material that was studied by Lee et al. (2010) where its thermal conductance was increased by mixing in a graphite powder additive.
The temperature distribution contours produced in all three simulated cases have trends similar to that shown previously under the concentric pipe analysis in section 3.1.2 and hence are not presented here.

Table 3-9 shows the summary of results for all three cases and Figure 3.20 shows the average fluid temperature along the full length of the piping for the three simulated cases.

**Table 3-9: Summary of simulation results for concentric piping configurations with changing grouting thermal conductivity values**

<table>
<thead>
<tr>
<th>Case #</th>
<th>Grout Thermal Conductivity (w/m-K)</th>
<th>Inlet Temp (°C)</th>
<th>Outlet Temp (°C)</th>
<th>Delta T (°C)</th>
<th>Heat Transferred (kW)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.70</td>
<td>8.00</td>
<td>9.05</td>
<td>1.05</td>
<td>2.38</td>
</tr>
<tr>
<td>2</td>
<td>0.85</td>
<td>8.00</td>
<td>8.60</td>
<td>0.60</td>
<td>1.36</td>
</tr>
<tr>
<td>3</td>
<td>3.40</td>
<td>8.00</td>
<td>9.89</td>
<td>1.89</td>
<td>4.27</td>
</tr>
</tbody>
</table>
As expected, the grout with the highest thermal conductivity (Case 3) yielded the highest heat transfer from the surrounding grout into the fluid and which resulted in the highest fluid outlet temperature. The fluid temperature profiles in Figure 3.20 showed a linear trend with different slopes as expected. However, the rate of heat transfer and consequently the fluid temperature did not increase linearly with the grout thermal conductivity. That is, the increase in the grout thermal conductivity by 100% increased the heat transfer rate by 75%. This is due to the reason that the thermal circuit correspond to the heat transfer from ground to the fluid comprised of several resistances hence, the variation of one thermal resistance in the circuit does not influence the overall heat transfer coefficient linearly.

3.2.3 Flow Rate of Heat Transfer Fluid

In this section, the impact of volume flow rate of the heat transfer fluid entering the concentric VGLHE is investigated. Three simulation cases are conducted as described below:
1- Case 1: Fluid volume flow rate of heat transfer fluid is $2.82 \times 10^{-4} \text{ m}^3/\text{s}$ or 4.47 GPM (presented in section 3.1.2)

2- Case 2: Fluid volume flow rate of heat transfer fluid is halved to $1.41 \times 10^{-4} \text{ m}^3/\text{s}$ or 2.24 GPM

3- Case 3: Fluid volume flow rate of heat transfer fluid is doubled to $5.64 \times 10^{-4} \text{ m}^3/\text{s}$ or 8.94 GPM

Table 3-10 summarizes the results for all three cases and Figure 3.21 shows the average fluid temperature along the full length of the piping for the three simulated cases

**Table 3-10: Summary of simulation results for concentric piping configurations with varying volume flow rate of heat transfer fluid**

<table>
<thead>
<tr>
<th>Case #</th>
<th>Fluid Volume Flow Rate (m$^3$/s)</th>
<th>Reynolds Number</th>
<th>Inlet Temp (°C)</th>
<th>Outlet Temp (°C)</th>
<th>Delta T (°C)</th>
<th>Heat Transferred (kW)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>$2.82 \times 10^{-4}$</td>
<td>11,270</td>
<td>8.00</td>
<td>9.05</td>
<td>1.05</td>
<td>2.38</td>
</tr>
<tr>
<td>2</td>
<td>$1.41 \times 10^{-4}$</td>
<td>5,635</td>
<td>8.00</td>
<td>9.88</td>
<td>1.88</td>
<td>2.13</td>
</tr>
<tr>
<td>3</td>
<td>$5.64 \times 10^{-4}$</td>
<td>22,540</td>
<td>8.00</td>
<td>8.56</td>
<td>0.56</td>
<td>2.55</td>
</tr>
</tbody>
</table>
The rate of heat transfer rate into the heat transfer fluid is calculated using the following equation as:

$$ Q = \dot{m} \ c \ \Delta T $$

Eq. 3-1

Where,

- $Q$ is the Heat Transfer Rate into the system, kW
- $\dot{m}$ is the mass flow rate of the fluid, kg/s
- $c$ is the specific heat capacity of the fluid, J/kg*K
- $\Delta T$ is the temperature difference between the inlet and outlet of the fluid, °C

The results show a decrease in the fluid outlet temperature with an increase in the fluid volume flow rate, as expected, while the overall heat transfer rate was slightly increased with an increase in the volume flow rate. This could be due to the increase in the
Reynolds number and hence the forced convection. The results also indicate that an increase in the fluid volume flow rate by 100% resulted in a decrease in the fluid temperature by 80%.

3.2.4 Concentric VGLHE in an “Energy Micro-Pile”

In this section, simulation results are presented for a case depicting the integration of the concentric piping configuration in an “Energy Micro-Pile” installation. As mentioned in Chapter 1, energy piles are thermal foundation piles which serve both as a foundation to transfer the structural load of the building to the ground and as a ground energy heat exchanger for GSHP systems. In comparison to conventionally drilled VGLHE boreholes, this technology can provide significant installation cost savings since no additional drilling will be required for GSHP ground loop.

The foundation pile type simulated in this section is a hollow bar micro-pile. Micro-piles in general are deep foundation piles that typically have small diameters (less than 300 mm), they are drilled and grouted-in-place once placed into the ground. As shown in Figure 3.22, a typical hollow bar micro-pile consists mainly of a sacrificial drill bit at the bottom, a threaded steel hollow bar (the pile body itself), couplers to extend the overall length of the micro-pile and the grouting around the steel pile to provide the grout/ground bond.

Once the hole is drilled, the grouting product is then pumped inside the hollow bar and after passing through one or more nozzles in the sacrificial drill bit, it diffuses and fills up the gap between the pile and the ground. Although the function of the grouting around the pile is to bind the outside surface of the threaded steel hollow bar to the adjacent soil, the typical end product usually has the grouting inside of the hollow bar steel pile.
Figure 3.22: Typical hollow bar micro-pile (Drbe et al. 2013)

In order for this micro-pile system to be simulated as an energy micro-pile, the following modifications and assumptions are considered:

1- The inside of the hollow bar will be considered not to be filled with grout (i.e. inside is to be flushed so the grouting is only around the outside of the micro-pile)
2- A smaller inner HDPE pipe is considered to be inserted and concentrically fitted inside of the hollow bar micro-pile to act as the conduit for carrying the down-flow heat transfer fluid.

3- The annular region created between the inner HDPE pipe and the outer threaded steel hollow-bar is the space where the up-flow heat transfer fluid flows and ultimately leaves the micro-pile at the top.

Figure 3.23 shows the top and the side view of this configuration along with the dimensions used in the simulation.

![Diagram of energy micro-pile geometry domain](image)

**Figure 3.23: Top view and side view of energy micro-pile geometry domain used in simulation.**

The threaded hollow bar dimensions and thicknesses were based on an actual product manufactured by Ischebeck (CTS/TITAN IBO Micropile). Table 3-11 shows the dimensions and parameters used in the simulation. The heat transfer fluid (propylene glycol mixture), the high-density polyethylene (HDPE) piping and the grouting material are the exact same ones used in the initial three simulations and hence are not shown in this table (see Table 3-11 for these properties).
Table 3-11: Parameters and dimensions of micro-pile used in simulation

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value/Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Vertical depth of Micro-Pile</td>
<td>60 m</td>
</tr>
<tr>
<td>Micro-Pile Type/Model</td>
<td>Titan IBO Micropile – 103/78</td>
</tr>
<tr>
<td>Threaded Hollow-Bar O.D.</td>
<td>103.00 mm</td>
</tr>
<tr>
<td>Threaded Hollow-Bar I.D.</td>
<td>78.00 mm</td>
</tr>
<tr>
<td>Inner HDPE Pipe O.D.</td>
<td>60.50 mm</td>
</tr>
<tr>
<td>Inner HDPE Pipe I.D.</td>
<td>49.30 mm</td>
</tr>
<tr>
<td>Drill Bit Size</td>
<td>112 mm</td>
</tr>
<tr>
<td>Borehole Diameter</td>
<td>168 mm</td>
</tr>
<tr>
<td>Threaded Steel Bar Density</td>
<td>7850 kg/m³</td>
</tr>
<tr>
<td>Threaded Steel Bar Specific Heat Capacity</td>
<td>490 J/kg K</td>
</tr>
<tr>
<td>Threaded Steel Bar Thermal Conductivity</td>
<td>50 W/mK</td>
</tr>
</tbody>
</table>

To further achieve a more realistic condition, the actual ground temperature profile was considered through the utilisation of a User Defined Function (UDF). Note that in the earlier cases, the ground temperature was considered to be constant over the entire depth. Available ground temperature data in South Western Ontario for depths of 60 m could only be obtained for the Goderich area from a previous Geophysical experimental study by Markle (2011). Figure 3.24 shows the ground temperature profile as a function of depth and as can be seen, the ground temperature becomes almost constant at 9°C below 11 m of depth.
This variable ground temperature profile was applied as the temperature boundary condition to the grouting wall in the simulation domain through a UDF. The temperature profile in the top 11 m of the ground was mathematically approximated through polynomial regression while the ground temperature below 11 m was set to a constant 9°C. The polynomial regression model for the ground temperature in the first 11 m was found to be:

\[ T_g = 273.15 + 0.0009 Y^3 - 0.0384 Y^2 - 1.2842 Y + 0.725 \]
Where,

\[ T_g = \text{Ground Temperature (°K)} \]
\[ Y = \text{Ground Depth (m)} \]

**Appendix C** shows the code written for the UDF to apply the variable ground temperature as the temperature boundary condition for the grouting wall in simulation.

Constant fluid velocity and constant fluid temperature were applied as the inlet boundary conditions for the entering fluid. In the present simulation, the inlet fluid velocity was the same as the one used in the experimental site (Esen et al. 2009) for the winter operating mode. The fluid inlet temperature, however, was selected to be -6°C based on the performance data of commercially available GSHP systems (ClimateMaster – Tranquility 30 Model 064). These heat pump systems with the use of antifreeze mixtures as the heat transfer fluid (as in the present case) can have inlet fluid temperatures as low as -8.8°C and still stay operational.

### 3.2.4.1 Results

Figure 3.25 shows the variation in the average fluid temperature at 10 metre increments along the full length of the piping inside the energy micro-pile. The piping length was referenced from 0 m at the top of the inner pipe is considered at 0 m to 120 m at the top of the annular region (i.e. for a 60 m deep concentric geometry, the full length of the piping is 120 m). The behaviour of the temperature fluid is similar to the concentric simulation conducted in section 3.1.2 where it shows two distinct regions of temperature increase. The first region, between 0 m and 60 m, represents the down-flow fluid temperature increase in the inner HDPE pipe while the second steeper region, between 60 m and 120 m, represents the up-flow fluid temperature increase in the annular region created between the threaded steel bar and the inner HDPE pipe. The temperature rise trend in both regions is semi-linear and they deviate from the linear behaviour observed previously in the concentric simulation section 3.1.2 due to the varying ground temperature in the top 11 m (from 0 m to 11 m in for the fluid in the inner region and from 109 m to 120 m for the fluid in the annular region). The Reynolds numbers within the inner pipe and the annulus region were calculated to be 15,519 and 15,965,
respectively (the hydraulic diameter of the annulus was region was calculated by subtracting the outer diameter of the inner HDPE pipe from the inner diameter of the outer threaded hollow-bar).

![Graph: Temperature of fluid along length of loop](image)

**Figure 3.25: Temperature of fluid along full length of pipe**

Table 3-12 shows the summary of the results for the energy micro-pile fitted with concentric piping. The total change in the temperature of the heat transfer fluid is 6.3°C and corresponding total heat gained by the fluid is 12.0 kW. The temperature increase and the heat transferred to the fluid inside the annular region were found to be 5.08°C and 9.68 kW, respectively, which is approximately 4.2 times more than the temperature increase and heat transferred to the inner HDPE pipe.
Table 3-12: Simulation results of energy micro-pile

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value (Unit)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet Temperature</td>
<td>-6.0°C</td>
</tr>
<tr>
<td>Outlet Temperature</td>
<td>0.3°C</td>
</tr>
<tr>
<td>Temperature Change in Fluid</td>
<td>6.3°C</td>
</tr>
<tr>
<td>Total Heat Transferred</td>
<td>12.0 kW</td>
</tr>
<tr>
<td>Fluid Volume Flow Rate</td>
<td>0.000479 m³/s (7.53 gpm)</td>
</tr>
<tr>
<td>Temperature Increase in inner region</td>
<td>1.22°C</td>
</tr>
<tr>
<td>Heat Transferred in inner region</td>
<td>2.33 kW</td>
</tr>
<tr>
<td>Temperature Increase in annular region</td>
<td>5.08°C</td>
</tr>
<tr>
<td>Heat Transferred in annular region</td>
<td>9.68 kW</td>
</tr>
</tbody>
</table>
Chapter 4 : CONCLUSIONS

4 Conclusions

This research was conducted with the objective of furthering the understanding of the effect of various parameters on the heat transfer process in Vertical Ground Loop Heat Exchangers (VGLHEs) and energy pile systems. These two system components are essential to any Ground Source Heat Pump (GSHP) system and an improved understanding of the influence of these parameters will lead to the design optimization and selection processes of such renewable energy system. It is very critical to properly size the system, as any reduction in the length or depth of the proposed VGLHE would generate substantial cost savings due to the associated high drilling costs. While many studies have been reported in the literature that investigated the heat transfer process in VGLHEs, there is a scarcity of detailed parametric investigation of VGLHE. Hence, the work presented in this thesis is a step forward in understanding the effects of such parameters and the possible improvements that could be attained from the findings. The present research undertook an extensive CFD parametric investigation to compare and evaluate the performance of VGLHEs for different geometrical, thermophysical and operational parameters.

In the first part of this study (Chapter 2), a 3D numerical model was developed using the commercial CFD software FLUENT to simulate the heat exchange process in a typical VGLHE. The model development included mesh dependency test and validation against published experimental results to ensure that the model correctly simulated the underlying physical process.

In the second part of the study (Chapter 3), a detailed parametric study was conducted that considered all important parameters related to the heat transfer process in a VGLHE. These parameters of interest were divided into three categories; geometrical, thermophysical and operational.

For geometrical parameters, the piping loop configuration was varied and three practical configurations were simulated in the CFD model; U-Tube, Concentric and Spiral
The outlet temperature of the heat transfer fluid was considered as the comparison parameter in all simulations. It was obtained as the average temperature of the outlet cross-section. For consistency in comparisons, all three geometries maintained the same volume of the heat transfer fluid and same pipe length (60 metres). The results showed that the U-Tube piping configuration achieved the highest heat transfer rate and thus the highest fluid outlet temperature, followed by the spiral and concentric piping configurations, respectively. At first, this may indicate that the U-Tube configuration is the ideal geometry in field applications; however, there are economic factors that when considered would alter this supposition. The economic factor here would be the drilling cost during installation, which is usually measured per foot of depth to be drilled. This drilling cost for the U-Tube piping configuration would be approximately three times that of the spiral piping configuration since the spiral case had a much shallower total depth than other two piping configurations (10.22 metres vs 30 metres). However, this cost saving advantage must be weighed against the possible reduction of the temperature difference between the soil and the spiral VGLHE for very shallow applications. This is due to the typical temperature profile of the soil and the fact that at shallower depths the temperature of the soil is highly affected by the temperature of the ambient air above (see Figure 1.5). Further investigations would be required to specifically evaluate the depth and establish a criterion at which the spiral configuration becomes more feasible.

Although the concentric configuration has shown an inferior performance when compared to the other two configurations, it becomes the most practical and economically feasible option in certain cases. This configuration would potentially be the best suited for structures where micro-piles with small overall borehole diameters are considered and drilled. As seen in the section 3.2.4, few modifications would be required to fit the existing design of the hollow core micro-pile with an inner HDPE pipe thus converting it to an energy pile. Fitting a hollow core micro-pile with U-tube or spiral piping configuration would be more difficult due to the small borehole diameters of the micro-piles and the limited available space to install two pipes. On the other hand the concentric configuration makes use of the outer hollow-core steel casing by utilizing it as the annular pipe. The decision of using the concentric configuration over the other two
would still need to be further evaluated to confirm that the required design heating/cooling load can be met with planned number of structural micro-piles and their depths.

For thermophysical parameters, the thermal conductivity of the pipe and the grout were varied, simulated and analysed. The piping configuration selected for these simulations was the concentric piping and was unchanged in order to properly evaluate the effect of each parameter variation. The following cases were simulated and evaluated:

1- Piping thermal conductivity variation (Thermal conductivities of copper and HDPE are 387.6 w/m.K and 0.4 w/m.K, respectively):

   a. Case 1: Inner and outer pipes’ material is HDPE
   b. Case 2: Inner and outer pipes’ material is Copper
   c. Case 3: Inner pipe’s material is HDPE and outer pipe’s material is Copper
   d. Case 4: Inner pipe’s material is Copper and outer pipe’s material is HDPE

2- Grout thermal conductivity variation:

   a. Case 1: Grouting thermal conductivity is 1.7 w/m-K
   b. Case 2: Grouting thermal conductivity is halved to 0.85 W/m-K
   c. Case 3: Grouting thermal conductivity is doubled to 3.40 W/m-K

For the piping material, the results showed that in a concentric piping configuration, the combination of a highly conductive annular piping material, like copper, and a highly insulative inner pipe material, like HDPE, resulted in the highest heat transfer between the ground and the fluid.

This interesting observation is useful since it became applicable in the Energy Mirco-Pile simulation in section 3.2.4. The hollow core of the micro-pile (the outer pipe of the concentric configuration) is already constructed from a highly conductive material, steel, and therefore inserting a highly insulative inner pipe, constructed from HDPE, would yield this preferred combination of material selection.
For the grout thermal conductivity, it was found that the grout with the highest thermal conductivity (Case 3), yielded the highest heat transfer from the surrounding grout into the fluid, as expected. This bentonite grout mixture exhibits higher thermal conductivity than the other two types due to the graphite powder additive (Lee et al. 2010). Although all three cases showed a linear trend (with different slopes as seen in Figure 3.20), the rate of heat transfer did not increase linearly with the change in grout thermal conductivity (i.e. doubling the grout thermal conductivity, increased the heat transfer rate by 75%). This finding becomes useful for the designer or the owner when calculating the expected payback period for using a thermally enhanced grouting material versus a typical bentonite grout. The consideration should then be given to the savings generated by the 75% increase in the heat transfer rate versus the cost premium for using the thermally enhanced grout.

For the operational parameters, the volume flow rate of the heat transfer fluid was varied, simulated and the results were analysed. The piping configuration selected for these simulations was the concentric piping and was unchanged in order to properly evaluate the effect of each parameter variation. The following cases were simulated and evaluated:

1- Case 1: Fluid volume flow rate of heat transfer fluid is $2.82 \times 10^{-4}$ m$^3$/s or 4.47 GPM (presented in section 3.1.2)

2- Case 2: Fluid volume flow rate of heat transfer fluid is halved to $1.41 \times 10^{-4}$ m$^3$/s or 2.24 GPM

3- Case 3: Fluid volume flow rate of heat transfer fluid is doubled to $5.64 \times 10^{-4}$ m$^3$/s.

It was found that although the fluid outlet temperature decreased with increasing the flow rate, the overall heat transfer rate into the fluid increased. The highest heat transfer rate was achieved with Case 3.

It should be noted that in the cases simulated, as the flow rate, $\dot{m}$, was increased, the temperature difference between the inlet and outlet, $\Delta T$, decreased at slower rate and hence the total heat transfer, $Q$, increased. This increase, however, was noted to become smaller as the flow rate increased (it increased by 0.25 kW in the first increase and by 0.17 kW in the second increase). This indicates that there is an optimal point at which the
Heat transfer rate would be maximized by increasing the fluid flow rate, however, the long term soil heat retention rate and the required fluid pumping power must be considered in the overall optimization process.

For the “Energy Micro-Pile” simulation, the design of a hollow-core micro-pile was fitted with an inner HDPE piping to utilize it as a concentric VGLHE and the steel core acted as the annular pipe. This is a novel configuration for incorporating a VGLHE inside a structural micro-pile and from the literature research it has not been researched before. The 60 m deep energy micro-pile simulation was able to extract 12 kW of heat from the ground. A typical 60 m deep U-tube VGLHE would extract 4 kW of heat from the ground, however, the typical U-tube piping diameter size is usually smaller than the concentric HDPE piping used in the energy micro-pile simulation (30 mm vs 49.30 mm). More research and experimentation would be required for this type of Energy Micro-pile, nonetheless, these simulation results are promising and demonstrate the potential for this application.

4.1 Recommendations for future work

While conducting the research for this work, there were many areas that were noted to be potential research topics on their own. Other areas that were researched also need to be expanded upon to enhance the understanding of the system. These include:

- Incorporating the group effect and soil-structure interactions into the parametric study. The present work focused on parametric variation effect on VGLHEs individually; however, there are always more than one borehole installed in real-world applications field. Understanding the thermal interactions between multiple VGLHEs would be critical.

- Experimental testing of the micro-pile configuration. Due to the promising nature of the hollow-core micro-pile, detailed experimental testing is vital to demonstrate its practical feasibility in the field.

- Gathering local ground and soil temperature information. During the course of this research, it was very challenging to locate detailed ground temperature
information for geographical areas in Ontario to incorporate in to the User Defined Function (UDF) of the simulation. Finding or producing this information for the future test areas would further improve the accuracy of model predictions and performance evaluations of VGLHEs.

- Performing simulations in the cooling (summer) mode. The simulations in this research focused on the heating (winter) mode of operation which is the more prevalent mode of operation for a northern country like Canada. However, in order to understand the full parametric effect, the summer operation should also be simulated.

- Analysing and evaluating the depth and criterion at which a typical spiral piping configuration becomes more feasible than the U-tube piping configuration. As mentioned in the first part of this conclusion chapter, the spiral geometry required less boreholes depth when compared to the U-Tube geometry, however, caution should be taken since shallower ground depths have soil temperatures that are highly affected by the ambient air temperatures above. The pitch of the spiral geometry should also be considered and its variance effects on the VGLHE’s performance should be evaluated.
References:


Fluent 13 Documentation (2010), Fluent Inc.


Heating, Refrigerating and Air-Conditioning Engineers, Inc., 1791 Tullie Circle, N.E.,
Atlanta, GA, USA.


Lenhard, R., & Malcho, M. (2013). Numerical simulation device for the transport of
dgeothermal heat with forced circulation of media. *Mathematical and Computer
Modelling*, 57(1-2), 111-125. doi:10.1016/j.mcm.2011.06.011

Course for Pile Foundation Design and Construction*, Ipoh.

doi:10.1016/j.rser.2006.10.003

Natural Resources Canada. (2002). Earth Energy, Case Study: Ground-Source Heat
Pumps Produce Savings for Commercial Building, Ottawa. *Natural Resources Canada*.


University Press.

ISSN 0276-1084.

Available from ProQuest Dissertations.

induced mechanical behaviour of energy pile foundations. *Renewable and Sustainable


Appendix A

Screenshots of meshes used for dependency test

Mesh #1 (Nodes count: 610,302)

Mesh #2 (Nodes count: 6,140,106)
Appendix B
Experimental Data

Tabulated inlet and outlet temperatures of water-antifreeze solution for Heating and Cooling Modes of Operation (Highlighted rows were simulated for validation).
Esen et al. (2009)

<table>
<thead>
<tr>
<th>Time</th>
<th>Cooling mode</th>
<th>Time</th>
<th>Heating mode</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>T Inlet (°C)</td>
<td></td>
<td>T Outlet (°C)</td>
</tr>
<tr>
<td>9:00</td>
<td>29.6</td>
<td>9:00</td>
<td>13.6</td>
</tr>
<tr>
<td>9:15</td>
<td>31.3</td>
<td>9:15</td>
<td>13.2</td>
</tr>
<tr>
<td>9:30</td>
<td>33</td>
<td>9:30</td>
<td>12.6</td>
</tr>
<tr>
<td>9:45</td>
<td>33.5</td>
<td>9:45</td>
<td>12.3</td>
</tr>
<tr>
<td>10:00</td>
<td>34</td>
<td>10:00</td>
<td>12</td>
</tr>
<tr>
<td>10:15</td>
<td>34.8</td>
<td>10:15</td>
<td>11.5</td>
</tr>
<tr>
<td>10:30</td>
<td>35.7</td>
<td>10:30</td>
<td>11.1</td>
</tr>
<tr>
<td>10:45</td>
<td>35.85</td>
<td>10:45</td>
<td>10.9</td>
</tr>
<tr>
<td>11:00</td>
<td>36</td>
<td>11:00</td>
<td>10.7</td>
</tr>
<tr>
<td>11:15</td>
<td>36.9</td>
<td>11:15</td>
<td>10.5</td>
</tr>
<tr>
<td>11:30</td>
<td>37.8</td>
<td>11:30</td>
<td>10.5</td>
</tr>
<tr>
<td>11:45</td>
<td>37.9</td>
<td>11:45</td>
<td>10</td>
</tr>
<tr>
<td>12:00</td>
<td>38</td>
<td>12:00</td>
<td>9.6</td>
</tr>
<tr>
<td>12:15</td>
<td>38.4</td>
<td>12:15</td>
<td>9.4</td>
</tr>
<tr>
<td>12:30</td>
<td>38.8</td>
<td>12:30</td>
<td>9.2</td>
</tr>
<tr>
<td>12:45</td>
<td>39</td>
<td>12:45</td>
<td>9.2</td>
</tr>
<tr>
<td>13:00</td>
<td>39.33</td>
<td>13:00</td>
<td>9</td>
</tr>
<tr>
<td>13:15</td>
<td>39.7</td>
<td>13:15</td>
<td>8.9</td>
</tr>
<tr>
<td>13:30</td>
<td>40</td>
<td>13:30</td>
<td>9</td>
</tr>
<tr>
<td>13:45</td>
<td>40.5</td>
<td>13:45</td>
<td>8.9</td>
</tr>
<tr>
<td>14:00</td>
<td>41</td>
<td>14:00</td>
<td>8.8</td>
</tr>
<tr>
<td>14:15</td>
<td>41.5</td>
<td>14:15</td>
<td>8.5</td>
</tr>
<tr>
<td>14:30</td>
<td>42</td>
<td>14:30</td>
<td>8.4</td>
</tr>
<tr>
<td>14:45</td>
<td>42.5</td>
<td>14:45</td>
<td>8.3</td>
</tr>
<tr>
<td>15:00</td>
<td>42.8</td>
<td>15:00</td>
<td>8</td>
</tr>
<tr>
<td>15:15</td>
<td>43</td>
<td>15:15</td>
<td>7.5</td>
</tr>
<tr>
<td>15:30</td>
<td>43.5</td>
<td>15:30</td>
<td>7.2</td>
</tr>
<tr>
<td>15:45</td>
<td>43.85</td>
<td>15:45</td>
<td>7.1</td>
</tr>
<tr>
<td>16:00</td>
<td>44</td>
<td>16:00</td>
<td>7</td>
</tr>
</tbody>
</table>
Appendix C
User Defined Function (UDF)

UDF code written to apply the variable ground temperature as boundary condition for the grout wall in FLUENT simulation (based on ground temperature profile shown in figure below code)

```c
#include "udf.h"
DEFINE_PROFILE(walltemp, thread, nv)
{
    float x[3]; /* this will hold the position vector*/
    float y;
    face_t f;
    begin_f_loop(f, thread)
    {
        F_CENTROID(x,f,thread);
        y = x[1];
        if (y < -11)
            F_PROFILE(f, thread, nv) = 282.15;
        else
            { F_PROFILE(f, thread, nv) = 273.15 + 0.0009*(y*y*y) - 0.0384*(y*y) - 1.2842*(y) + 0.725;
            }
    }
    end_f_loop(f, thread)
}
```

Ground temperature profile which the UDF is approximating (data is for the Goderich region - Markle, 2011)
Curriculum Vitae

EDUCATION:

- Master of Engineering Science Candidate, Mechanical & Materials Engineering
  The University of Western Ontario, London, Ontario
  2015 (Expected – Part time)
- Bachelor of Engineering Science, Mechanical & Materials Engineering
  The University of Western Ontario, London, Ontario
  Class of 2007 – Graduated with Distinction

WORK EXPERIENCE:

Mat 4Site Ltd, Toronto, Ontario
Mechanical Project Manager
January 2014 – Present
Engaged in the Mechanical design and construction stages of various commercial and institutional facilities. (Initial design to construction and delivery)

Western University, London, Ontario
Teaching/Research Assistant – Mechanical Engineering Department
January 2012 – Dec 2013
Duties:
- Field of study in the area of CFD and heat transfer simulations for ground loops.
- Prepared and conducted tutorial sessions for engineering students.
- Prepared and supervised engineering labs and experiments for students.

Crossey Engineering Ltd, Toronto, Ontario
Mechanical EIT
March 2009 – Dec 2011
Duties:
- Design and Tendering Stage:
  - Prepared preliminary schedule and timeline of tasks for review by senior manager.
  - Participated in regular meetings with architects and consultants in the design team.
  - Prepared Design Based Memorandums, assessments and investigation reports.

FSC Architects & Engineers, Whitehorse, Yukon
Mechanical EIT (1 year contract)
April 2008 – March 2009
Involved in the preliminary and detailed design of mechanical systems related to HVAC, plumbing, drainage and service piping for Commercial buildings and facilities
Duties:
• Building load calculations (Heat load, ventilation and water requirements)
• Equipment sizing and selection (HVAC, plumbing and related components)
• Duct and pipe routing, CAD drafting and preparation of mechanical design drawings.
• Project documentation and in-house electronic filing.
• Meetings with clients, contractors and suppliers to ensure progress and on time delivery of projects.

Skills:
• LEED BD+C Accredited Professional.
• Acquainted & familiar with provincial building and plumbing codes, ASHRAE and NFPA.
• Proficient in using: HAP, CHVAC, AutoCAD, ANSYS, Solid Works and CAEPIPE
• Strong academic experience in mechanical engineering, PLCs, Fluid Machinery and CFD.

Awards and Scholarships:
• 2013: Ontario Graduate Scholarship (OGS) amounted to $15,000.
• 2012: Ontario Graduate Scholarship (OGS) amounted to $15,000.
• 2012: Queen Elizabeth II Graduate Scholarship in Science and Technology, $15,000.
• 2007: Graduated with Distinction: Dean’s Honor List in all years.
• 2004: Ian Duerden Memorial Engineering Award ($ 1000)
• 2002: The University of Western Ontario Entrance Scholarship ($ 2000)

Affiliations and Extracurricular:
• Professional Engineers of Ontario (PEO) – EIT Member
• American Society of Heating, Refrigerating and Air-Conditioning Engineers (ASHRAE)
• Canada Green Building Council (CaGBC)
• Ontario Society of Professional Engineers (OSPE) – Associate Resident Member
• Representative of the Mechanical Engineering Department at the Society of Graduate Studies (SOGS) council at Western
• Member of intramural soccer teams at University and social club tournaments
• High interest in amateur photography and photo editing