CHALMERS

Modelling the performance of underground heat exchangers and storage systems

Master of Science Thesis in the Master's Programme Structural Engineering and Building Performance Design

DAVID VAN REENEN

Department of Civil and Environmental Engineering
Division of Building Technology
Building Physics
CHALMERS UNIVERSITY OF TECHNOLOGY
Göteborg, Sweden 2011
Master's Thesis 2011:83

Modelling the performance of underground heat exchangers and storage systems

Master of Science Thesis in the Master's Programme Structural Engineering and Building Performance Design

DAVID VAN REENEN

Department of Civil and Environmental Engineering
Division of Building Technology
Building Physics
CHALMERS UNIVERSITY OF TECHNOLOGY

Göteborg, Sweden 2011

Modelling the Performance of Underground Heat Exchangers and Storage Systems

Master of Science Thesis in the Master's Programme Structural Engineering and Building Performance Design
DAVID VAN REENEN

Cover:
The illustration on the cover shows a diagram showing the S-function of a ground heat exchanger, the mesh around a borehole, and the resulting temperature distribution.

Modelling the Performance of Underground Heat Exchangers and Storage Systems

Master of Science Thesis in the Master's Programme Structural Engineering and Building Performance Design
DAVID VAN REENEN
Department of Civil and Environmental Engineering
Division of Building Technology
Building Physics
Chalmers University of Technology

ABSTRACT

Geothermal heat pumps are systems that combine a closed loop underground heat exchanger or open loop groundwater heat exchanger along with a heat pump. These systems use the ground (or ground water) as a source of heat in the winter or cooling in the winter. When heating and cooling are used on a seasonal basis, the storage of thermal energy can be accomplished. This is useful to reduce the energy demand of buildings. The purpose of this thesis project was to investigate and quantify the performance of ground source heat pumps and underground storage systems. This study focused on the development of a model of underground heat exchangers and their integration into complete building systems. The behaviour of ground source heat pumps and underground storage was numerically evaluated using a combination of the commercial finite element software COMSOL Multiphysics and Matlab/Simulink. Various models of underground heat exchangers have been developed that consider transient conditions of full three dimensional heat transfer interaction between the ground, the underground pipe network, and the external environment. The model simplifies the flow in the pipe network to one dimension flow with transverse heat transfer coupled with a full three dimensional model of the ground to analyze the heat transfer in the entire system. The underground heat exchanger has been integrated into a building model. These models are used to simulate the dynamic behaviour of a building, including all relevant thermal parameters such as heating, cooling, and control systems. Using S-functions in Simulink, the geothermal heat pump has been integrated into the building to observe its overall performance. A number of underground heat exchanger configurations have been studied and their effect on the overall thermal performance of buildings has been analyzed. The cycling of temperatures in the ground during heating and cooling seasons has been observed. The increased cycling of indoor temperature inside a building has also been observed. This cycling is partially due to the slower response of the heat pump and the fact that the heat pump turns off and on due to temperature limits.

Key words: Simulation, geothermal heat pumps, underground heat storage, thermal, whole building simulation

II

Contents

Preface

In this thesis project the development of models simulating the performance of geothermal heat pumps and their integration in buildings has been completed. This project was carried out from September 2010 to June 2011 as a joint project between Chalmers University of Technology, Sweden and Technical University of Eindhoven, Netherlands. The project was jointly supervised by Professor Angela Sasic Kalagasidis of the Division of Building Technology in the Department of Civil and Environmental Engineering at Chalmers University of Technology and Assistant Professor A.W.M. (Jos) van Schijndel of the Division of Building Physics and Services of the Department of Architecture, Building, and Planning of the Technical University of Eindhoven.

The project was initiated at Chalmers University of Technology in September 2010 through December 2010. It was continued at the Technical University of Eindhoven in January 2011 through April 2011. The project was finalized again at Chalmers University of Technology in May and June 2011.

I would like to thank both Angela Sasic Kalagasidis and Jos van Schijndel for their help and guidance in the development and conclusion of this project. They have both been very helpful and supportive. I would also like to thank my colleague, Natalie Williams Portal, for her support and helpfulness as we worked on separate projects with the same supervisors.

Gothenburg June 2011

David van Reenen

Notations

Roman upper case letters

Bi	Biot number, [–]
COP	Coefficient of performance, [–]
D	Depth of a pipe in the ground, [m]
D_h	Hydraulic radius, [m]
Fo	Fourier number, [–]
L	Length or thickness, [m]
$\dot{M}$	Mass flow rate, [kg/s]
Nu	Nusselt number, [–]
P	Pressure, [Pa]
Pr	Prandtl number, [–]
Q	Heat flow rate, [W]
R_e	Reynolds number, [–]
T	Temperature, [K]

Roman lower case letters

c_p	Specific heat capacity, [J/(kg·K)]
h	Enthalpy, [J/kg]
k	Heat pump efficiency, [–]
l_c	Characteristic length, [m]
η_t	Heat exchanger efficiency, [m]
q	Heat flux, [W/m^2]
u	Fluid velocity, [m/s]
$\boldsymbol{u}$	Fluid velocity vector, [m/s]
t	Time, [s]
v	Absolute viscosity of water, [m^2/s]
x, y, z	Cartesian coordinates [m]

Greek lower case letters

α_0	Heat transfer coefficient [W/K·m^2]
λ	Thermal conductivity [W/mK]
ρ	Density [kg/m^3]
q	Heat flow [W/m^2]

Subscripts

c	Convective
ext	Exterior
g	Ground
i	Interior
in	Inlet
l	Liquid
o	Outer

1 Introduction

1.1 Background

Warming of the global climate is now considered unequivocal as evidenced by warming temperature in the air and the oceans, melting of snow and ice, and the rising sea levels (IPCC, 2007). One way to combat global warming is to reduce the energy demand used for the heating and cooling of buildings.

Geothermal heat pumps are systems that combine a closed loop ground heat exchanger or open loop groundwater heat exchanger along with a heat pump. These systems use the ground (or ground water) as a source of heat in the winter and/or cooling in the summer. In heating mode the ground is used as a condenser while in the summer the ground is used as an evaporator. A typical installation consists of a borehole containing two small diameter tubes linked at the bottom with a U–bend. This is termed a borehole heat exchanger (BHE). For each kWh of heating or cooling energy that the systems output, an input of 0.22 to 0.35 kWh of electricity is required. This results in a 30 to 50% lower power consumption than typical air–to–air heat pump system (Sanner et al., 2003).

Energy storage systems are also used to achieve a reduction in total energy use of buildings. Excess energy can be captured when available and stored for use when there is a lower supply or higher demand. This can result in a reduction in the total and peak energy use of buildings. In buildings, heat is normally the type of energy that is captured and stored. These storage systems are used to store heat captured during warm periods and then used to heat the building during colder periods. This can be done on a diurnal or seasonal basis. It can also be used for storing energy that can only be generated during limited periods. An example of this is storing solar energy. Heat storage can be valuable on both a short term and long term basis. In the short term, energy can be stored for use in diurnal temperature variation or during an emergency loss of energy supply. Long term storage can allow for seasonal storage of energy such as described above. Underground thermal energy storage systems can consist of either open aquifer storage or underground heat storage systems. These systems can be used for cooling of a building, but also for the storage of solar or other waste heat that can then be used for seasonal heating (Sanner et al., 2003).

Ground heat storage, using geothermal heat pumps, is an example of long term storage. It is not as useful for short term storage, but can be successfully used for the seasonal storage of energy (Hellström, 1991).

Duffie and Beckman (1975) described three requirements for heat storage:

1. It must be able to receive and discharge heat with relatively small temperature differences.

2. It should have small energy losses

3. It should be inexpensive.

Previous studies have been conducted on geothermal heat pumps and underground storage systems. They have generally consisted of simplified one dimensional or two dimensional systems and many have been limited to the long term response of the systems. This study will focus on the development of a three dimensional model of the ground and underground heat exchangers. It will include complex modelling of underground heat exchangers, such as BHEs, and their integration into buildings. It

will also examine the performance of BHEs on their own and as part of a complete building system.

1.2 Objective and scope

The aim of the project was to investigate underground heat exchangers and model their short and long term performance. This study focused on the development of a model for the integration of geothermal heat pumps into buildings and examined their use as heat storage systems. Numerical analysis was executed using a combination of Matlab/Simulink and the commercial finite element software COMSOL.

This study is related to previous research conducted on the principles of heat extraction from the ground by horizontally and vertically buried pipes. A three dimensional model of underground heat exchangers was developed in COMSOL that considered both steady state and transient conditions. This model included a full three dimensional representation of the ground and the borehole. Building models in Matlab/Simulink have also been studied. The two models were combined to observe the overall performance of a model building with an integrated geothermal heat pump system. A parametric study of a simplified building model along with a case study of an existing building is presented to accomplish this.

This thesis report documents details of the development of a model for underground heat exchanger along with some theoretical background of the model. It details the integration of this model into a building model and the performance of the system as part of a typical building and a case study of an existing building.

1.3 Limitations

This study has focused on the development of a model for ground–coupled heat pumps. It has focused on two types of systems: a horizontal pipe parallel and near the ground surface and BHEs. Other types of systems were not considered.

The models developed also make assumptions regarding the thermal properties of the ground. The ground is assumed to be homogeneous and in the case studies uniform thermal properties are assumed. The model does, however, allow for the properties to be varied. There is also assumed to be no movement of ground water.

2 Theoretical Background

2.1 Underground heat exchanger

Ground–source heat pumps were originally developed for residential buildings. These can consist of several different types of systems. These include ground–coupled, groundwater, and surface water heat pumps. (ASHRAE, 2003).

This study has focused on ground–coupled heat pumps (GCHP). GCHPs combine an underground heat exchanger with a heat pump in a closed loop. They use the earth as a heat source when operating in heating mode using a fluid to transfer heat from the ground to the evaporator of the heat pump. In cooling mode, the cycle is reversed and the ground is used as a heat sink (Sanner et al., 2003).

The underground heat exchanger can be one of two types. A horizontal system consists of pipes laid horizontally in the ground close to the surface. These systems take a large amount of ground surface area and can be affected by changes in the outdoor weather. Vertical systems, often termed borehole heat exchangers (BHEs), consist of two small diameter tubes linked at the bottom with a U–bend. These connected pipes are then inserted into a vertical borehole ranging in depth from 15 to 200 m (ASHRAE, 2003). A schematic diagram of a typical BHE is shown in Figure 2.1.
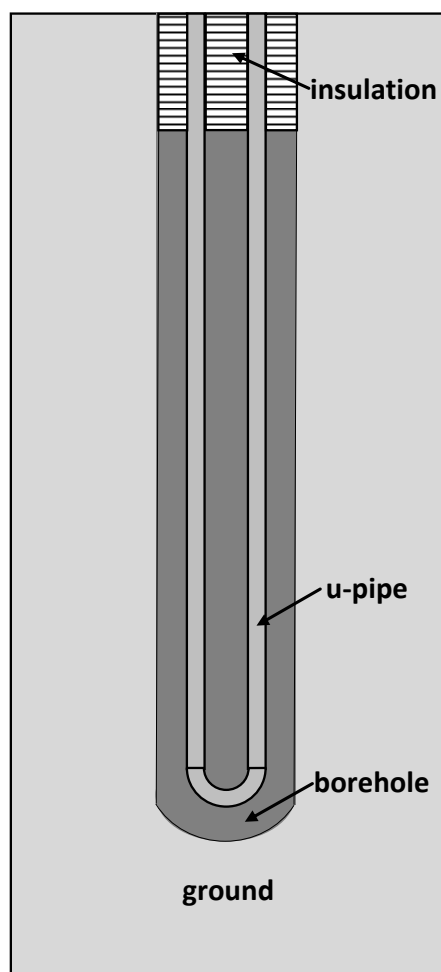

Figure 2.1   Schematic diagram of a BHE.

One detailed study presented a selection of analytical solutions of heat conduction in the ground (Claesson and Dunand, 1983). The goal of this study was to assess the potential for extracting heat from the ground in different situations using mathematical methods. It considered two dimensional heat extraction with various pipe configurations that could be solved analytically. It was limited to horizontal pipes laid in the ground.

Studies have also been undertaken to model the performance of BHEs in two dimensions. One such study compared eight different single borehole cross sections (Acuna and Palm, 2009). This study considered steady state heat transfer to the ground to obtain a borehole thermal resistance and compared the performance of the different configurations.

2.2 Heat transfer

Heat transfer is the process involving the transport energy through materials across a temperature difference. This study focused on heat transfer by both conduction and convection. Heat transfer in the ground was limited to conduction. Heat transfer in the pipes of the underground heat exchanger was simulated as convection through the pipe with transverse heat loss to the ground. As a result, the temperature in the pipe increases or decreases as it flows from the inlet to the outlet of the system. This section details the basic theory and equations used for solving the energy balance equations for developing a full model of the underground heat exchanger.

2.2.1 Heat transfer by conduction

Heat flow by conduction is governed by Fourier's law. The density of heat flow through a solid by heat conduction in one dimension is defined by:

$$q = -\lambda \frac{\Delta T}{\Delta x} \tag{2.1}$$

Fourier's law in differential form is shown in Equation (2.2). This equation considers temperature, T, as a function of the space coordinates x, y, and z along with time.

$$q = -\lambda \nabla T \tag{2.2}$$

Conduction in materials is modelled using Fourier's Second Law. Equation (2.3) shows the version of the heat equation used in this study.

$$\rho c_p \frac{\partial T}{\partial t} - \nabla \cdot (\lambda \nabla T) = Q \tag{2.3}$$

Heat flow by a moving fluid is governed by heat convection. The heat flux carried by a moving fluid through a control volume by heat convection is defined by

$$q = \rho \cdot u \cdot h \tag{2.4}$$

In the general case enthalpy is a function of both the internal temperature and pressure of the fluid.

$$dh = \left(\frac{\partial h}{\partial T}\right)_P \cdot dT + \left(\frac{\partial h}{\partial P}\right)_T \cdot dP \qquad (2.5)$$

$$h = \int_T c_p \cdot dT + \int_P \left(\frac{\partial h}{\partial P}\right)_T \cdot dP \qquad (2.6)$$

In the case of the underground heat exchanger, the fluid flowing through the system can be considered as non–compressible, meaning:

$$\left(\frac{\partial h}{\partial P}\right)_P = 0 \qquad (2.7)$$

And since c_p is constant

$$h = c_p \cdot T \qquad (2.8)$$

The heat flow by convection is then

$$q = \rho \cdot u \cdot c_p \cdot T \qquad (2.9)$$

The energy balance for a control volume leads to the differential equation.

$$\rho C_p \frac{\partial T}{\partial t} = -\nabla q = -\rho \cdot u \cdot C_p \cdot \nabla T \qquad (2.10)$$

The above is the final equation for modelling the heat transfer in the entire underground heat exchanger system.

2.2.2 A pipe with transverse heat flow

As an alternative to a full three dimensional model of fluid flow and heat transfer in a pipe, a simplified one dimensional model was examined. Figure 2.2 shows a simple pipe with a surface resistance surrounded by a steady–state temperature of T(x) along the length of the pipe. Around the pipe the temperature is constant. Temperature differences inside the pipe and perpendicular to the flow are neglected. The liquid flows at a constant mass flow rate $\dot{M}_l$.
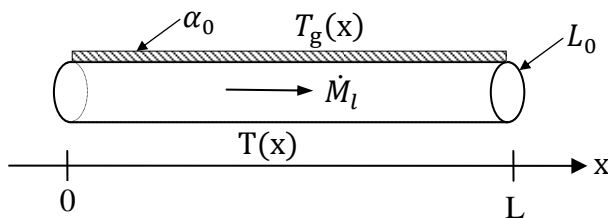

Figure 2.2   Pipe with transverse heat flow.

The analytical solution for an air channel with transverse heat flow can easily be determined (Hagentoft, 2001). This solution is applied to a fluid flow in a pipe in the

ground with a circumference of L_0. The temperature of the fluid in the pipe is solved for based on a known temperature in the ground.

The convective heat flow out of the channel is:

$$Q_c(x) = c_p \dot{M}_l T(x) \tag{2.11}$$

The differential equation for the heat balance between the fluid and the ground is:

$$-\frac{d}{dx}[Q_c(x)] + L_0 \alpha_0 \left(T_g(x) - T(x)\right) = 0 \tag{2.12}$$

The solution for the basic case with a constant ground temperature, pipe size, ground conductivity, and flow through the pipe is shown below.

$$T(x) = T_g + (T_{in} - T_g)^{-x/l_c} \tag{2.13}$$

The length l_c [m] is known as the characteristic length for the interaction between the convective heat flow and transverse heat loss along the channel. It is defined by:

$$l_c = \frac{c_p \dot{M}_l}{L_0 \alpha_0} \tag{2.14}$$

2.3   Modelling of conduction and convection

Modelling of the heat transfer in the underground heat exchanger has been accomplished using the computer software package COMSOL Multiphysics. COMSOL is a finite element modelling software package that contains a number of predefined physics interfaces, including heat transfer and fluid flow. It also facilitates the development of models using arbitrary partial differential equations input by the user. The software is then used to solve these equations using the finite element method with various mathematical solvers. Coupling between various physical interfaces or between entirely different models with varying geometries can also be accomplished.

In this project, COMSOL version 4.0a was initially used to develop the models of the underground heat exchangers. Later in the project it was found there were some difficulties when trying to integrate the final models in Matlab as an S–function. There appeared to be some memory leaks in that version software, resulting in memory errors when running simulations for multiple iterations. Due to this, a switch to COMSOL 3.5a was initiated. Since this was an older version, it also had a more mature MATLAB interface resulting in a more straightforward approach for the use of S–functions.

COMSOL can be used to model heat transfer by conduction, convection, and surface radiation using its heat transfer module. Full practical and theoretical details of this module are available in the *COMSOL Heat Transfer Module User's Guide* (COMSOL AB, 2008). Heat transfer via conduction, such as shown in Equation (2.3) can be modelled as heat transfer through convection in fluids as shown in Equation (2.10).

For heat transfer in fluids, including both conduction and convection, the equation modelled in COMSOL is shown in Equation (2.15).

$$\rho c_p \frac{\partial T}{\partial t} + \rho c_p \mathbf{u} \cdot \nabla \mathrm{T} = \nabla \cdot (k\nabla T) + Q \qquad (2.15)$$

Note that the velocity, **u**, is in bold typeface, indicating that it is a vector. Also note that k is used to indicate the thermal conductivity of the materials as opposed to λ which is used in the remainder of the report.

In order to model the heat transfer between the ground and the pipe and the corresponding heat transfer between the pipe and the ground linear extrusion coupling has been used. An edge extrusion variable, T_geom1, has been created in the model of the ground as shown in Figure 2.3. This variable is mapped to a source which is the line in the ground, which represents the pipe. The destination for this variable is then specified as the line in the pipe model. The corresponding extrusion variable for the temperature of the fluid in the pipe, T_geom2, is mapped in the same way.
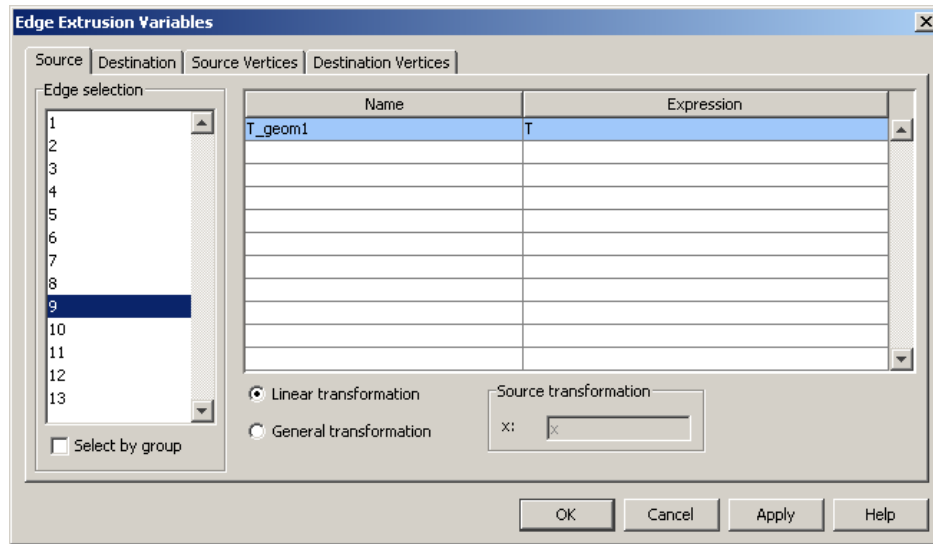
Figure 2.3   Linear Edge Extrusion variable specified in the model of the ground.

Equations were then input to evaluate the heat transfer between the ground and the pipe. The heat transfer from the pipe to the ground was input as a weak term added to the edge equation setting for the pipe edge as shown below.

<div align="center">

test(T)·alpha0·L0·(T_geom2-T)   *(2.16)*

</div>

The heat flux into the pipe from the ground could simply be added as a domain heat source with the equation.

<div align="center">

alpha0·L0·(T_geom1-T)   *(2.17)*

</div>

This model could then be used to simulate the thermal performance of a horizontal pipe in the ground. For more information on the how the model was developed in complete details of the model developed in COMSOL see Appendix A.

3 Development of an Underground Heat Exchanger Model

In the process of developing the model of an underground heat exchanger a number of steps were undertaken to verify its performance against several different analytical solutions. The verifications present the development of the model from conduction in the ground and convection in the pipe modelled independently to the model of one dimensional pipe flow fully coupled to three dimensional conduction in the ground.

The first section of this chapter details verifications that consider conduction in the ground. The simulations that were conducted are:

- Semi infinite ground region with a line heat source
- Transient response – temperature step change in a cylinder

Modelling of heat convection in a pipe with a moving fluid was then considered. The simulations that were conducted are:

- Pipe flow modelled with a one dimensional PDE
- Pipe flow modelled with a predefined heat transfer module
- Comparison of one dimensional pipe convection to coupled fluid dynamics and heat transfer
- Transient response – one dimensional pipe convection

With the verification of uncoupled conduction in the ground and convection in the pipe complete, coupling of the heat transfer was examined considering two dimensional conduction in the ground. In this case steady state coupling was examined.

Finally, coupling of one dimensional flow with three dimensional conduction was examined. This was done to conclude the verification of the final developed models. The simulations that were conducted are:

- Fluid flow with a constant ground temperature
- Fluid flow with a coupled ground temperature
- Fluid flow in a pipe near ground surface
- Transient response

The above verifications and developments are presented below in the following sections.

3.1 Heat conduction

3.1.1 Line model with constant heat flux from a horizontal pipe

A line model of an infinite steady–state horizontal pipe acting as a heat sink can be solved analytically (Claesson and Dunand, 1983). The ground is considered as a semi–infinite region with a single pipe in the ground. The rate of heat exchange from the ground to the pipe is q (W/m). The ground is homogeneous and isotropic with a constant thermal conductivity of λ (W/m·K).
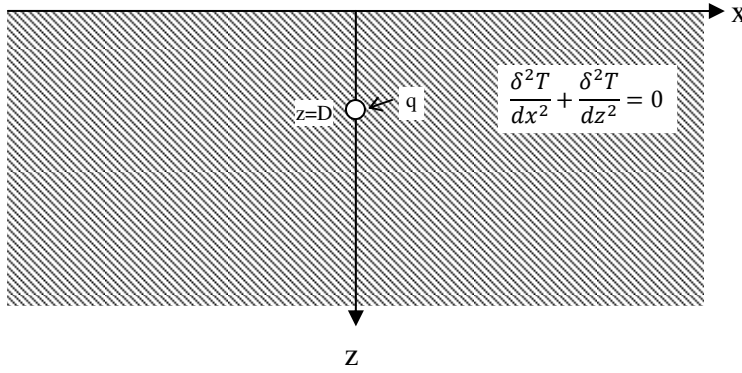
Figure 3.1   Simple two dimensional model of a horizontal pipe in the ground.

The steady state temperature in the ground for a point sink can be solved analytically with Equation (3.1) shown below.

$$T(x,z) = \frac{q}{2 \cdot \pi \cdot \lambda} \cdot ln \frac{\sqrt{x^2 + (z - D)^2}}{\sqrt{x^2 + (z + D)^2}} \qquad (3.1)$$

As an analytical verification of the heat transfer module in COMSOL, A two dimensional model was developed using the 'Heat Transfer in a Solid' physics.

A reference case to benchmark the COMSOL model to the analytical equation was accomplished using the same properties as in a previous study (Claesson and Dunand, 1983). The properties used were:

$$\lambda = 1.5 \text{ W/m·K} \qquad\qquad D = 1 \text{ m} \qquad\qquad q = 10 \text{ W/m}$$

Several simulations were conducted to compare the analytical solution with the results from COMSOL.

The first simulation consisted of a ground temperature of 10 °C with a heat source of 10 W/m at a depth of 1 m in the ground. The resulting temperature difference, as calculated by COMSOL is shown in Figure 3.2.
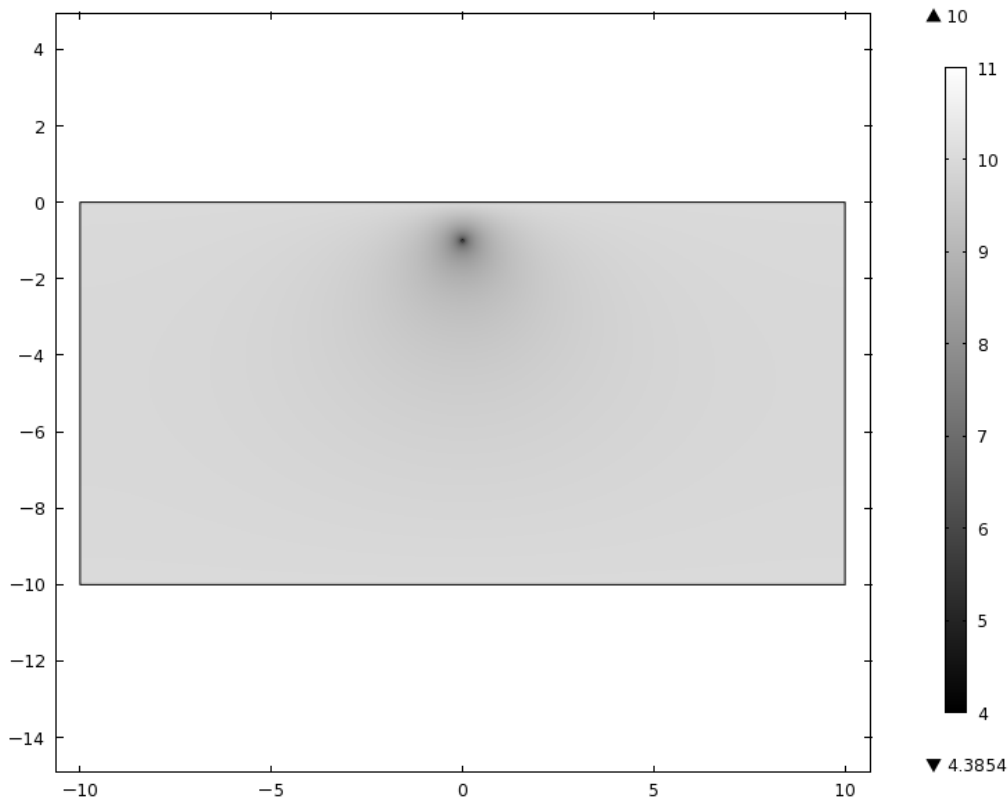
Figure 3.2   Temperature [°C] distribution around a pipe at depth of 1 m in the ground.

A series of simulations were also conducted to determine the sensitivity of results to changes in the domain size of the ground since the analytical solution considers a semi–infinite ground region. The results of these simulations were compared to the analytical solution to get an approximate requirement for the ground domain size in future simulations. The results of this comparison are shown in Figure 3.3. This figure shows the temperature along a vertical line through the centre of the pipe from the ground surface to a depth of 10 m. The lines on the figure indicate that the analytical solution is approached as the domain size is increased. Figure 3.4 shows the temperature difference between the analytical solution and the COMSOL simulation for the location at the centre line of the pipe at a 10 m depth.

These figures indicate that a domain size of 10 m by 20 m is quite close to the analytical solution but there is still a visible difference. With a domain size of 20 m by 40 m the difference in temperatures at a 10 m depth is below 0.05 °C. In future simulations, based on this result, the domain size of the ground around the pipe was maintained at greater than 20 m.
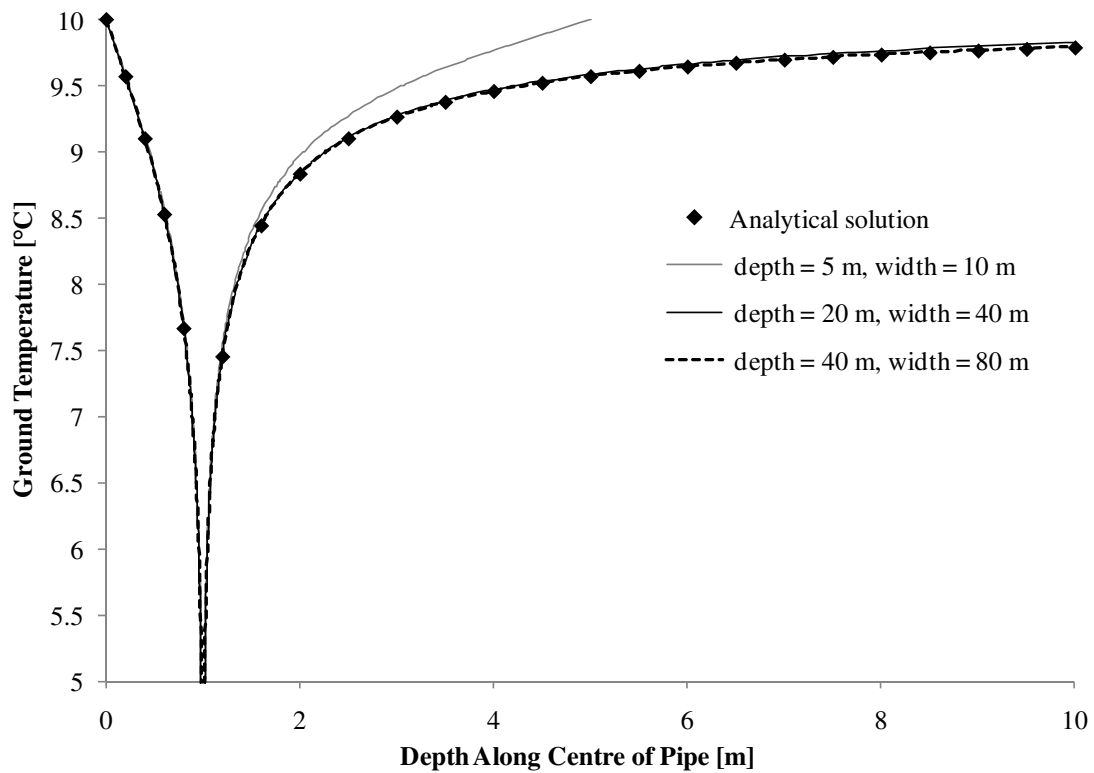
Figure 3.3   Ground domain size comparison.
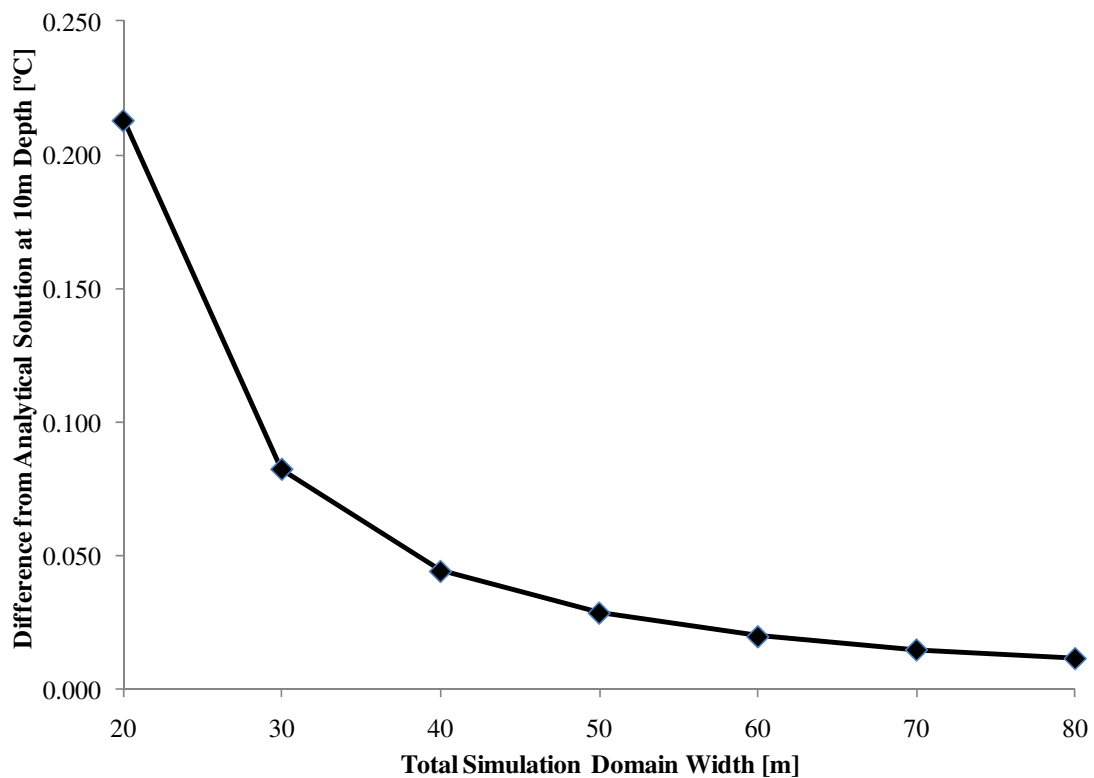
Figure 3.4   Temperature difference between COMSOL simulation and analytical solution along the centre of the pipe at a depth of 10 m.

3.1.2 Step Response for a cylinder

As a validation of transient heat conduction, the step response of a cylinder to a change in temperature was also modelled in COMSOL. The dimensionless solution to the temperature is show in Equation *(3.2)* (Hagentoft, 2001). This solution for temperature T is for a long cylinder with an initial temperature of T_0 with a surface temperature of T_1. Full details of the solution are found in the source.

$$u(\text{r}, \text{t}) = 2\mathbf{Bi} \sum_{\text{n}=1}^{\infty} e^{-\beta_n^2 \mathbf{Fo}} \frac{J_0(\text{r} \cdot \beta_\text{n}/r_0)}{(\beta_\text{n}^2 + \mathbf{Bi}^2) \cdot J_0(\beta_\text{n})} \qquad (3.2)$$

Where,

$$u = \frac{T - T_0}{T_1 - T_0} \qquad (3.3)$$

The β numbers are the positive roots of the equation:

$$\beta J_1(\beta) = \mathbf{\textit{Bi}} \cdot J_0(\beta) \qquad (3.4)$$

In COMSOL a three dimensional cylinder was constructed as shown in Figure 3.5. The cylinder was simulated as a solid object with only heat transfer by conduction. This was done just to test the transient solvers in COMSOL.

Figure 3.6 shows a comparison of the COMSOL and Analytical solution for the surface temperature of the cylinder. The results show a good agreement between the two solutions.
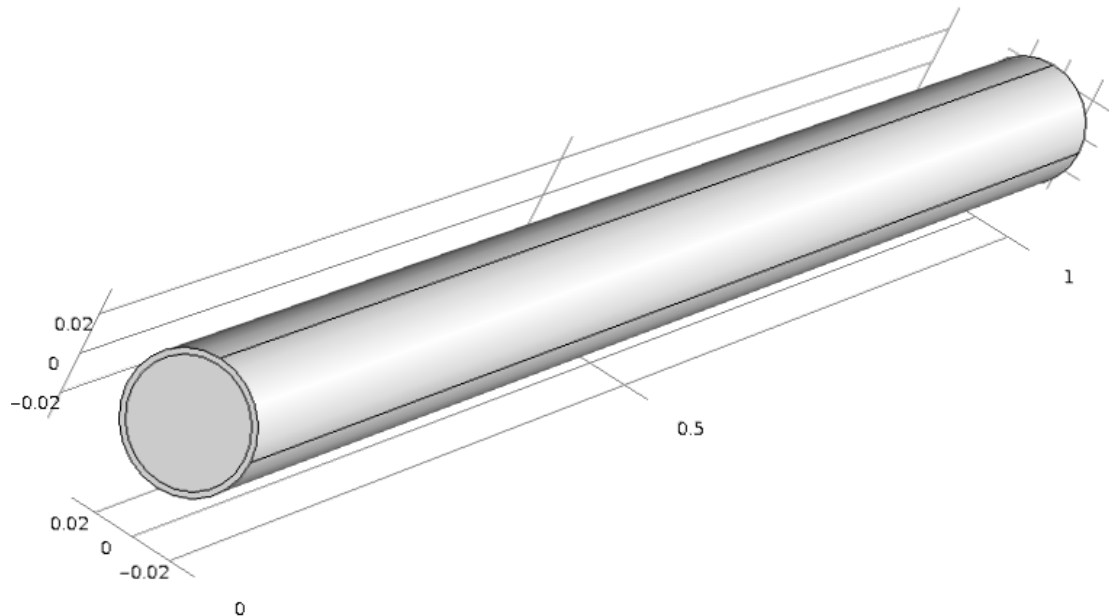
Figure 3.5   Cylinder geometry used in step change simulation.
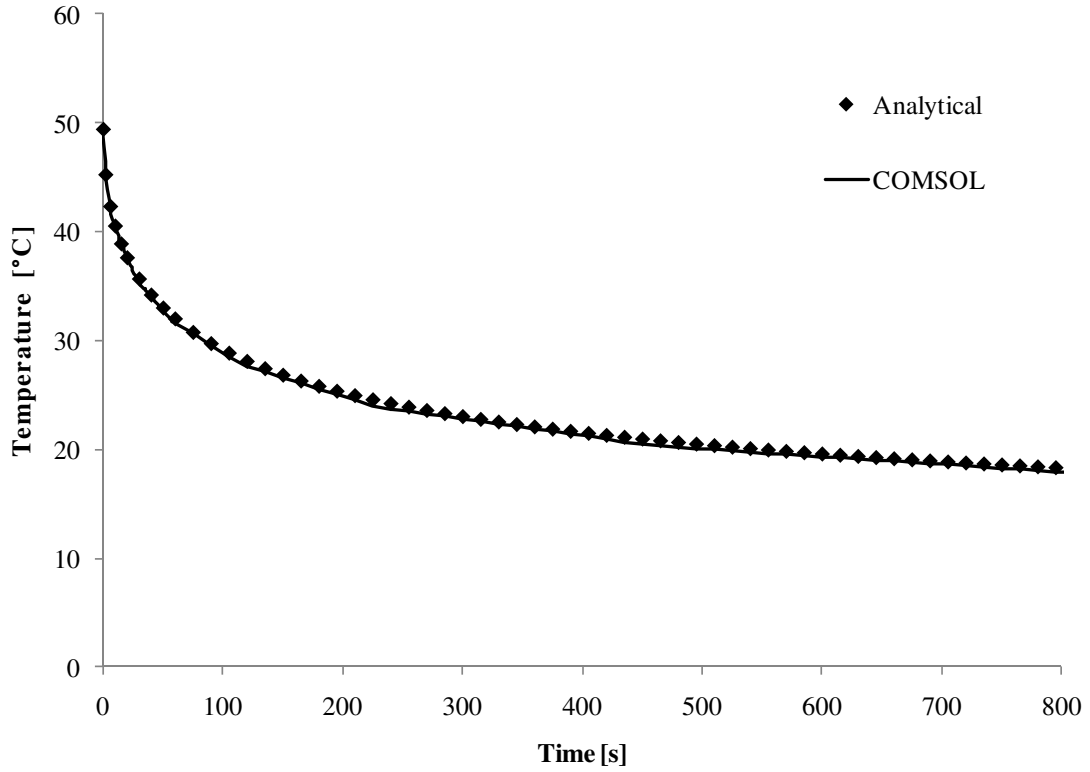
Figure 3.6   Response of the surface temperature along a long cylinder to a step change in the inner temperature of the cylinder.

3.2   Heat convection in a pipe

3.2.1 Pipe flow modelled with a one dimensional PDE

COMSOL can be used to solve arbitrary partial differential equations (PDEs) in both a coefficient and general form. In this section the coefficient form of the partial differential equation has been used to solve a one dimensional heat transfer problem. A model was developed to verify COMSOLs modelling of fluid flow in a pipe with transverse heat flow as described in Section 2.2.2.

In COMSOL, Equation *(2.12)* was modelled using the coefficient form PDE interface. Equation (3.5) shows the implementation of this PDE that was used to model a steady state simulation of the pipe (COMSOL AB, 2008).

$$e_a \frac{\partial^2 u}{\partial t^2} + d_a \frac{\partial u}{\partial t} + \nabla \cdot (-c \nabla u - \alpha u + \gamma) + \beta \cdot \nabla u + au = f \qquad (3.5)$$

The convection equation was modelled by setting $\beta = c_p \cdot \dot{M} \cdot T$, $a = L_0 \alpha_0$, and $f = L_0 \alpha_0 T_0(x)$. Note that the heat capacity, c_p, and the mass flow rate, $\dot{M}_l$, were both assumed to be constant. Since there was assumed to be negligible conduction in the pipe fluid, the parameters c, α and γ were all set to zero. Steady state conditions were also assumed and e_a and d_a were also set to zero. This resulted in Equation (3.6).

Equation:

$$-c_p \dot{M} \nabla T - L_0 \alpha_0 T = L_0 \alpha_0 T_0(x)$$

(3.6)

In COMSOL:

$$\beta \cdot \nabla u + au = f$$

To compare the COMSOL model with the analytical solution the determination of the surface resistance α_0 around the pipe was required. This surface resistance consists of three components: the surface resistance between the fluid and the pipe, the thermal resistance of the pipe, and the contact resistance between the ground and the pipe. In this validation it was assumed that water was flowing through the pipe.

The surface resistance between the water and the pipe can be calculated by considering the conditions of the flow through the pipe. Realistic flow conditions in the pipe are shown in Table 3.1. These conditions were found in table of sizing information on ground heat exchangers (ASHRAE, 2003).

Table 3.1   Flow conditions.

Description	Symbol	Value	Units
diameter of pipe	D	0.06	m
velocity of water	V	0.14	m/s
density of water	ρ	1000	kg/m^3
heat capacity of fluid	Cp	4200	W·s/kg

To determine if the flow was turbulent or laminar flow, Reynolds Number was calculated using Equation *(3.7)*. The result of 8400 indicated that the flow was turbulent.

$$R_e = \frac{u \cdot D_h}{v}$$

(3.7)

The Nussselt number for turbulent flow was then found using the Dittus and Boetler relation for heating (Rohsenow, Hartnett, and Cho, 1998) as shown in Equation *(3.8)*.

$$Nu = 0.024 \cdot Re^{0.8} \cdot Pr^{0.4}$$

(3.8)

The convection heat transfer coefficient, α_c, was then determined using Equation (3.9) from the Nussselt number.

$$Nu = \frac{\alpha_c D_h}{\lambda_w}$$

(3.9)

The resulting heat transfer coefficient was calculated at approximately 700 W/(m^{2}K).

The thermal resistance of the pipe was calculated using Equation (3.10) and then inverted to find the heat transfer coefficient (Claesson and Dunand, 1983).

$$m_{pw} = \frac{1}{2\pi\lambda_p}\ln\left(\frac{R_o}{R_i}\right) \tag{3.10}$$

The ability of the ground to remove heat from the area around the pipe was assumed to infinite. This assumption was made in order to simulate a larger temperature drop along the length of the pipe. This was done to make the comparison between the COMSOL simulation and the analytical solution easier to visualize.

Using the PDE interface in COMSOL and solving the equation as shown in Equation (3.6) a validation of COMSOL was accomplished. The results of this are shown in Figure 3.7. This shows an exact match between the analytical and COMSOL results.

Figure 3.7   Comparison of the analytical solution to COMSOL's PDE interface solution for flow in a channel with transverse heat loss.

3.2.2 Pipe flow modelled using a predefined heat transfer module

The heat transfer module in COMSOL is designed for heat transfer by conduction, convection, and radiation. This allows for the simulation of heat transfer in gases, liquids, and solids. The heat transfer in fluids module allows for simulation of conduction and convection in a moving fluid. Equation (2.15) shows the differential equation for heat transfer in fluids. Using the same parameters as in Section 3.2.1 steady–state transverse heat loss in a pipe was simulated using COMSOL. The results, shown in Figure 3.8, indicate an exact match with the analytical solution.
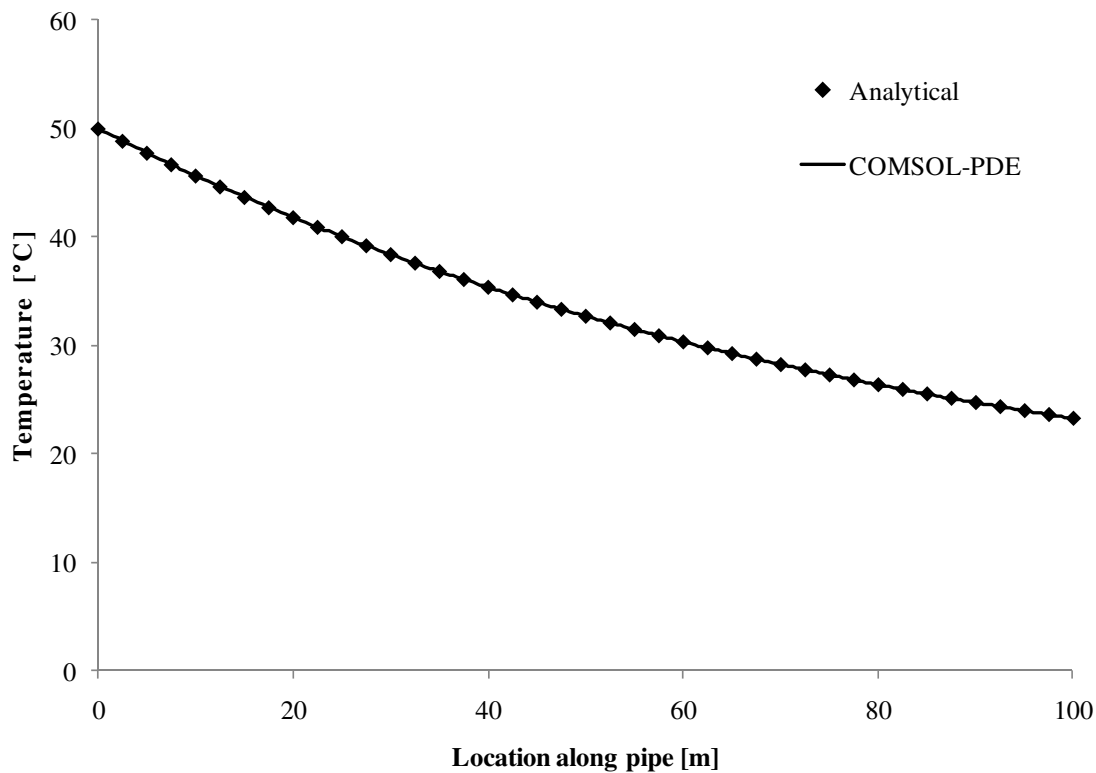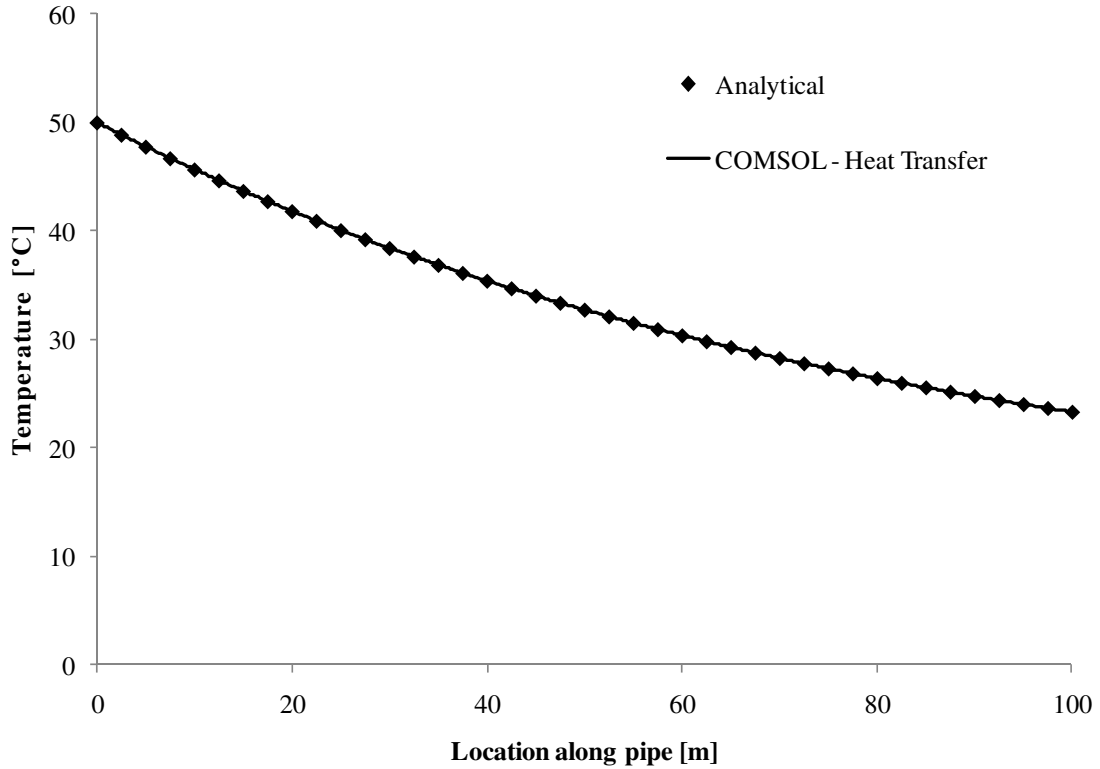
Figure 3.8   Comparison of an analytical solution to COMSOL's heat transfer interface solution for flow in a channel with transverse heat loss.

3.2.3 Coupled fluid dynamics and heat transfer

To analyze the transverse heat transfer from the fluid in the pipe to the ground several simulation were attempted in COMSOL using the interface for coupled heat transfer and turbulent flow. In COMSOL this model is called 'Conjugate Heat Transfer' and is part of COMSOL's CFD Module. Full details of the CFD Module can be found in the COMSOL CFD Module User's Guide (COMSOL AB, 2008). Some theoretical background regarding the modelling of the fluid dynamics is shown below.

Fluid flow in the pipe has been modelled using the $k - \varepsilon$ turbulence model. This model introduces two transport equations and two dependent variables:

- k, the turbulent kinetic energy, and
- ε, the dissipation rate of turbulence energy.

Turbulent viscosity is modelled by Equation *(3.11)*.

$$\mu_T = \rho C_\mu \frac{k^2}{\varepsilon} \qquad\qquad (3.11)$$

where,

   C_μ is a model constant.

The transport equation for *k* is shown below in Equation *(3.12)*.

$$\rho \frac{\partial k}{\partial t} + \rho \mathbf{u} \cdot \nabla k = \nabla \cdot \left(\left(\mu + \frac{\mu_T}{\sigma_k} \right) \nabla k \right) + P_k - \rho \varepsilon \qquad (3.12)$$

Where the production term is as indicated below in Equation *(3.13)*.

$$P_k = \mu_T \left(\nabla \mathbf{u} : (\nabla \mathbf{u} + (\nabla \mathbf{u})^T) - \frac{2}{3} (\nabla \cdot \mathbf{u})^2 \right) - \frac{2}{3} \rho k \nabla \cdot \mathbf{u} \qquad (3.13)$$

Equation *(3.14)* shows the transport equation for ε.

$$\rho \frac{\partial \varepsilon}{\partial t} + \rho \mathbf{u} \cdot \nabla \varepsilon = \nabla \cdot \left(\left(\mu + \frac{\mu_T}{\sigma_\varepsilon} \right) \nabla \varepsilon \right) + C_{\varepsilon 1} \frac{\varepsilon}{k} P_k - C_{\varepsilon 2} \rho \frac{\varepsilon^2}{k} \qquad (3.14)$$

As a validation of the one dimensional channel flow simulation shown in Section 2.2.2, an axisymetric model of a 50 m length of pipe with a constant temperature boundary conditions was assembled. The geometry of the model is shown in Figure 3.9. In order to limit the size and number of elements in the model, it included only the fluid in the pipe and the pipe itself. The temperature around the outer edge of the pipe was assumed to be constant. This was done to match the assumption of constant ground temperature made in the channel flow simulations above.

In COMSOL the 'Conjugate Heat Transfer Interface' is set up to model heat transfer through a fluid in collaboration with a solid where heat is transferred by conduction. The interface for conjugate heat transfer includes models for turbulent flow including fast moving fluids that have a high Reynolds number. This interface also adds functionality for calculating the dispersion of heat transfer due to turbulence. This is a complex model that was used to validate the much simpler one dimensional pipe flow model.

The temperature in the water along the centre of the pipe is compared to the one dimensional analytical solution as shown in Figure 3.10. The results show a reasonably close agreement between the two solutions. The two dimensional, axisymetric model shows a short flat section where the turbulent flow develops followed by a slightly larger decrease in temperature along the length of the pipe. The analytical model appears to show a smaller decrease in temperature, but does provide a good model for pipe flow.
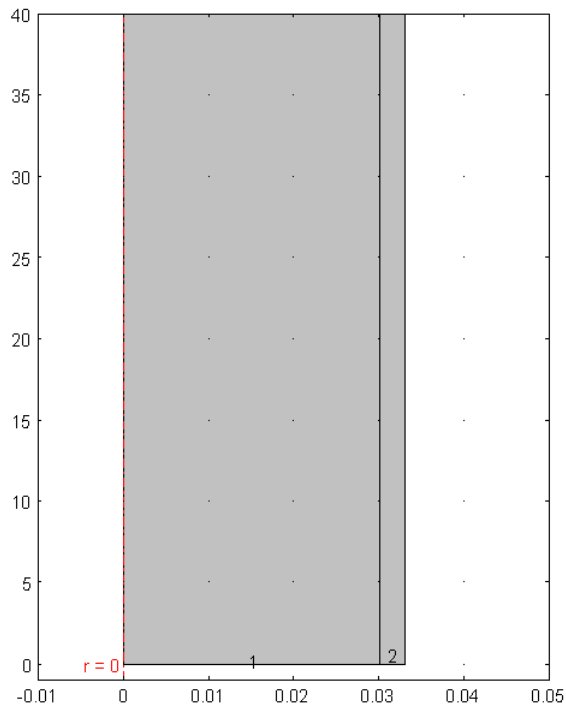
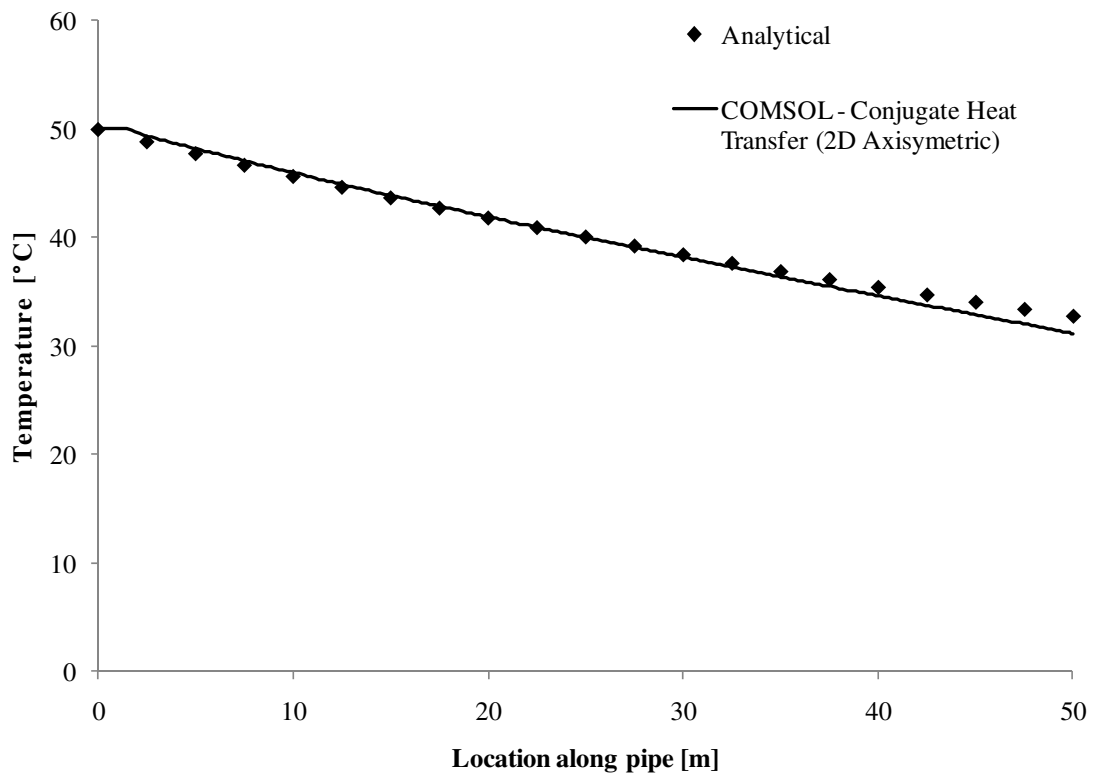*Figure 3.9*   *Geometry of axisymetric pipe simulated using the 'Conjugate Heat Transfer' interface.*

Figure 3.10   *Steady–state comparison of three dimensional conjugate heat transfer and one dimensional channel flow.*

3.2.4 Transient simulation of one dimensional fluid flow

A transient simulation was attempted to validate the performance of COMSOL in a one dimensional pipe as it responds simply to a step change in the temperature of the water at the inlet. This simulation assumed no transverse heat transfer along the length of the pipe and no conduction in the pipe fluid. The fluid in the pipe was initially at 12 °C. At time zero the water at the inlet was changed to 50 °C. Due to the step change in the inlet temperature, a hot wave should move down the pipe until the temperature in the entire pipe is 50 °C.

When using COMSOL's default time dependent solver, oscillation in the solved temperature were observed as shown in Figure 3.11. To reduce the oscillations and improve the accuracy of the simulations, several modifications were made in COMSOL. These changes included reduction of the time step, reduction in the element size, and reduction of the tolerance of the solver.

COMSOL also has an additional feature for handling numerical instabilities within the heat transfer interface. One of the techniques is to add terms to the transport equations. This is termed artificial diffusion and can be used to stabilize the solution. Details of the methods can be found in the COMSOL documentation (COMSOL AB, 2008). With artificial diffusion the oscillations can be controlled as shown in Figure 3.12. The general performance of the simulation is similar to an exact step change. There is, however, some smoothing of the transition from the initial to inlet temperature of the pipe. The impact of this will be relatively small, since the simulated response, on average, is similar to the exact solution.

Figure 3.11   Transient simulation of convective heat flow responding to a step change in the inlet temperature using COMSOL's default solver.

Figure 3.12   Transient simulation of convective heat flow responding to a step change in the inlet temperature with solver stabilization.

A second transient simulation of a one dimensional pipe as it responds to a step change in the temperature of the water at the inlet was conducted. In this simulation transverse heat transfer along the length of the pipe was considered. Heat loss to a medium at constant temperature was used. The fluid in the pipe was initially at 12 °C. At time zero the water at the inlet was changed to 50 °C. The results of this simulation are shown in Figure 3.13. These results show that as the fluid flow through the pipe, the temperature changes from the initial ground temperature to the steady state temperature solution as shown in Figure 3.8.
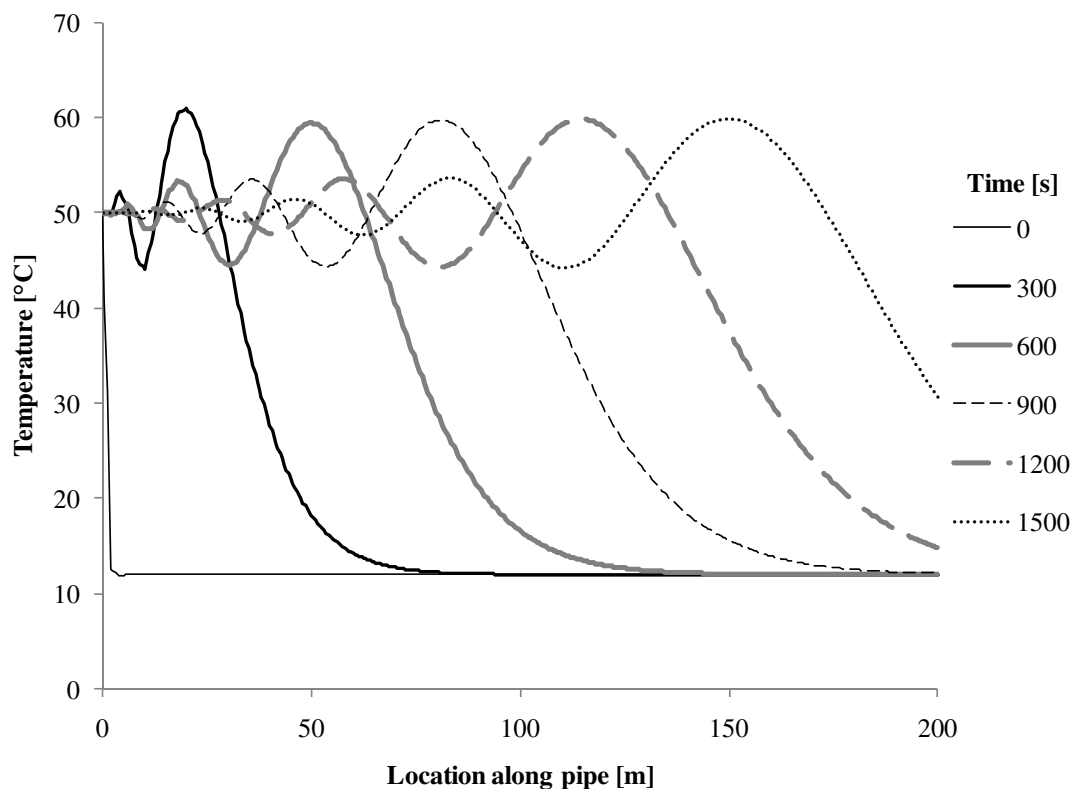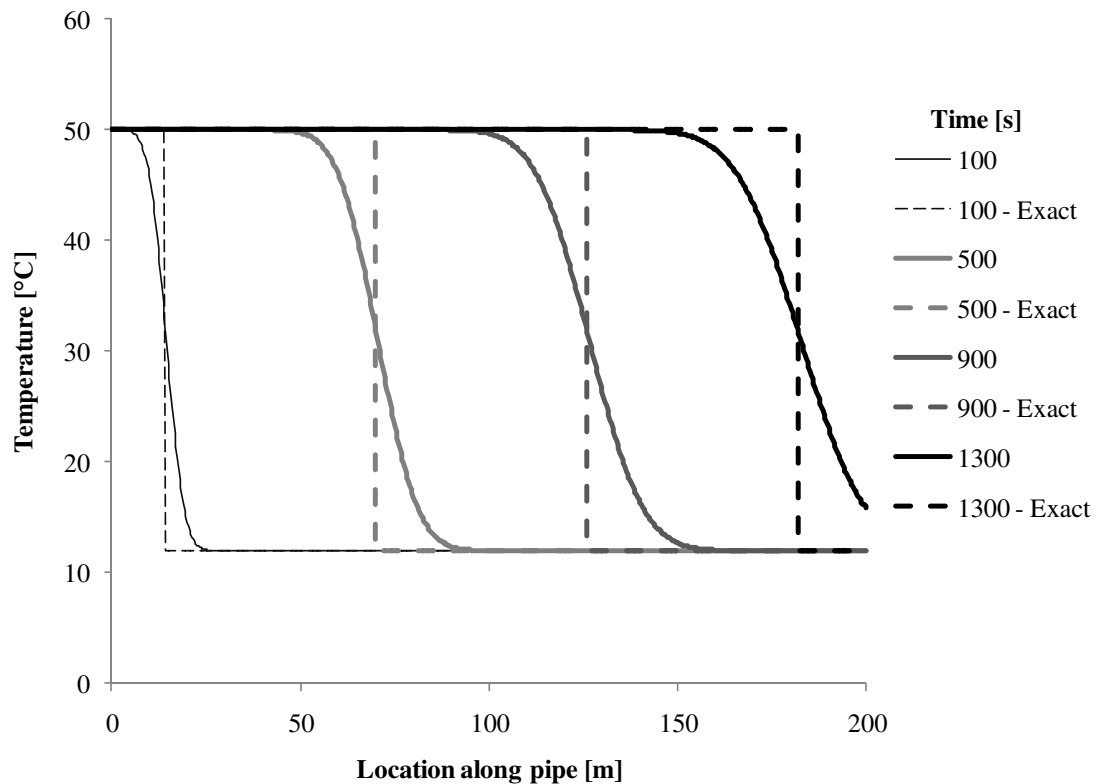
Figure 3.13   Transient simulation of convective heat flow responding to a step change in the inlet temperature from an initial colder ground temperature.

In order to examine the periodic response of the temperatures in the pipe, a model was examined with a periodically varying temperature at the inlet of the pipe.   The temperature in the surrounding pipe was assumed to be constant at a temperature of 12 °C.   This was also the initial temperature for both the ground and the pipe.   The inlet temperature, T_i, varied according to Equation (3.15) with a mean inlet temperature, T_{mean}, of 50 °C, an amplitude, T_{amp}, of 10 °C, a time period, t_p, of one hour, and with zero time delay, t_d.

$$T_i = T_{mean} + T_{amp} \cdot \sin\left(\frac{2 \cdot \pi \cdot (t - t_d)}{t_p}\right) \tag{3.15}$$

The general response of the temperature in the pipe to the varying inlet temperature is show in Figure 3.14.   The resulting temperature response at the outlet of the pipe formed a regular periodic function as shown in Figure 3.15.   There is a reduction in the mean temperature and amplitude as well as a time delay when compared to the original inlet temperature.   The time delay is 1430 s, the time it takes for water to move from the inlet to the outlet based on an average velocity of 0.14 m/s.
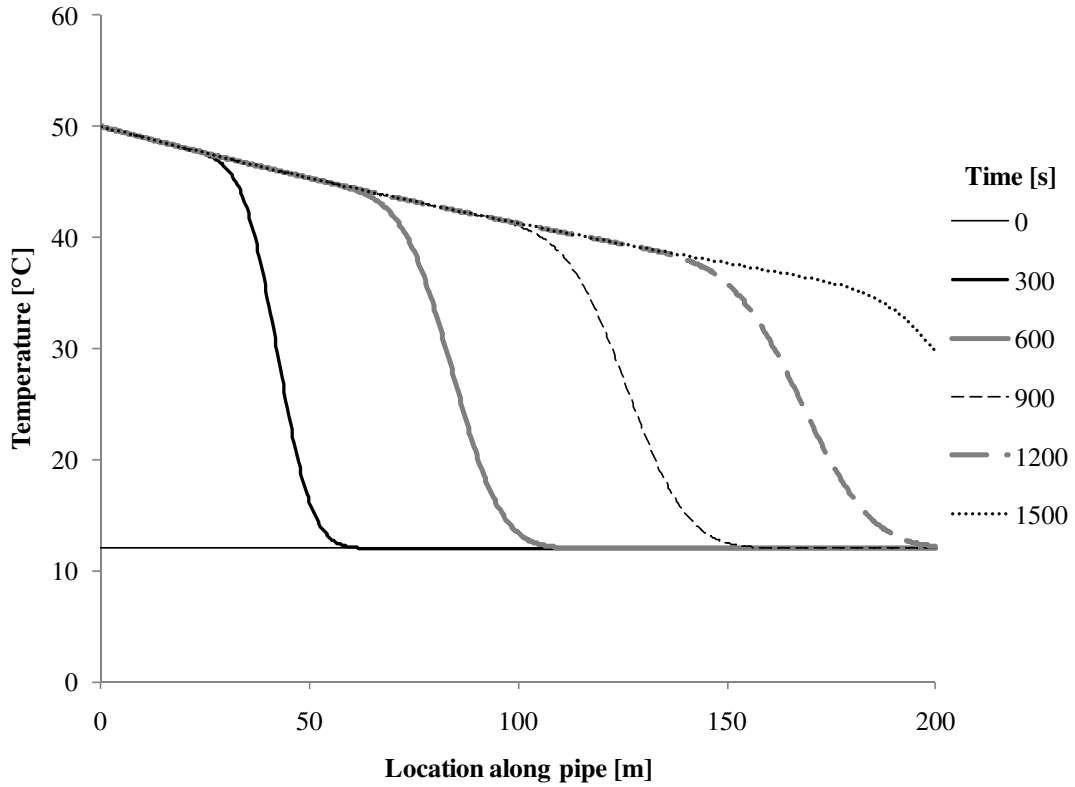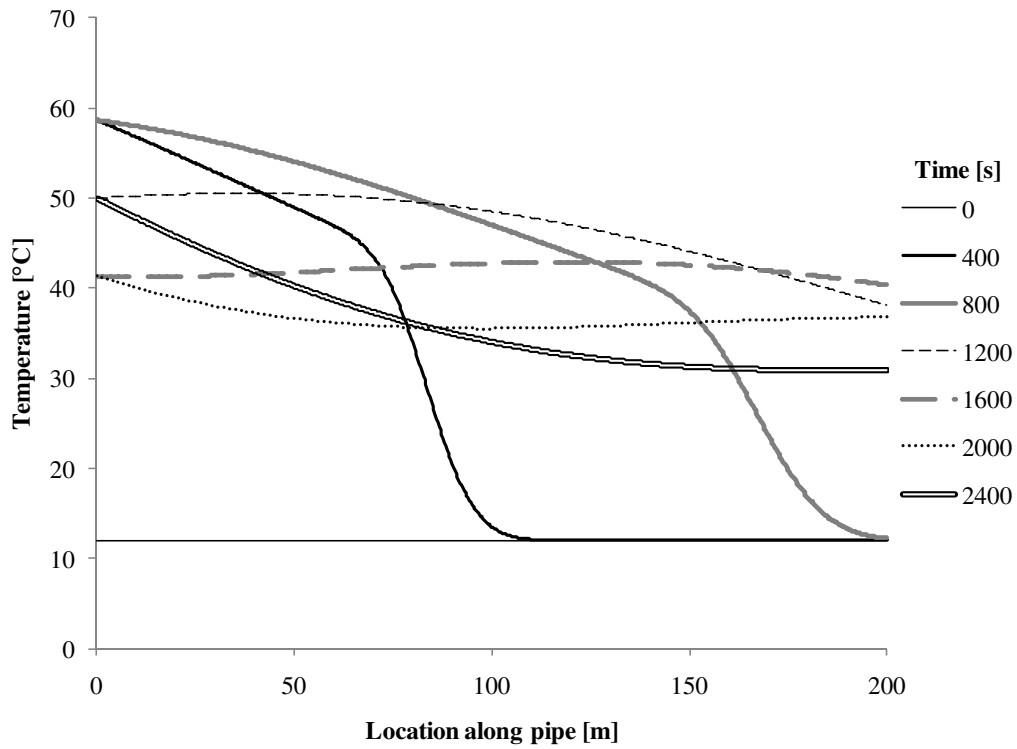
Figure 3.14   Periodic transient simulation of convective heat flow responding to a periodic inlet temperature from an initial colder ground temperature.
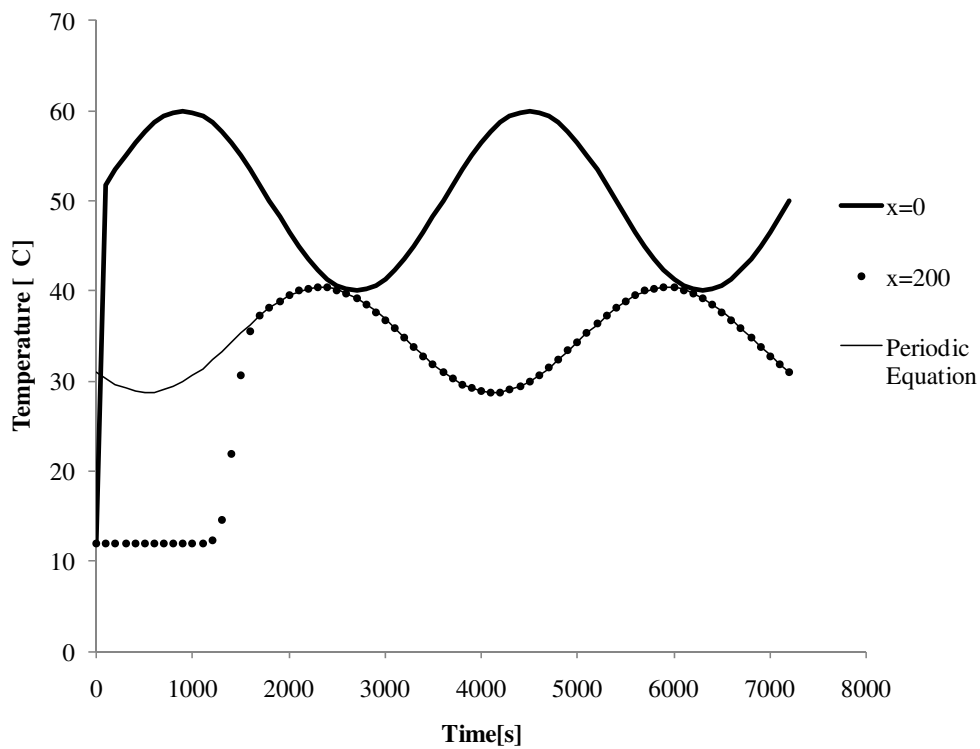
Figure 3.15   Time response of the temperature at the pipe boundaries subject to a periodically varying inlet temperature from an initial colder ground temperature.

3.3 Two dimensional conduction coupled to one dimensional convection

With separate verifications of conduction in the ground and convection in the pipe complete, coupling of the heat transfer was examined. The first coupled simulation, presented here, considers two dimensional conduction in the ground with one dimensional convection in the pipe. The development of a two dimensional model was achieved using linear extrusion coupling as described in Section 2.3. In this simulation steady state coupling was examined.

A simple pipe through a two dimensional domain was developed as shown in Figure 3.16. A separate one dimensional model of the pipe was also developed. It is shown in Figure 3.17. In order to couple the two models, an edge extrusion variable was first used in the model of the ground. This variable is mapped to a source, which is the line in the ground representing the pipe. The destination for this variable is then specified as the line in the pipe model. The corresponding extrusion variable for the temperature of the fluid in the pipe is mapped in the same way to the line in the ground. The heat transfer between the pipe and the ground was then modelled with the following equation.

$$q = L_0 \alpha_0 \big(T_0(x) - T(x) \big) \tag{3.16}$$

In COMSOL this heat transfer from the pipe to the ground is automatically calculated for each time step and for each element in the simulation.
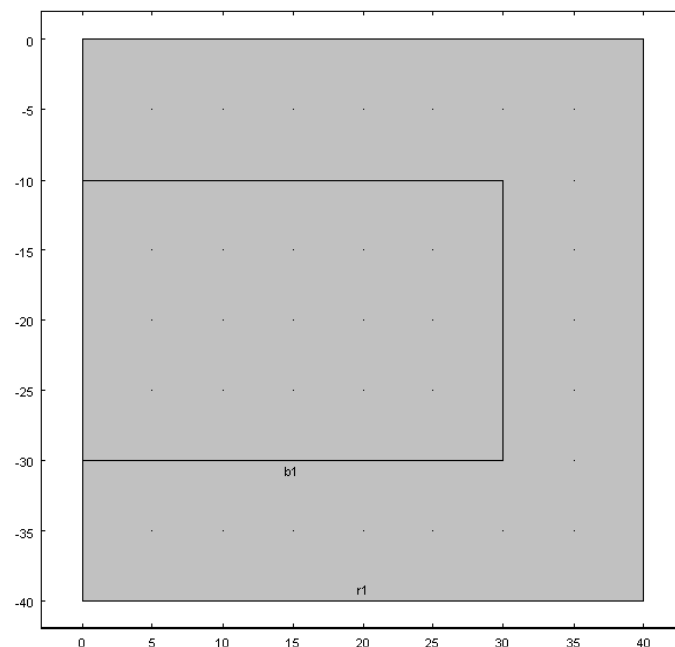
Figure 3.16   Model of the ground in a two dimensional domain.
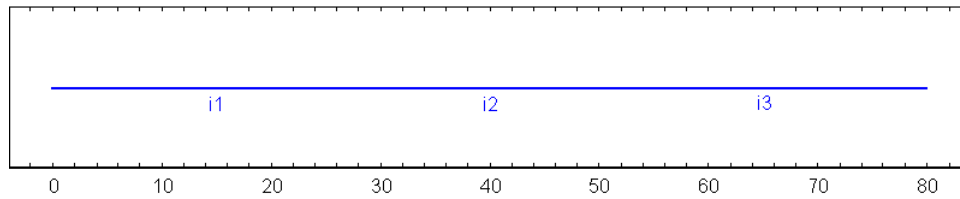
Figure 3.17   Model of the one dimensional pipe.

To compare the model to the analytical solution shown in Section 2.2.2, a model was developed where only heat was transferred from the ground to the pipe. This was done to get an identical result to previous solution. The results are shown in Figure 3.18. A full coupling of the heat transfer between the pipe and the ground was attempted in three dimensions as shown in the next section.
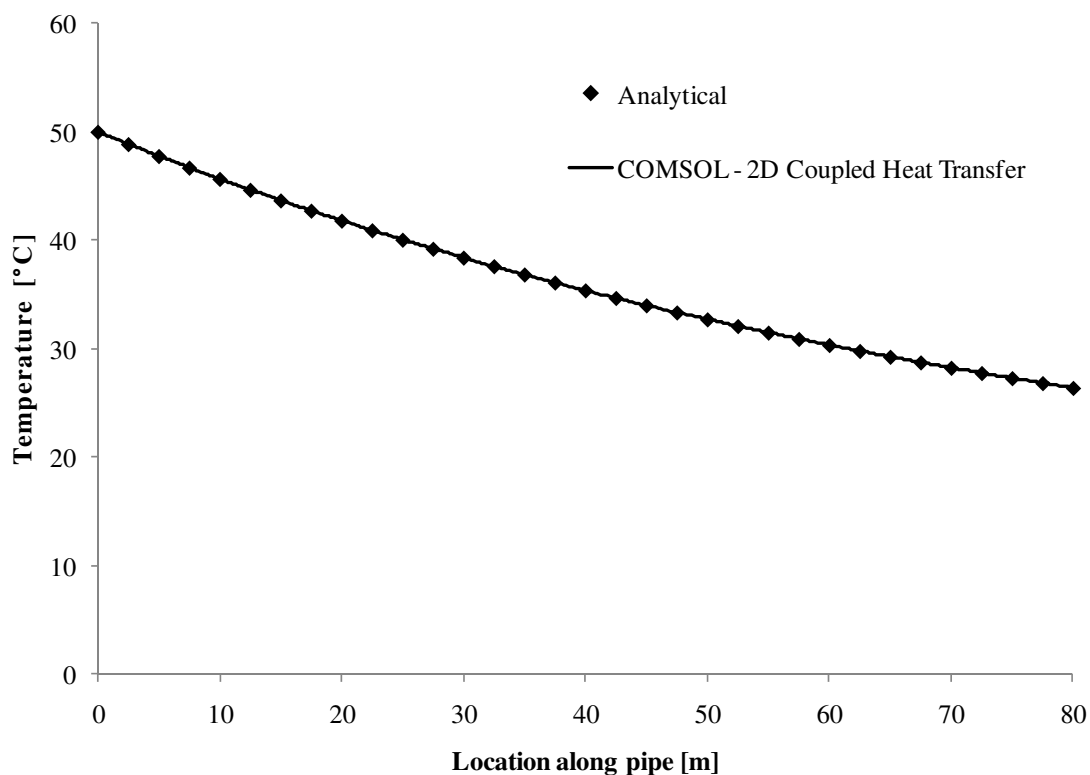
Figure 3.18   Comparison of an analytical solution of transverse heat loss in a pipe to a one dimensional pipe model coupled to a two dimensional ground model using COMSOL's Heat Transfer Interface.

3.4 Three dimensional conduction coupled to one dimensional convection

3.4.1   Fluid flow with a constant ground temperature

The first step in the development of a three dimensional model was to model a single horizontal pipe in a three dimensional region of ground located in the centre of the domain with constant boundary conditions. The model was first simulated in steady state to see impact and general performance. The temperature cross section

perpendicular to the pipe can then be compared to the temperature distributions obtained in Section 3.1.1 based on an assumption of a constant temperature cross section. The geometry of the pipe and the ground can be seen below in Figure 3.19.
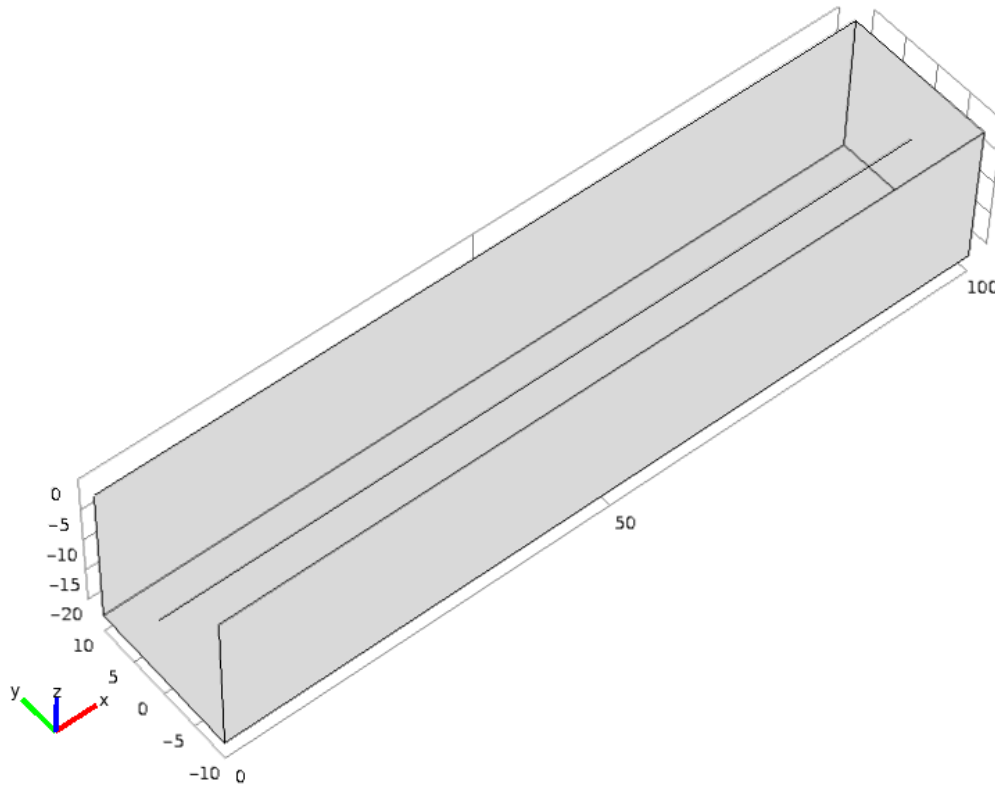
Figure 3.19   Geometry of the three dimensional simulation.

The results of the simulation, when compared to the analytical solution from Equation *(2.13)* are shown in Figure 3.20. The agreement between the simulation and the analytical solution, as in previous cases, is very good. This indicates that the one dimensional pipe flow model is able to simulate the heat transfer from the pipe to the ground with sufficient accuracy.

Figure 3.20   Comparison of an analytical solution of transverse heat loss in a pipe to a one dimensional pipe model coupled to a three dimensional ground model with constant ground temperature using COMSOL's Heat Transfer Interface.

3.4.2   Fluid flow with a coupled ground temperature

An attempt was made to validate the full coupling between the fluid flow in the pipe and the ground. This was accomplished by constructing a two dimensional axisymetric model in COMSOL using the heat transfer interface. An 800 m long pipe was simulated with the geometry as shown in Figure 3.. Water flowed along the pipe at a rate of 0.14 m/s. The flow turbulence was not modelled. The conduction of the water in the radial direction was assumed to be very high to model heat transfer towards the ground. The purpose of this validation was to verify the full coupling between the heat transfer in the pipe with the heat transfer in the ground. The geometry of the three dimensional model is shown in Figure 3.22.
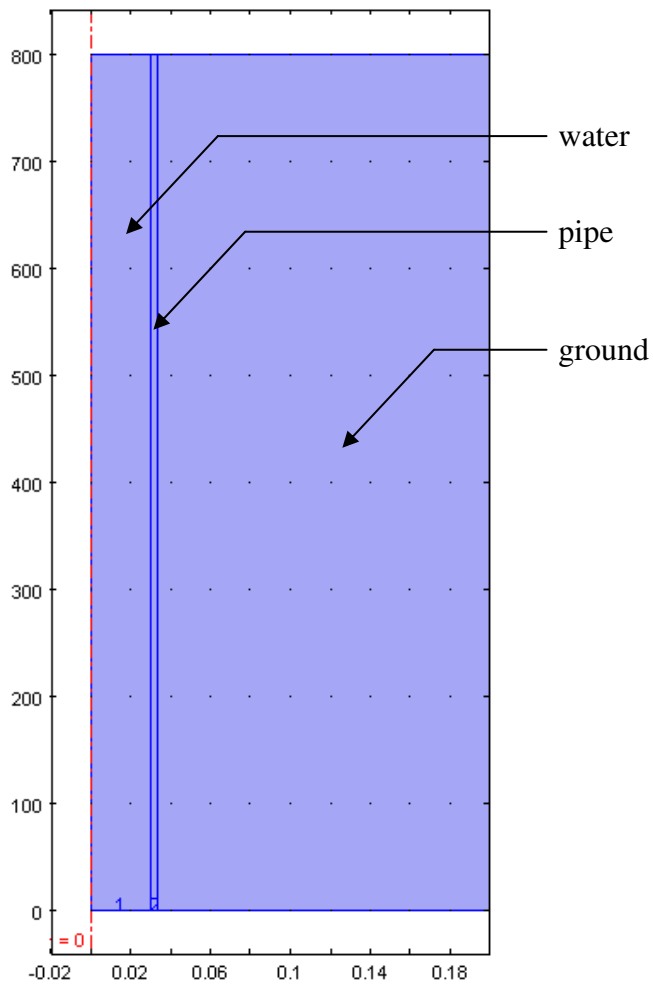
Figure 3.21   Geometry of the two dimensional axisymetric simulation.

Figure 3.22   Geometry of a three dimensional, fully coupled model with a pipe in the centre of the ground.

The results from the comparison between the two models are shown in Figure 3.23. There appears to be a reasonable agreement between the two models. It appears that

the heat transfer in the axisymmetric simulation is occurring at a somewhat faster rate than in the simplified model. This could be due to some of the assumptions made in calculating the heat transfer coefficient used in determining the quantity of transverse heat flow along the pipe. The coupled model gives an acceptable, but somewhat conservative, approximation of the heat transfer to the ground from the pipe.
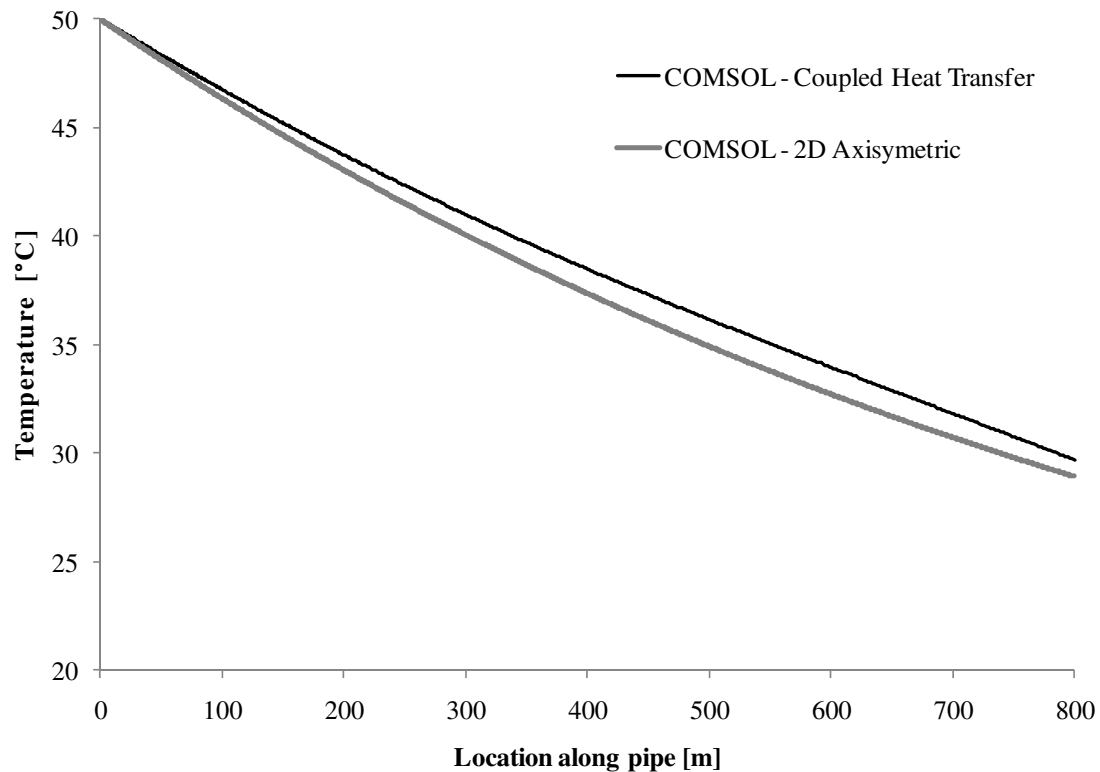
Figure 3.23   Comparison of an analytical solution of transverse heat loss in a pipe to a a two dimensional axisymetric model of the water, pipe, and ground.

3.4.3   Fluid flow in a pipe near ground surface

As an additional validation of the model, a simple model of a horizontal pipe laid in the ground was developed. This model consisted of an 800 m long pipe buried 1 m into the ground. The geometry of the model is shown in Figure 3.24.
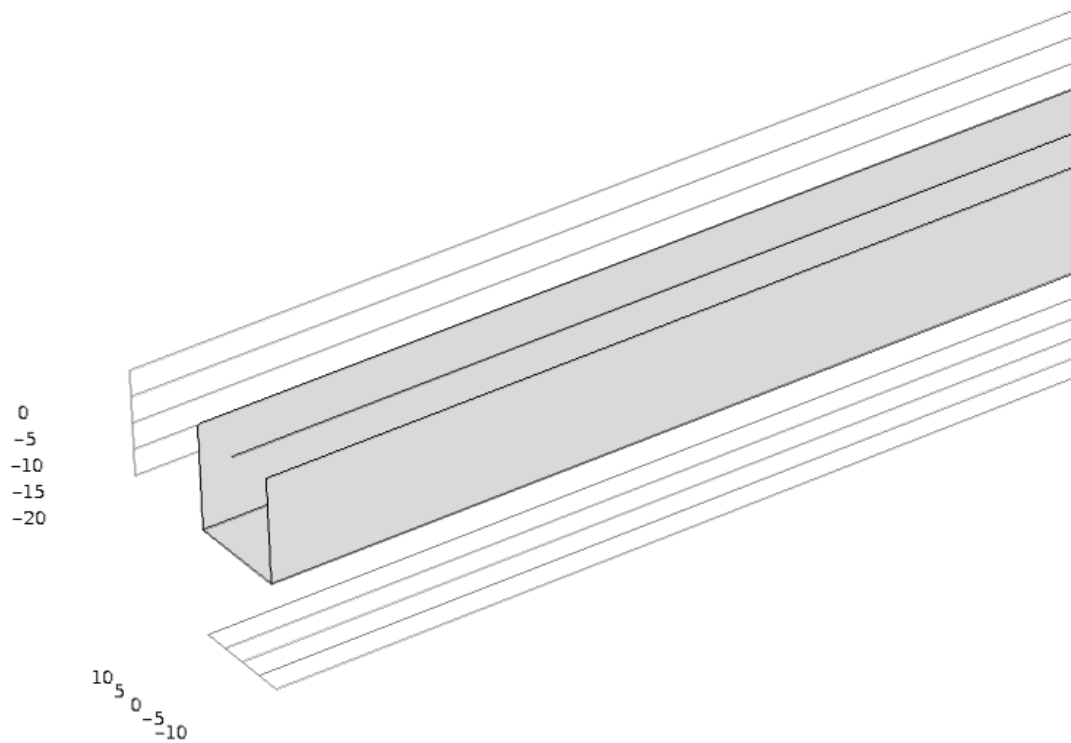
Figure 3.24   Geometry of a three dimensional, fully coupled model with a pipe at a depth of 1 m in the ground.

During this validation an analytical solution was compared to the numerical simulation results. The equation for the analytical solution was the same as used in Equation *(2.13)*. The only difference was in the calculation of α_0. An equivalent thermal resistance of the ground was added to heat transfer coefficient. The calculation of this was based on an equation for the resistance of a pipe buried in the ground (Claesson and Dunand, 1983). The additional transfer coefficient, α_g, is calculated with:

$$\alpha_g = \frac{\lambda}{R \cdot \ln\left(\frac{2 \cdot D}{R}\right)} \qquad (3.17)$$

With this additional transfer coefficient accounted for in the analytical solution a close match is developed to the COMSOL model. The result is shown in Figure 3.25
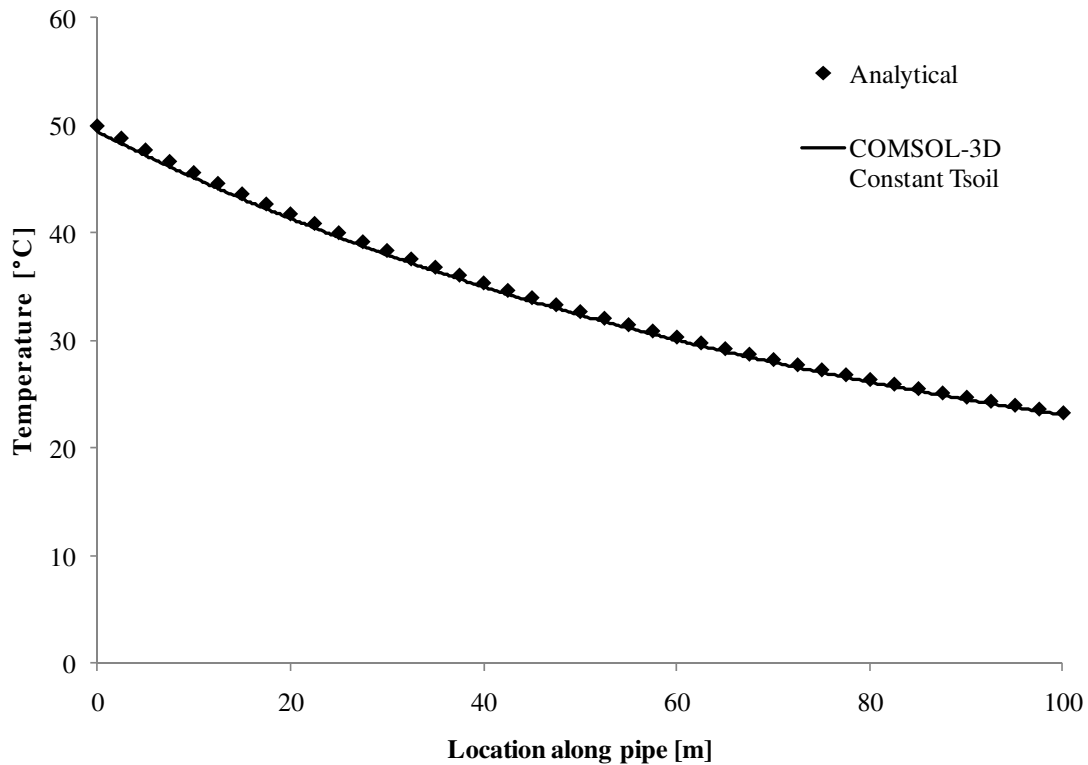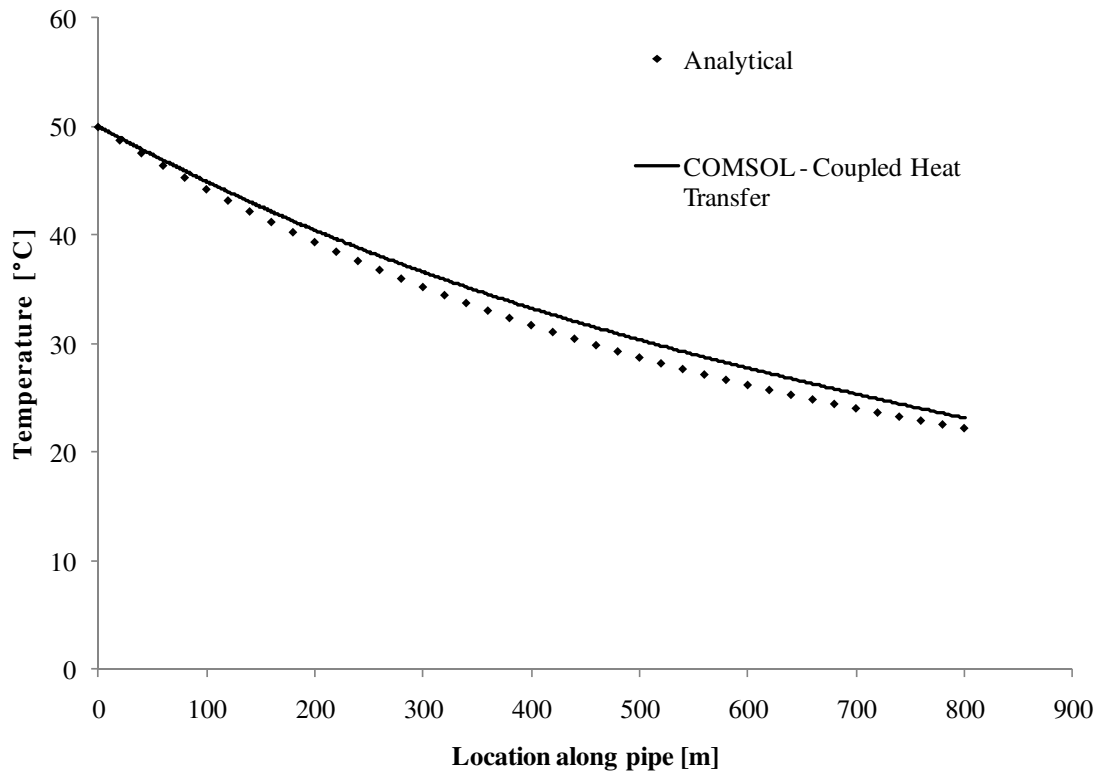
Figure 3.25 Comparison of an analytical solution of transverse heat loss in a pipe to a one dimensional pipe model coupled to a three dimensional ground model with a coupled ground temperature.

The above simulation was done along an 800 m long domain. It had a depth and width of 60 m. In order to assess the required domain size around the pipe a parametric study was conducted where the depth and width were varied from 20 to 80 m with a step size of 20 m. The results of this study are shown in Figure 3.26 and Figure 3.27. These show almost no effect on the temperature distribution in the pipe as the width and depth of the domain is increased above 20 m. Figure 3.26 shows that each of the domain sizes yielded identical results. Figure 3.27 shows the results from the 20 and 40 m size domain. The results for the 60 and 80 m size domain were the same as those from 40 m. Based on this result, future simulations will use a similar domain size.
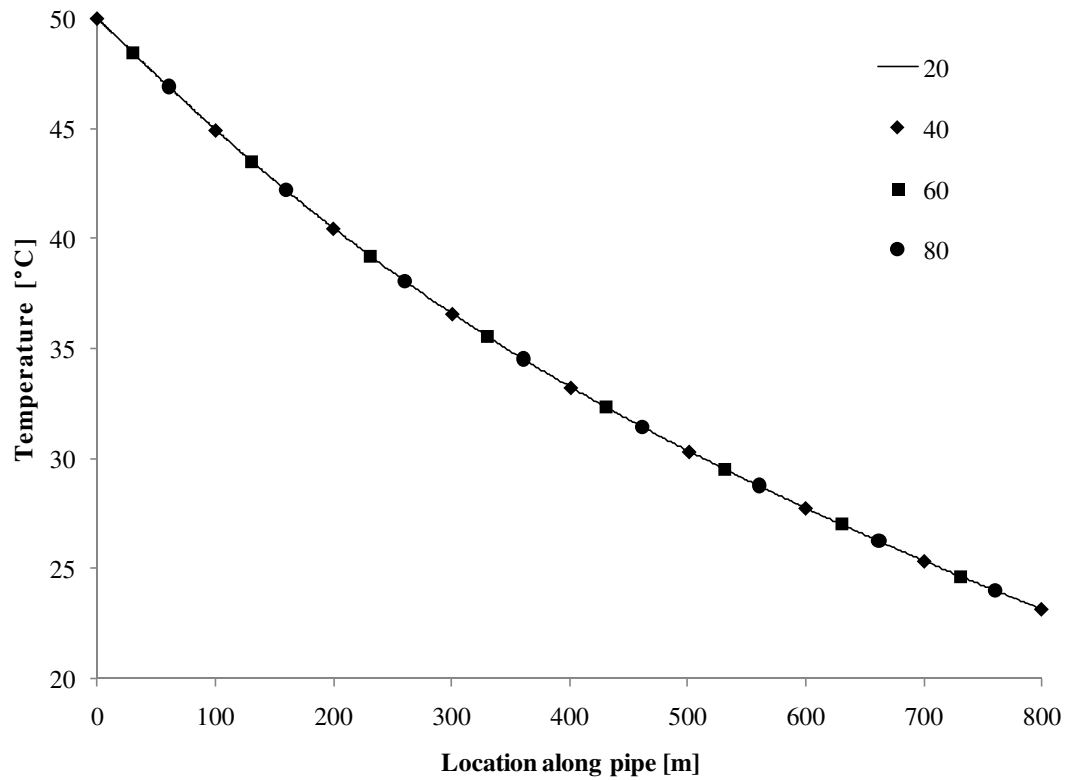
Figure 3.26 *Comparison of domain size in a one dimensional pipe model fully coupled to a three dimensional ground mode.*
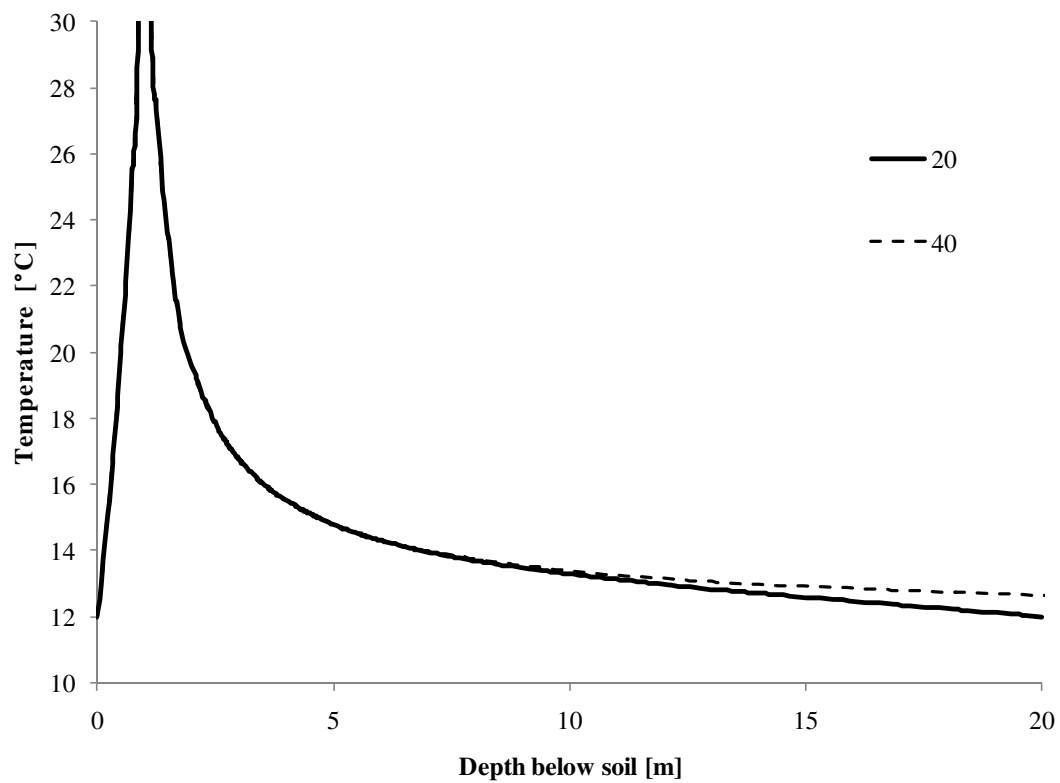
Figure 3.27 *Temperature distribution at x=10 m along the centre of the pipe with various domain sizes.*

3.4.4 Transient performance of the model

Two simple analyses were conducted to examine the transient performance of the pipe flow in the fully coupled ground. The analyses involved a step and periodical change in the inlet temperature of the pipe. The temperature at the boundaries of the ground was assumed to be constant.

For the step change analysis, the fluid in the pipe was initially at 12 °C. At time zero the water at the inlet was changed to 50 °C. The temperature response of the water in the pipe and the ground in the surrounding area was observed as the fluid flowed along the pipe. The results of the step change are shown in Figure 3.28. This shows how the temperature in the ground approaches the steady state shown in Figure 3.25. The pace of how quickly the temperature approaches the steady state solution depends on the density and thermal capacity of the ground.
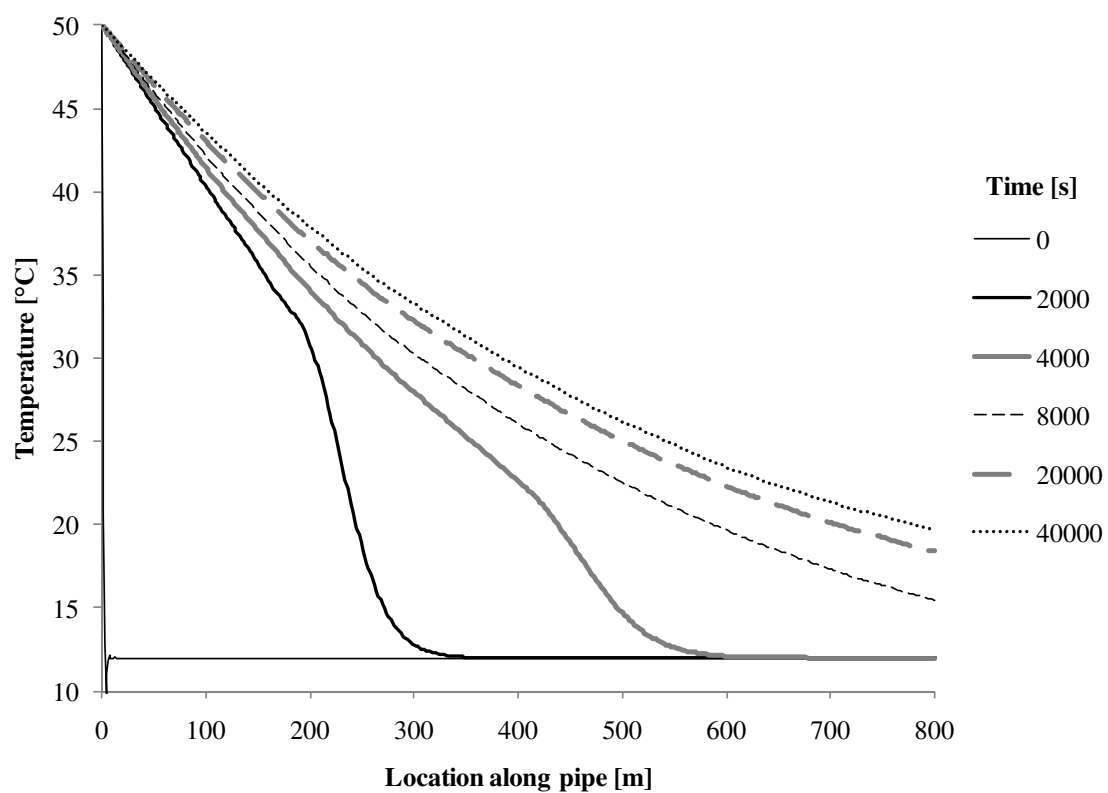
Figure 3.28   Transient simulation of convective heat flow responding to a step change in the inlet temperature from an initial colder ground temperature.

4   Models of Underground Heat Exchangers

Several models were completed in COMSOL of the underground heat exchangers. Models of both horizontal pipes and borehole exchangers have been developed. This chapter describes each of the models. This chapter also describes the process of converting the underground heat exchanger model into an S–function for use in Simulink. Finally, the response of the BHE model to a step changes in the inlet temperature of the fluid flowing through the system is also examined using the developed S–functions.

4.1   The COMSOL models

4.1.1   Single linear horizontal pipe

The first model created was of a single horizontal pipe in the ground. The pipe is modelled as a straight pipe at a constant depth in the ground. The model can also be used to simulate any horizontal pipe placed at a constant depth in the ground where curves or elbows in the pipe or additional pipes do not have a significant effect on each other. For example, an entire horizontal pipe loop could be simulated if segments of the loop do not thermally interact with other segments. The model could also be modified, however, to be used in other configurations of horizontal pipes.
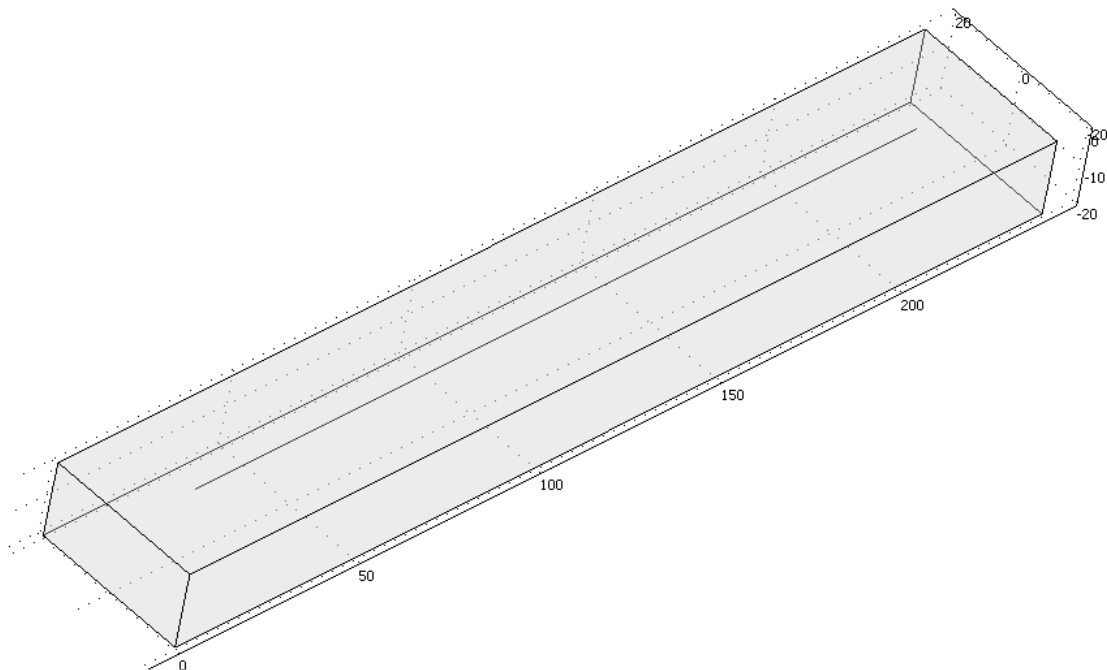
Figure 4.1   Final model of a horizontal underground heat exchanger.

The model allows the user to input a number of different properties and parameters as variables in the model. These values include the characteristics of the pipe, the initial temperatures in the domains, and the thermal characteristics of the ground. A full list of the properties and variables that the user can enter are shown in Table 4.1.

Table 4.1   List of global constants for horizontal underground heat exchanger.

Input Variable	Description	Units
L0	Inner circumference of the pipe	m
alpha0	Heat transfer coefficient from fluid to ground	$W/(m^2 \cdot K)$
Tgin	Initial ground temperature	°C
Ti	Initial inlet temperature of the pipe	°C
xarea	Cross sectional area of the pipe	°C
flow_velocity	Velocity of the fluid in the pipe	m/s
Tsoiltop	Initial exterior temperature at ground surface	°C
Tsoilbottom	Initial and long term temperature of ground	°C
k_ground	Conductivity of the ground	$W/(m \cdot K)$
rho_ground	Density of the ground	kg/m^3
cp_ground	Heat Capacity of the ground	$J/(kg \cdot K)$
k_fluid	Conductivity of the pipe fluid	$W/(m \cdot K)$
rho_fluid	Density of the pipe fluid	kg/m^3
cp_fluid	Heat Capacity of the pipe fluid	$J/(kg \cdot K)$

The model of the ground considers conduction through the ground using the thermal characteristics of k_ground, rho_ground, and cp_ground. The ground is assumed to be isotropic with a constant density and heat capacity throughout. The model could be modified, however, to consider variation in these properties within the ground to model different layers of soil and rock. The model, as mentioned previously, also assumes there is no diffusion of water within the ground.

Adiabatic boundary conditions are used along all of the boundaries of the ground except for the top and bottom. The space around the pipe domain was selected such that the adiabatic boundary conditions can accurately represent real situations. A distance of 20 m was chosen based on the results obtained from Section 3.4.3. The top and bottom boundary both have a temperature specified at the boundary. The variables Tsoiltop and Tsoilbottom are used to input the top and bottom temperature, respectively. The top temperature is the outdoor dry bulb temperature while the bottom temperature is the long term average temperature of the ground.

The model of the pipe consists simply of a one–dimensional line with a length, in this case, of 200 m as shown in Figure 4.2. The model of the pipe considers only convection in the pipe using the thermal characteristics of k_fluid, rho_fluid, and

cp_fluid. The velocity of the fluid in the pipe is assumed to be constant at a rate of flow_velocity. Since only convection is being simulated only one boundary condition must be specified. In this case the temperature at the inlet (x=0) is specified. The return temperature is calculated during the simulations.
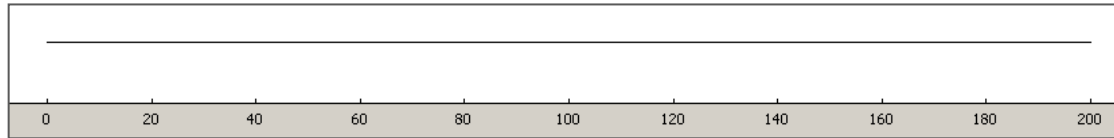
Figure 4.2 Linear model of a horizontal pipe.

4.1.2 Single loop BHE

The BHE is modelled using a three dimensional model of the ground with a cylindrical borehole in the centre. The pipe is modelled as two straight pipe segments running vertically through the borehole. One segment represents the flow in the pipe running down to the bottom and the other segment represents the fluid returning. Figure 4.3 shows the model of the ground with the borehole.
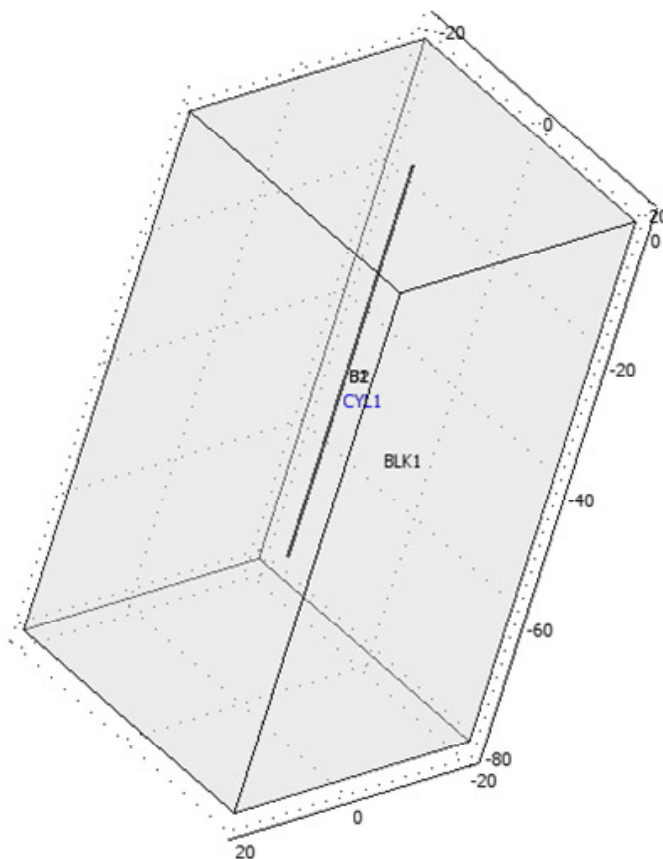
Figure 4.3 Model of the ground and the single loop BHE.

This model, as with the horizontal pipe model, allows the user to input a number of different properties and parameters as variables in the model. The values input are the same as for the horizontal pipe, with the addition of specifying the characteristics of the grout or filler material used within the borehole. A full list of the properties and

variables that the user can enter are shown in Table 4.2. The boundary conditions of the ground are identical to those described in the section above.

Table 4.2 List of global constants for the BHE models.

Input Variable	Description	Units
L0	Inner circumference of the pipe	m
alpha0	Heat transfer coefficient from fluid to ground	$W/(m^2 \cdot K)$
Tgin	Initial ground temperature	°C
Ti	Initial inlet temperature of the pipe	°C
xarea	Cross sectional area of the pipe	°C
flow_velocity	Velocity of the fluid in the pipe	m/s
Tsoiltop	Initial exterior temperature at ground surface	°C
Tsoilbottom	Initial and long term temperature of ground	°C
k_ground	Conductivity of the ground	$W/(m \cdot K)$
rho_ground	Density of the ground	kg/m^3
cp_ground	Heat capacity of the ground	$J/(kg \cdot K)$
k_grout	Conductivity of the grout	$W/(m \cdot K)$
rho_grout	Density of the grout	kg/m^3
cp_grout	Heat Capacity of the grout	$J/(kg \cdot K)$
k_fluid	Conductivity of the pipe fluid	$W/(m \cdot K)$
rho_fluid	Density of the pipe fluid	kg/m^3
cp_fluid	Heat Capacity of the pipe fluid	$J/(kg \cdot K)$

4.1.3   Models of multiple single loop BHEs

Several COMSOL models with multiple boreholes were developed to verify the performance of more complex models. For example, a model with four boreholes was developed as shown in Figure 4.4. This model was developed in the same way as the single U–pipe model. There is, however, a separate model for each pipe. In this study they are assumed to be operating in parallel.
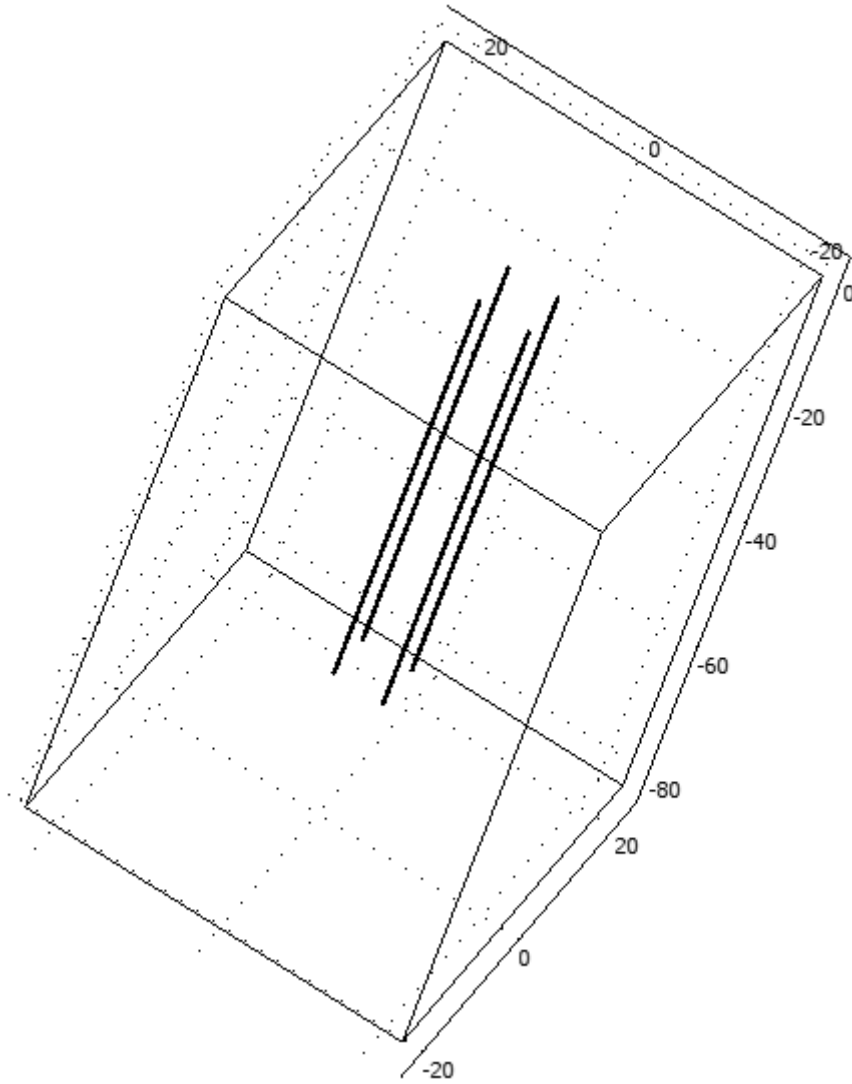
Figure 4.4   Model of the ground and four BHE systems.

An infinite array of BHE can also be modelled. The model is nearly identical to the single BHE shown in the previous section. The only difference is the spacing around the borehole. The boundaries of the simulation will be at the midpoint between the adjacent boreholes. For example, consider an array of boreholes, similar to what is shown in Figure 4.5, have a spacing of x m between them. In this case the same model of a single BHE, shown in the previous section, can be used. The only difference is the space around the borehole in the x and y direction would be equal to half the distance between the boreholes.
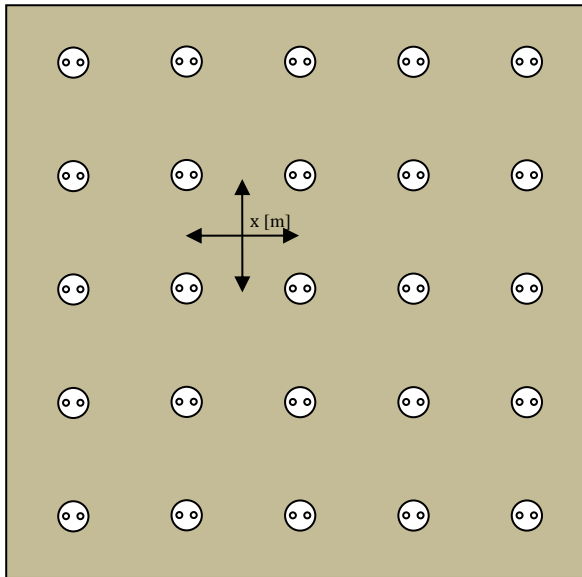
Figure 4.5   Top view of a sample array of BHEs.

4.2   Simulink and S–Functions

In order to simulate complex building models, the Simulink program was used. Simulink is a program for the simulation of time varying and embedded systems. It provides an interactive graphical environment that is integrated into Matlab. The graphical environment allows development of models using something similar to flow diagrams.

In Simulink S–Functions can be used to interact with complex functions written in various programming environments. In the Matlab documentation an S–Function is described as:

> *S–functions (system–functions) provide a powerful mechanism for extending the capabilities of the Simulink environment. An S–function is a computer language description of a Simulink block written in MATLAB®, C, C++, or Fortran.*

In these simulations Level 2 S–functions were used. Level 2 S–functions allow functions to be written in MATLAB that can interact with the Simulink environment in a highly customizable fashion.

Using these S–functions, COMSOL simulations can be integrated into a Simulink block. In COMSOL, there is an interface for incorporating simulations into Matlab functions. The COMSOL models are simply saved as a model m–file which allows them to be run in Matlab.

4.2.1   A simple embedded model using a S–function

A simple model was developed in COMSOL, converted to an m–file, and integrated into an S–function. This S–function was then used in a Simulink model which is shown in Figure 4.6. This model was developed simply for the purpose of testing the functionality of Simulink and COMSOL integration.
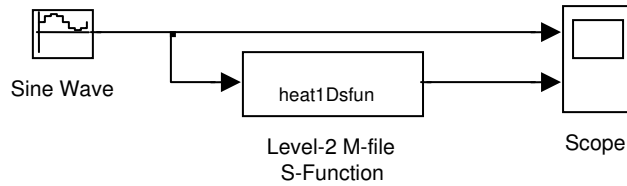
Figure 4.6   Simple Simulink model including an embedded S–function that invokes a COMSOL model.

The model created in COMSOL was a simple model of one dimensional heat flow. After being developed, it was then saved as a model m–file. This model m–file is written in Matlab script and it can be directly run using MATLAB. The S–function, heat1Dsfun, was developed from this m–file in the required format. The code for the function, which shows the form of the S–function, is shown in Figure 4.6. This code shows the required function blocks that are needed to create a Level–2 m–file. It includes the functions Setup, Update, and Output. The purpose of each of these functions in the context of running a transient COMSOL simulation is as follows:

- **Setup**. This function initiates the setup of the S–function, including the code for the creation of the COMSOL model.
- **Update**. This function updates the COMSOL model during each time–step in the Simulink simulation. For example, any new boundary conditions are applied to the model and the simulation is run until the next time step.
- **Output**. In this function, any desired outputs from the COMSOL simulation are output to the Simulink simulation.

Full details of the Level–2 S–functions can be found in the MATLAB documentation (MathWorks, Inc., 1984). The full code for the setup and updating of the heat1Dsfun simulation can be found in Appendix A.

```matlab
function heat1Dsfun(block)
% Level-2 M file S-Function demonstrating integration of a COMSOL model.
setup(block);
%endfunction

function setup(block)
  %% Register number of input and output ports
  block.NumInputPorts  = 1;
  block.NumOutputPorts = 1;
  %% Setup functional port properties to dynamically
  %% inherited.
  block.SetPreCompInpPortInfoToDynamic;
  block.SetPreCompOutPortInfoToDynamic;
  block.InputPort(1).DirectFeedthrough = true;
  %% Set block sample time to inherited
  block.SampleTimes = [-1 0];
  %% Set the block simStateCompliance to default (i.e., same as a built-in block)
  block.SimStateCompliance = 'DefaultSimState';
  %% Run accelerator on TLC
  block.SetAccelRunOnTLC(false);
  %% Register methods
  block.RegBlockMethod('PostPropagationSetup',    @DoPostPropSetup);
  block.RegBlockMethod('Outputs',                 @Output);
  block.RegBlockMethod('Update',                  @Update);
  %%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
  global fem
  global tout
  global solut
  %%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
  %  Initialize COMSOL Simulations  %%%
  %%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%

function Update(block)
  %%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
  %  Update COMSOL Simulations  %%%
  %%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%endfunction

function Output(block)
    global tout
    block.OutputPort(1).Data = tout;
%endfunction
```

Figure 4.7   Matlab S–function showing function block set–up. (Note that the code to initialize and update the COMSOL simulation has been removed).

4.3   S–Functions of underground heat exchangers

The procedure and format, as described in the previous section was used to integrate full COMSOL models of underground heat exchangers into Simulink models of buildings. This section describes each of the S–functions that were developed.

4.3.1   Single linear horizontal pipe

A Level 2 S–function of the horizontal pipe model was developed similar to the simple S–function developed in the previous section. The model developed in section 4.1.1 was saved as a model m–file and then programmed as part of a dynamic S–function. The function consists of the main model m–file, 'SinglePipeHoriz_sfun.m', along with a separate file containing a list of constants and parameters. This file is named 'SinglePipeHoriz_constants.m'. It contains details such as the characteristics of the pipe, the thermal properties of all the materials, parameters relating to the size of the mesh, and an option to output the temperature in the ground to a separate file. The full list of constants and parameters contained in this file are shown in Table 4.3.

Table 4.3   List   of   static   parameters   and   simulation   variables   in
SinglePipeHoriz_constants.m file.

Input Variable	Description	Units
Static Parameters		
horiz_pipe.pipelength	Length of pipe	m
horiz_pipe.pipedepth	Depth of pipe from ground surface	m
horiz_pipe.soilspace	Space of ground around the pipe in all horizontal directions and in depth	m
Simulation Variables		
horiz_pipe.L0	Inner circumference of the pipe	m
horiz_pipe.alpha0	Heat transfer coefficient from fluid to ground	$W/(m^2 \cdot K)$
horiz_pipe.Tgin	Initial ground temperature	°C
horiz_pipe.Ti	Initial inlet temperature of the pipe	°C
horiz_pipe.xarea	Cross sectional area of the pipe	°C
horiz_pipe.flow_velocity	Velocity of the fluid in the pipe	m/s
horiz_pipe.Tsoiltop	Initial exterior temperature at ground surface	°C
horiz_pipe.Tsoilbottom	Initial and long term temperature of ground	°C
horiz_pipe.k_ground	Conductivity of the ground	$W/(m \cdot K)$
horiz_pipe.rho_ground	Density of the ground	kg/m^3
horiz_pipe.cp_ground	Heat capacity of the ground	$J/(kg \cdot K)$
horiz_pipe.k_grout	Conductivity of the grout	$W/(m \cdot K)$
horiz_pipe.rho_grout	Density of the grout	kg/m^3
horiz_pipe.cp_grout	Heat Capacity of the grout	$J/(kg \cdot K)$
horiz_pipe.k_fluid	Conductivity of the pipe fluid	$W/(m \cdot K)$
horiz_pipe.rho_fluid	Density of the pipe fluid	kg/m^3
horiz_pipe.cp_fluid	Heat capacity of the pipe fluid	$J/(kg \cdot K)$

Table 4.4   List of mesh and output variables in SinglePipeHoriz_constants.m file.

Input Variable	Description	Units
Mesh Variables		
horiz_pipe.mesh_geom1_hauto	Auto meshing scale (1= very fine to 8=very coarse)	–
horiz_pipe.mesh_geom2_hmax	maximum size of elements	m
Output Variable		
horiz_pipe.outputdetails	include output of ground temperatures (1=true)	–

There are three dynamic inputs to the S–function to control the operation of the heat exchanger at each time step during the simulation. The three inputs are listed in Table 4.5. These dynamic inputs control the flow of the fluid through the pipe and the inlet temperature of the fluid entering it. The ground surface temperature is specified based on the outdoor weather conditions. This can be specified based on the available climate data. Neither solar radiation nor wind characteristics are included in the calculations. The impact of these variables is assumed to be negligible for the overall performance of the pipe.

Table 4.5   List of dynamic inputs to the single horizontal pipe S–function.

Input Number	Description	Units
1	Pipe inlet temperature	°C
2	Ground surface temperature	°C
3	Velocity of the fluid in the pipe	m/s

The completed S–function can now be used in a Simulink model. A simple model showing the three inputs can be seen in Figure 4.8. This model simply simulates the performance of the horizontal pipe in the ground based on the three inputs.
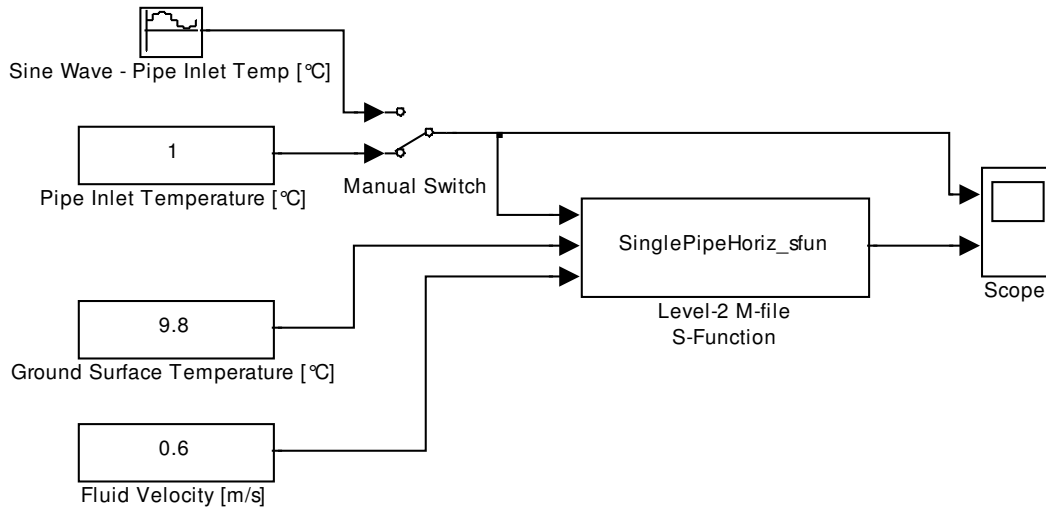
Figure 4.8   Simulink model including an embedded S–Function of the horizontal pipe model.

4.3.2   Models of BHEs

Several models of various configurations of BHEs were also developed. These models were developed in order to be able to quantify the performance of several different types of BHEs. The S–functions that were created are shown in Table 4.6.

Table 4.6   List of S–functions created for modeling BHEs.

S–function filename	Type of System Modelled	Constants File
InfSingleUpipe_sfun.m	Single or infinite array of boreholes	InfSingleUpipe_constants.m
Upipe_124_sfun.m	Model of one, two, or four boreholes	Upipe_124_constants.m
Upipe_4816_sfun.m	Model of four, eight, or sixteen boreholes	Upipe_4816_constants.m

Each of the three S–functions is linked to a file containing its relevant constants and parameters as listed in Table 4.6. Similar to the file for the horizontal pipe, the constants file contains details such as the characteristics of the pipe and the borehole, the thermal properties of all the materials, parameters relating to the size of the mesh, and an option to output the temperature in the ground to a separate file. The full list of constants and parameters contained in this file are shown in Table 4.7. Note that this file contains the constants that are common to all three of the S–functions. Constants that only apply to one or two of the functions are indicated in the table.

Table 4.7   *List of static parameters, mesh variables and the output variable in the*
            BHE constant files.

Input Variable	Description	Units
Static Parameters		
U_pipe.pipelength	Length of pipe	m
U_pipe.pipedepth	Depth of pipe from ground surface	m
U_pipe.soilspace	Space of ground around the pipe in all horizontal directions and in depth	m
U_pipe.boreholediameter	Diameter of the borehole	m
U_pipe.relativepipelocation	Relative location of the pipes in the borehole compared to the width of the borehole	–
U_pipe.TsoilinitialdegC	Initial ground temperature	°C
U_pipe. Boreholespace[1]	Spacing between the boreholes	m
U_pipe.nborehole[2]	Number of boreholes to simulate	–
Mesh Variables		
U_pipe.mesh_geom1_hauto	Auto meshing scale	–
U_pipe.mesh_geom2_hmax	Maximum size of elements	m
Output Variable		
U_pipe.outputdetails	Include output of ground temperatures (1=true)	–

Notes:

1. Boreholespace. This applies to all models. If one single borehole is to be simulated with the InfSingleUPipe model then this value should be equal to the pipespace value.
2. Number of Pipes. Only applies to Upipe148 and Upipe4816 models. In the Upipe148 only 1,4 and 8 are applicable values and in Upipe4816 only 4,8, and 16 are applicable.

Table 4.8   List of simulation variables in the BHE constant files.

Input Variable	Description	Units
Simulation Variables		
U_pipe.L0	Inner circumference of the pipe	m
U_pipe.alpha0	Heat transfer coefficient from fluid to ground	$W/(m^2 \cdot K)$
U_pipe.Tgin	Initial ground temperature	°C
U_pipe.Ti	Initial inlet temperature of the pipe	°C
U_pipe.xarea	Cross sectional area of the pipe	°C
U_pipe.flow_velocity	Velocity of the fluid in the pipe	m/s
U_pipe.Tsoiltop	Initial exterior temperature at ground surface	°C
U_pipe.Tsoilbottom	Initial and long term temperature of ground	°C
U_pipe.k_ground	Conductivity of the ground	$W/(m \cdot K)$
U_pipe.rho_ground	Density of the ground	kg/m^3
U_pipe.cp_ground	Heat Capacity of the ground	$J/(kg \cdot K)$
U_pipe.k_grout	Conductivity of the grout	$W/(m \cdot K)$
U_pipe.rho_grout	Density of the grout	kg/m^3
U_pipe.cp_grout	Heat Capacity of the grout	$J/(kg \cdot K)$
U_pipe.k_fluid	Conductivity of the pipe fluid	$W/(m \cdot K)$
U_pipe.rho_fluid	Density of the pipe fluid	kg/m^3
U_pipe.cp_fluid	Heat Capacity of the pipe fluid	$J/(kg \cdot K)$

As with the single horizontal pipe, there are three dynamic inputs to the S–function which control the operation of the heat exchanger at each time step during the simulation. The three inputs are the same as those for the horizontal pipe and are listed in Table 4.5.

The completed S–function can now be used in a Simulink model. A simple model showing the three inputs can be seen in Figure 4.9. This model simply simulates the performance of the horizontal pipe in the ground based on the three inputs.
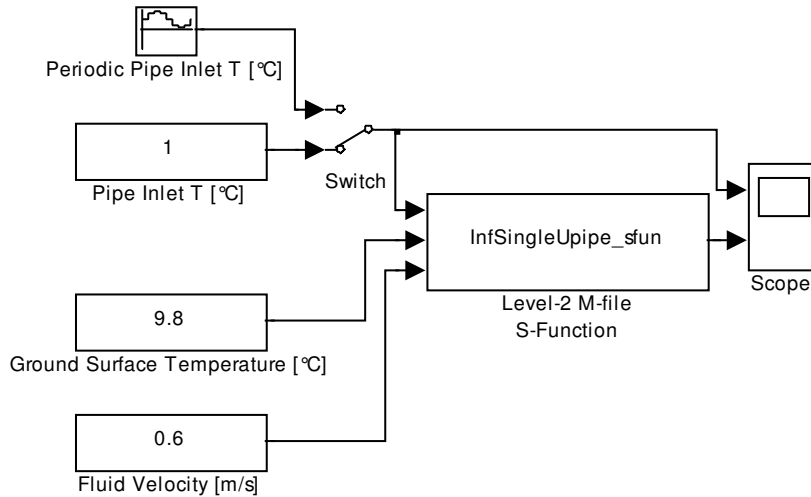
Figure 4.9   Simulink model including an embedded S–function of the BHE model.

4.4   Step response of BHEs

In order to implement a comparison between the different BHEs, it was decided to compare the performance of each of the models in response to a step change in temperature. This was done to obtain a simplified response of the heat exchangers as well as verify their expected performance. The simulations were conducted using the borehole S–functions 'InfSingleUpipe_sfun.m', 'Upipe_124_sfun.m', and 'Upipe_4816_sfun.m'. The tested borehole configurations are shown in Figure 4.10 and listed in Table 4.9.

Each simulation consisted of a ground domain, boreholes, and pipes at an initial constant temperature of 8 °C. At time zero, the flow through each of the boreholes begins at a velocity of 0.6 m/s at an inlet temperature of 4°C. In order to compare the step response between the pipes, the return temperature in the water has been recorded for one year. In cases with multiple boreholes, the flow through each borehole was assumed to be in parallel.

Figure 4.10   Top view of BHEs tested with a step change.

Table 4.9   List of boreholes simulated with a step change.

Number of pipes	Number of boreholes x–dir	y–dir	Simulated ground space [m]	Borehole spacing [m]
1	1	1	40	–
2	2	1	40	4
4	2	2	40	4
8	4	2	40	4
16	4	4	40	4
Infinite	∞	∞	–	4

Following the simulations, the outlet temperature of each of the BHEs was graphed over a period of one year. This is shown in Figure 4.11. In cases with multiple pipes, the average outlet temperature is shown. During the simulation, as the cool fluid continually flowed through the BHE, the temperature of the ground decreased. Since the ground temperature decreased over time, the return temperature from the BHE also decreased over time as shown in the figure. As expected, the reduction in pipe temperature is greater when there are more pipes in the ground.

The outlet temperature of each of the pipes after one year is shown in Figure 4.12. This figure shows the changes in outlet temperature based on the where a BHE is relative to others in given system. As expected, the reduction in pipe temperature is

greater when a BHE is in the core of a system and surrounded by other pipes. The temperature reduction is the greatest for corner BHE.
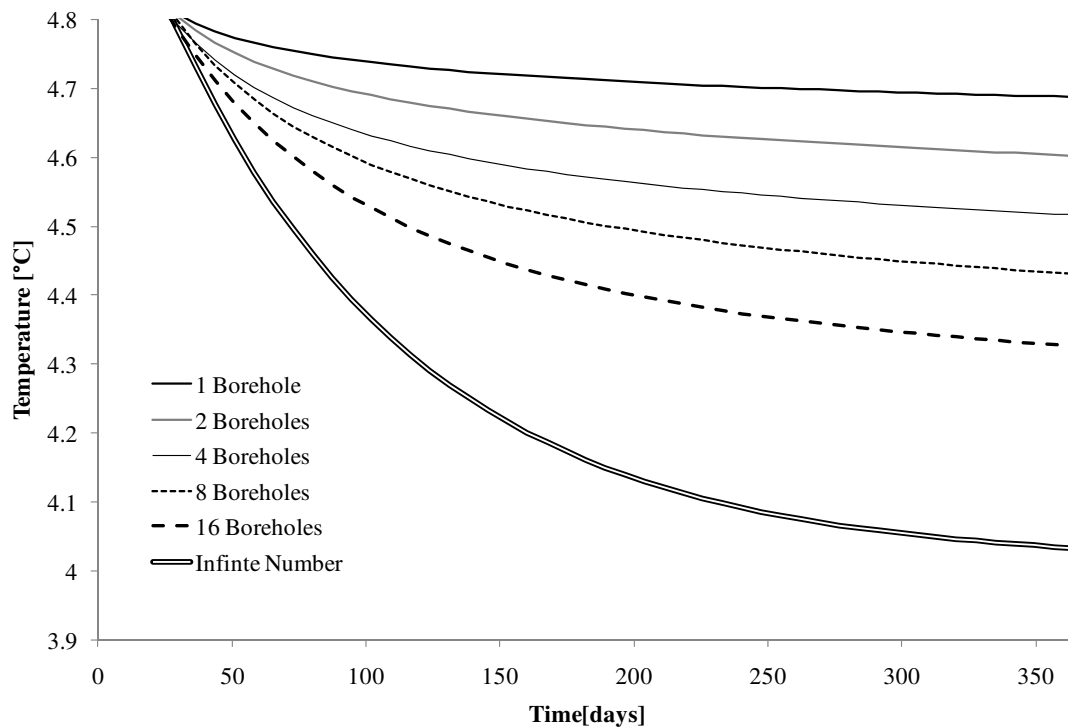
Figure 4.11   Outlet temperature of a BHE in response to a step change in inlet temperature.
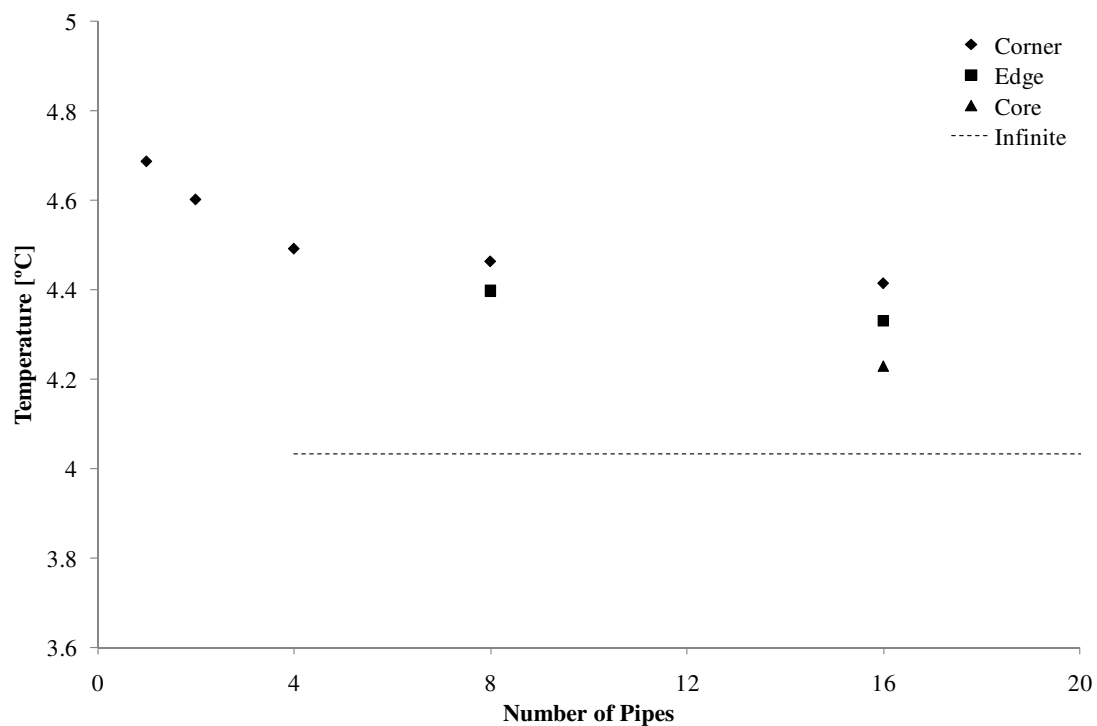
Figure 4.12   Outlet temperature after one year of BHEs in response to a step change in inlet temperature.

4.5 Heating and cooling step response of BHEs

A second step change simulation was conducted to compare the storage capability of different borehole configurations. The simulations were conducted using the same S–functions as in the previous sections. The tested borehole configurations were limited to those listed in Table 4.10.

Each simulation consisted of a ground domain, boreholes, and pipes at an initial constant temperature of 8 °C. At time zero, the flow through each of the boreholes begins at a velocity of 0.6 m/s at an inlet temperature of 4°C. This condition remained for a period of one half of a year (26 weeks). At this point the inlet temperature of the pipes went through a step change to a temperature of 12°C. In order to compare the step response between the pipes, the return temperature in the water has been recorded for one year. The temperature in the ground was also monitored. In cases with multiple boreholes, the flow through each borehole was assumed to be in parallel.
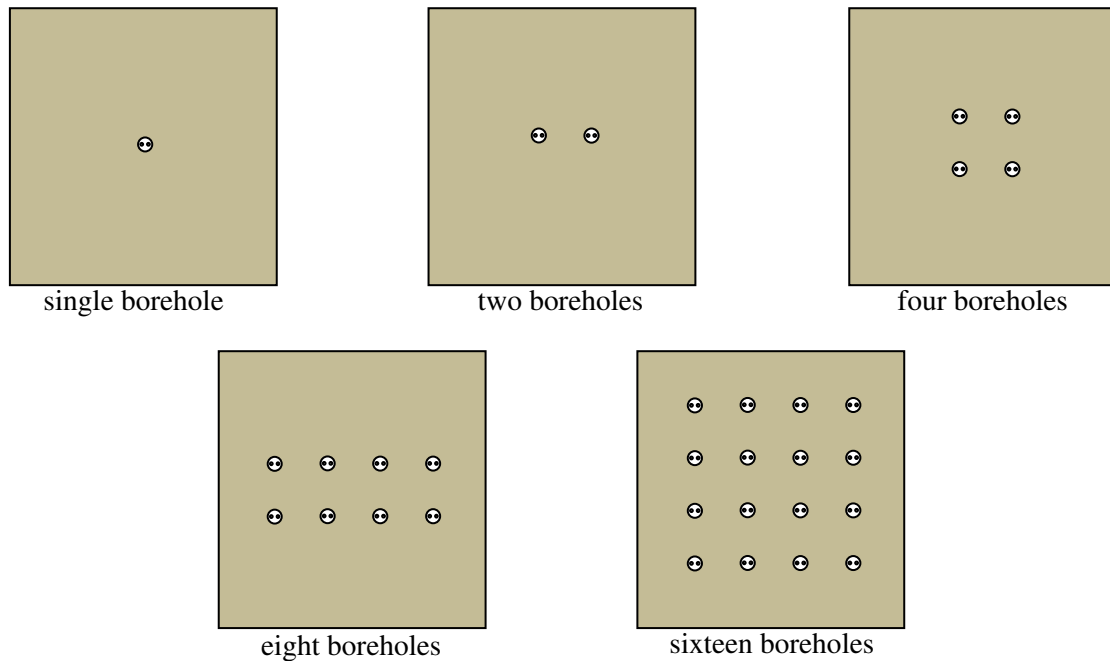
Table 4.10   List of boreholes simulated with two step changes.

Number of pipes	Number of boreholes x–dir	y–dir	Simulated ground space [m]	Borehole spacing [m]
1	1	1	40	–
4	2	2	40	4
16	4	4	40	4
Infinite	∞	∞	–	4

The outlet temperature of each of the pipes was graphed over a period of one year as shown in Figure 4.13. In cases with multiple pipes, the average outlet temperature is shown. During the initial half of the simulation, as the cool fluid flowed through the BHE, the temperature of the ground decreased in the same pattern as for the single step change. When the inlet temperature increased to 12 °C, configurations with a lower number of boreholes quickly stabilized to a slower steady decline. The configurations with more boreholes began at a lower temperature but eventually reached temperatures closer to 12 °C. This was due to a lower thermal capacity of ground around these pipes.
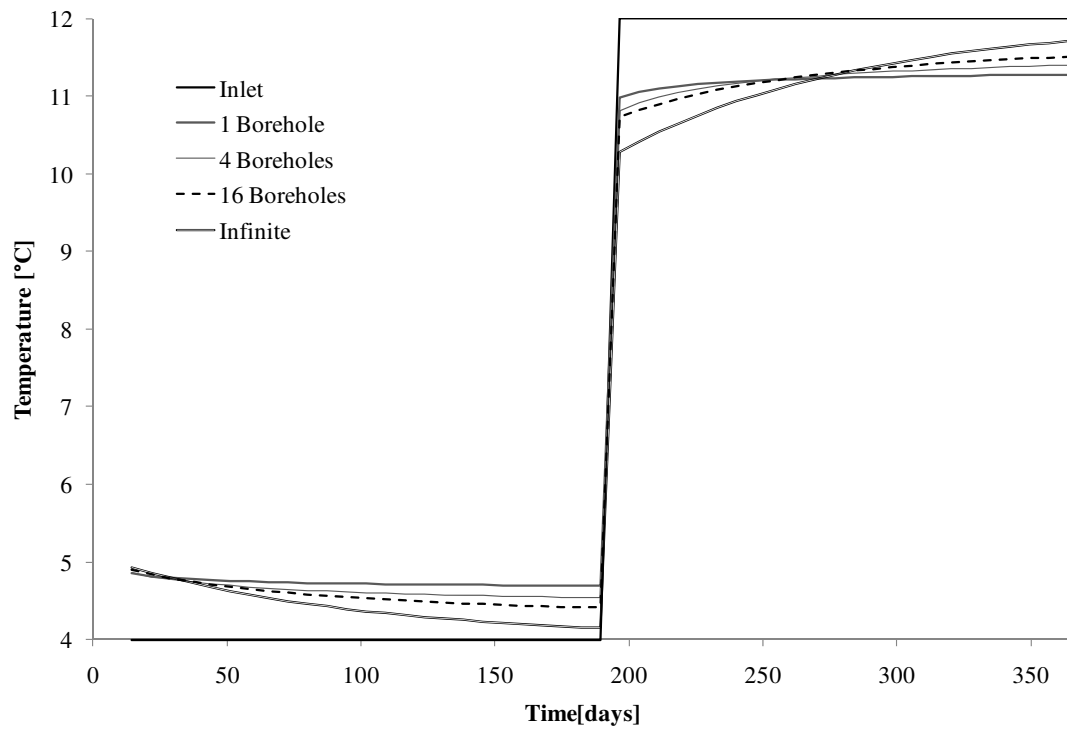
Figure 4.13   Outlet temperature after one year of BHEs in response to a step change in inlet temperature.

5 Integration of Underground Heat Exchangers in Building Models

In order to simulate the performance of the underground heat exchanger in the context of energy savings, it was required to enable the model to be simulated as an integrated part of a building and its mechanical systems. This can be used to understand the short and long term response of the systems to the exterior climate and the demands of the building.

In order to accomplish this, a numerical model of a building and its mechanical systems is required. The underground heat exchanger can then be integrated into this model in order to evaluate its impact on the building and its energy use. It can also be used to assess the energy storage abilities of the system.

This chapter describes the integration of the Simulink S–functions of the underground heat exchangers with a building system.

5.1 Integration into a building system

In order to accurately assess the performance of the underground heat exchangers it was necessary to integrate the model into a building system. This section will document the development a model for the heating and cooling system of a building and document each of the sub–systems within it. The final model of the heating and cooling system is shown in Figure 5.1. The upper part of the figure shows the model with three inputs and one output. The three inputs are listed and described below.

- Room temperature [°C] – The room temperature of the building zone or zones being considered. This is used to control the heating and cooling system.
- Outdoor temperature [°C] – This is required for the simulation of the underground heat exchanger.
- HVAC Return Temperature [°C] – This is the temperature of the fluid returning from the HVAC system.

The lower portion of Figure 5.1 shows the contents of the Heating/Cooling model. It contains the models for the heat pump, the underground heat exchanger, and the heating and control system. Each of these is described in further detail in the following sections.

This model has been included in a library UHSSLibrary.mdl. The model also has a file of constants, similar to those for the underground heat exchanger. This file, entitled 'UHSS_constants.m' contains constants and variables for each of the components of the Heating/Cooling model. The variables in the file are described in the sections of each model component.
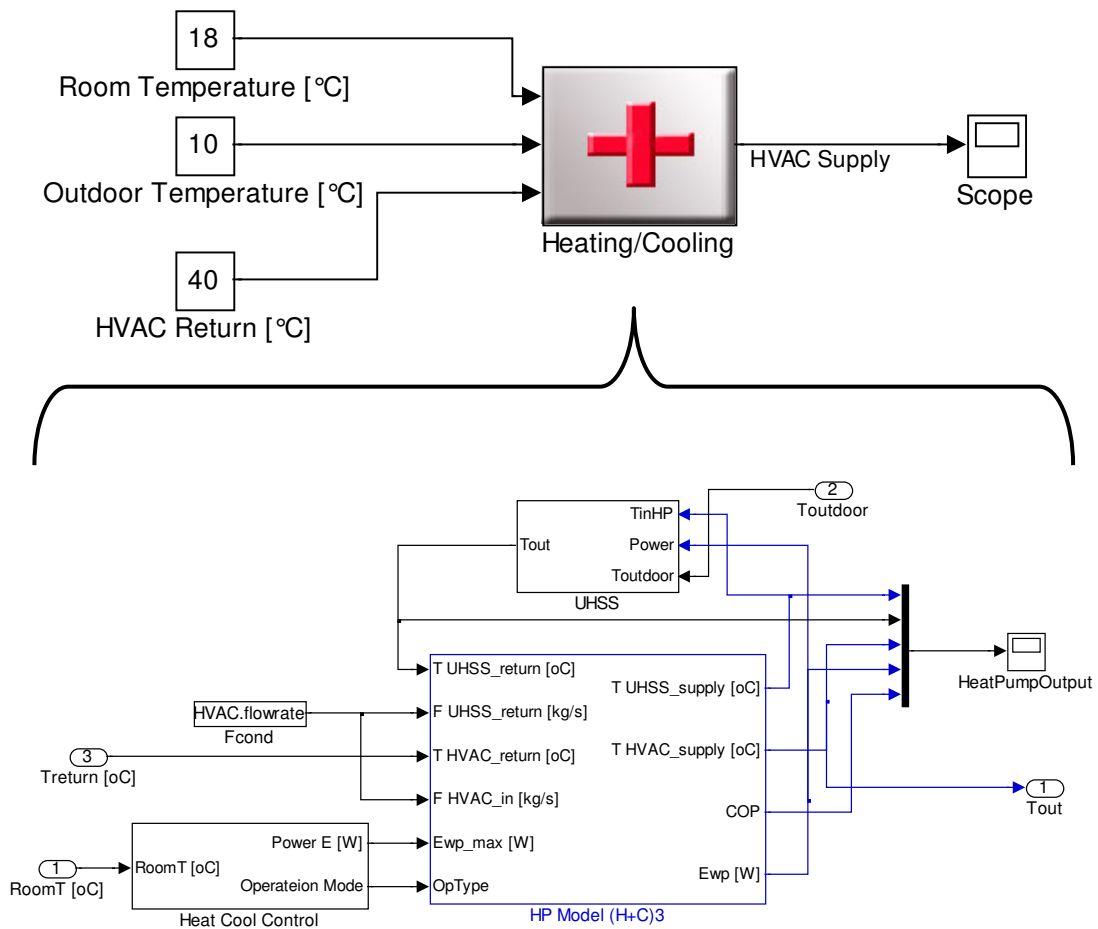
Figure 5.1   Model of the heating and cooling system.

5.1.1   Model of the heat exchangers

The model of the heat exchangers contains both the model for the underground heat exchanger along with a second heat exchanger between the heat pump and the underground heat exchanger. This model is shown in Figure 5.3.
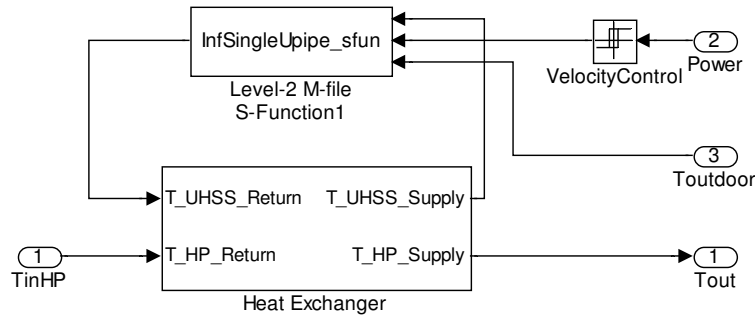
Figure 5.2   Model of the underground heat exchanger with a second heat exchanger.

The first step in the creation of this module was to develop a model of a heat exchanger. A heat exchanger is a device built for the efficient transfer of heat from one medium to another. In the case of this model it is used to transfer heat from the underground heat exchanger to the system of pipes that connects to the heat pump.

A simplified model was developed based on the steady state performance of a heat exchanger. This model allows for different flow rates coming from the underground heat exchanger and the heat pump. Figure 5.3 shows a schematic diagram of a heat exchanger.
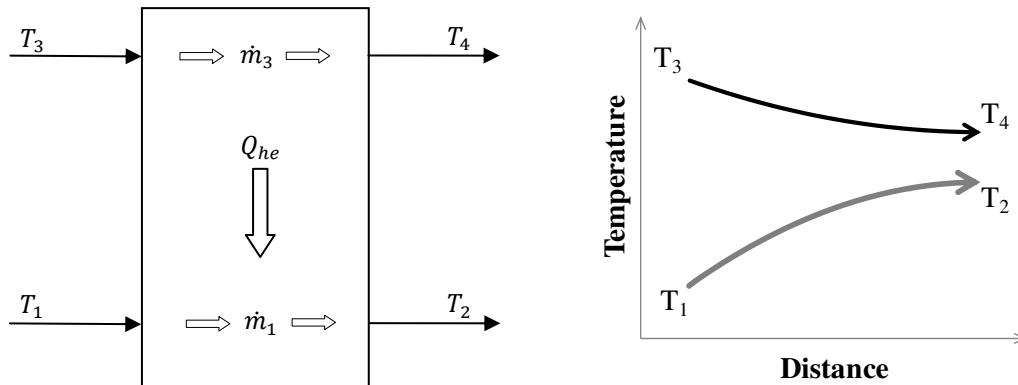
Figure 5.3   Schematic diagram of a heat exchanger.

There are two flows, $\dot{m}_1$ and $\dot{m}_3$, moving through the heat exchanger. For illustration purposes, assume that mass flow $\dot{m}_3$ loses heat at a rate of Q_{he} as it travels through the heat exchanger. By the principle of conservation of energy, the mass flow $\dot{m}_3$, will gain the same amount of heat. The resulting heat flow is shown below.

$$\begin{cases} Q_{he} = \dot{m}_1 \cdot c_{p1} \cdot (T_2 - T_1) \\ Q_{he} = \dot{m}_3 \cdot c_{p3} \cdot (T_3 - T_4) \end{cases} \qquad (5.1)$$

Based on the above equation, the ratio between the two flows is equal to the ratio between the two temperature differences.

$$\frac{\dot{m}_3}{\dot{m}_1} = \frac{(T_3 - T_4)}{(T_2 - T_1)} \qquad (5.2)$$

The efficiency, η_t, is defined as the ratio of the sum of the two temperature differences divided by the total temperature difference as shown below.

$$\eta_t = \frac{(T_3 - T_4) + (T_2 - T_1)}{(T_3 - T_1)} \qquad (5.3)$$

Based on the results of these equations and definitions, the temperatures T_2 and T_4 can be calculated.

$$T_2 = \frac{\eta_t \cdot (T_3 - T_1)}{\left(\frac{\dot{m}_3 \cdot c_{p3}}{\dot{m}_1 \cdot c_{p1}} + 1\right)} \cdot \frac{\dot{m}_3 \cdot c_{p3}}{\dot{m}_1 \cdot c_{p1}} + T_1 \qquad (5.4)$$

$$T_4 = T_3 - \frac{\eta_t \cdot (T_3 - T_1)}{\left(\frac{\dot{m}_3 \cdot c_{p3}}{\dot{m}_1 \cdot c_{p1}} + 1\right)} \qquad (5.5)$$

The completed model in Simulink is shown below in Figure 5.4.
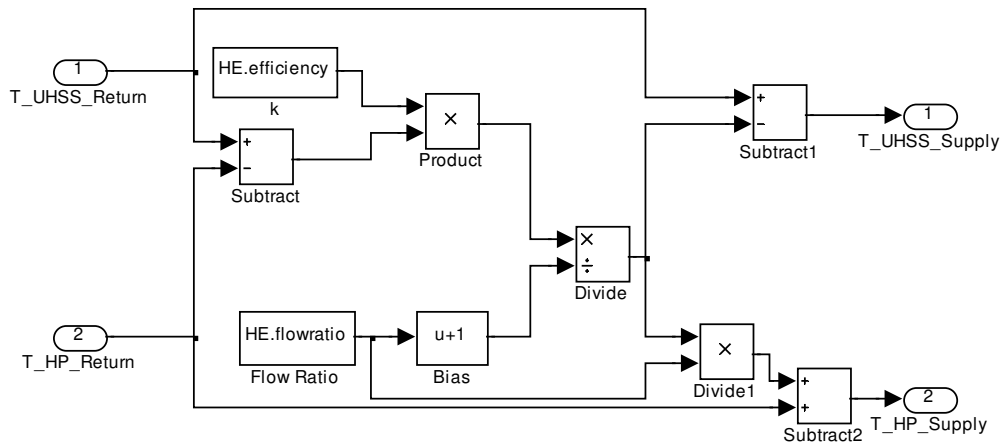
Figure 5.4   Simulink model of a heat exchanger.

The user has control over two variables that control the performance of the heat exchanger. These are listed in Table 5.1 and are included in the file 'UHSS_constants.m'.

Table 5.1   List of variables in the model of a heat exchanger.

Input Variable	Description	Units
HE.efficiency	Efficiency, η_t, of the heat exchanger	–
HE.flowratio	Ratio of flow in the underground heat exchanger to the flow in the heat pump	–

5.1.2 Modelling of a heat pump

Heat pumps were modelled with a series of equations and the COMSOL models of the underground heat exchangers were integrated with Simulink. The heat pump is modelled with some simple differential equations as shown in A. W. M. van Schijndel and de Wit (2003). The three equations used are shown below.

$$\begin{cases} COP = k \cdot \dfrac{0.5 \cdot T_{cin} + 0.5 \cdot T_{cout} + 273.15}{(0.5 \cdot T_{cin} + 0.5 \cdot T_{cout}) - (0.5 \cdot T_{vin} + 0.5 \cdot T_{vout})} \\[2mm] C_c \dfrac{dT_{cout}}{dt} = \dot{m}_{cin} \cdot c_w \cdot (T_{cin} - T_{cout}) + COP \cdot E_{hp} \\[2mm] C_v \dfrac{dT_{vout}}{dt} = \dot{m}_{vin} \cdot c_w \cdot (T_{vin} - T_{vout}) - (COP - 1) \cdot E_{hp} \end{cases} \qquad (5.6)$$

The model was developed for a heat pump with a coefficient of performance, COP, [–] as defined by the first equation in the set. In the ideal case the COP is equal to the average temperature of the condenser (in Kelvin) divided by the temperature difference between the compressor and the evaporator. To model more realistic situation the efficiency, k, is implemented.

The second two equations model the performance of both the condenser and the evaporator. In each case, the change in temperature of each reservoir is proportional to the mass flow, $\dot{m}$ through it; the electric power, E_{hp}, input, and the capacity, C, of the system. The specific thermal capacity of the fluid is denoted by c_w. The subscript c indicates the fluid at the condenser, v is the fluid at the evaporator, in is incoming, and out is outgoing.

This model was implemented as an S–function and modifications to the model were implemented to convert the heat pump into a reversible system to accommodate the use of the heat pump for both heating and cooling.

The user has control over a number of variables that control the performance of the heat pump. These are listed in Table 5.2 and are included in the file 'UHSS_constants.m'.

Table 5.2   List of variables in the model of a heat pump.

Input Variable	Description	Units
HP.WPTC2	Heating – High Temperature Limit	°C
HP.WPTC1	Heating – Low Temperature Limit	°C
HP.WPTH2	Cooling – High Temperature Limit	°C
HP.WPTH1	Cooling – Low Temperature Limit	°C
HP.WPkw	Efficiency of the heat pump	–
HP.Evap_Cap	Evaporator Capacity	J/(kg*K)
HP.Comp_Cap	Compressor Capacity	J/(kg*K)

The temperature limits are used in the control of the operation of the heat pump. The heat pump will turn off and resume operation based on the limits. For example the shut down once the heating reservoir is above the high temperature limit and will then resume operation once the temperature has fallen below the low temperature limit. The other variables are used in the calculations shown in Equation *(5.6)*.

The Simulink model of the heat pump is shown in Figure 5.5. The heat pump object calls an S–function model of the heat pump. The OnOffControl object at the bottom of the figure is used to shut off and resume operation of the heat pump based on the temperature limits described above.
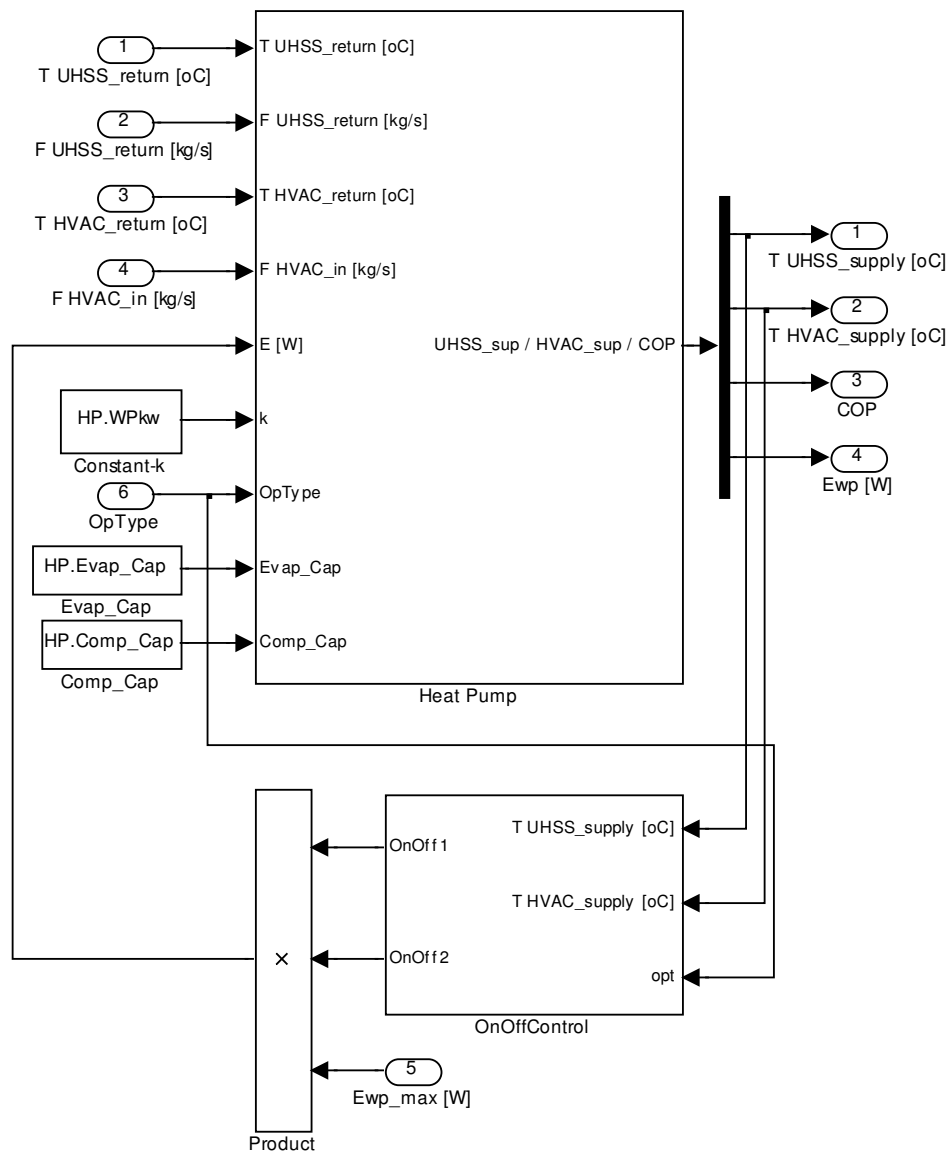
Figure 5.5   Simulink model of a heat pump.

5.1.3 Control of the heat pump

In order to control the temperature of a room or zone, the operation of the heat pump needed a control. A simple on/off control system was implemented for this. The model is shown in Figure 5.6. The power to the heat pump is controlled, as either on or off, for cooling and heating based on the temperature in the room (or zone). The control also outputs the required operation mode of the heat pump, either cooling or heating.
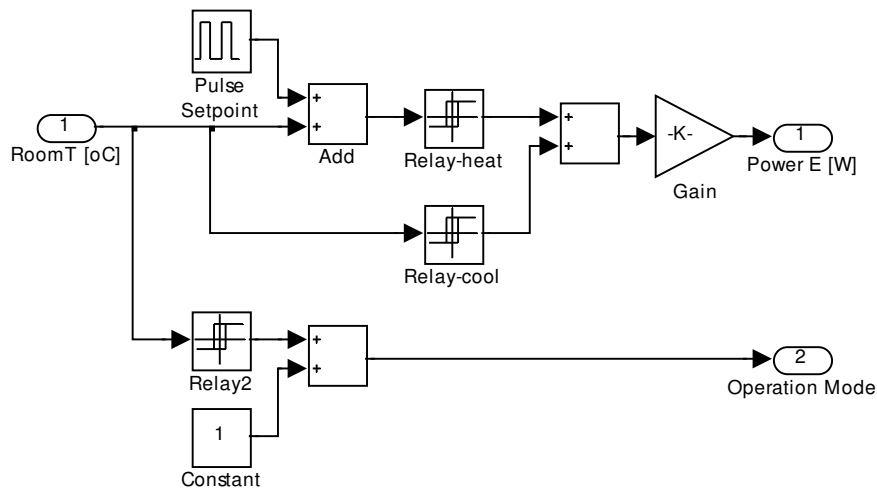
Figure 5.6   Simulink model of the heat pump control.

The user has control over the variables that control the operation of the heat pump control. These are listed in Table 5.3 and are included in the file 'UHSS_constants.m'. The variables include the electric power of the heat pump along with the low and high set points for the temperature. It also includes a balance temperature, above which the system operates in cooling mode and below which it operates in heating mode.

Table 5.3   List of variable for control of the heat pump.

Input Variable	Description	Units
HPC.egain	Electric power input to the heat pump	W
HPC.heat_high_setpoint	Heating – high setpoint	°C
HPC.heat_low_setpoint	Heating – low setpoint	°C
HPC.balance_temp	Balance temperature	°C
HPC.cool_high_setpoint	Cooling – high setpoint	°C
HPC.cool_low_setpoint	Cooling – low setpoint	°C

6 Performance of Underground Heat Exchangers

6.1 Response of a simple building model

In order to examine the integrated performance of BHEs, a number of simulations were conducted using a simple building model. In order to get a relatively straightforward comparison, a simple building model was used.

6.1.1 The ISE model of a building

ISE (Indoor temperature Simulink Engineering tool) is a simplified modelling tool for the simulation of the temperature response of a single zoned building (A. W. M. van Schijndel, 2003). It is used in Simulink with user–friendly graphical interface. ISE was designed to be used in the following situations:

1. A simple evaluation of the indoor temperature performance of a building
2. A simple estimate of both the heating a cooling potential and capacity
3. Allow for the evaluation and design of building systems and controls

The interface of the model in Simulink is shown in Figure 6.1.

The ISE model implements a model of a single building zone using the State Space equation in Simulink to simulate the performance of a building. The Simulink model is shown in Figure 6.2. The model treats the building as a single zone with a limited number of inputs. The model accounts for heat conduction and radiation through the wall and windows of the building and has additional inputs for internal heat gains and the output of the heating and cooling system. The model then computes the internal temperature in the zone. It also implements a temperature controller for regulating the heating and cooling output.
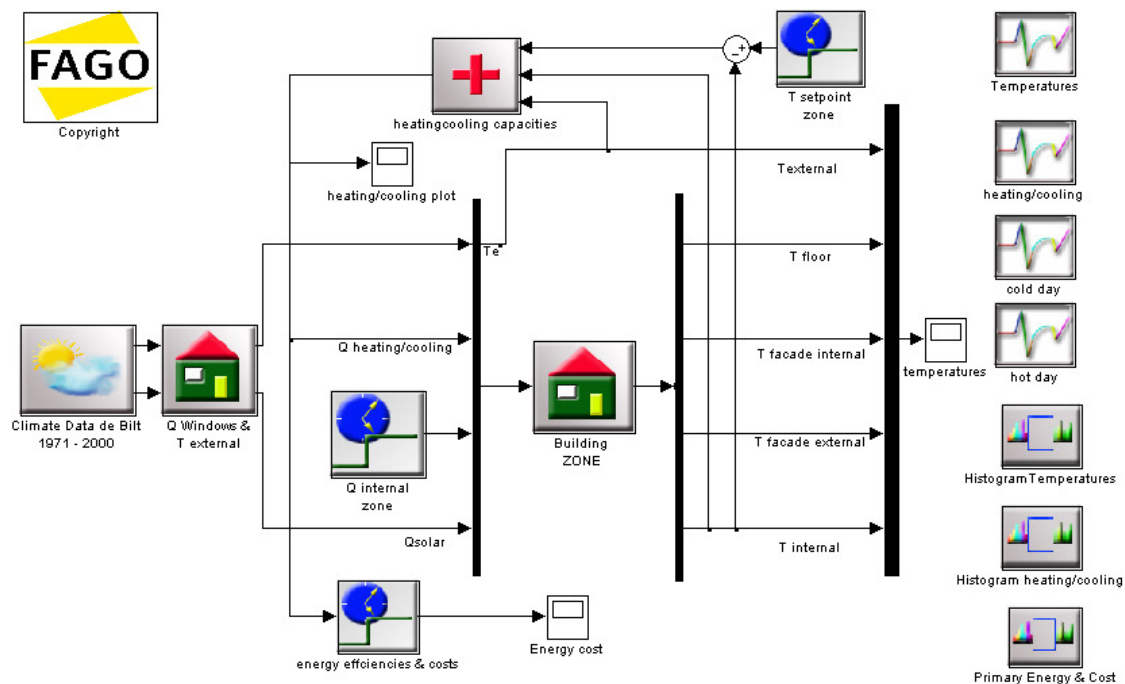
Figure 6.1   ISE (Indoor temperature Simulink Engineering tool).
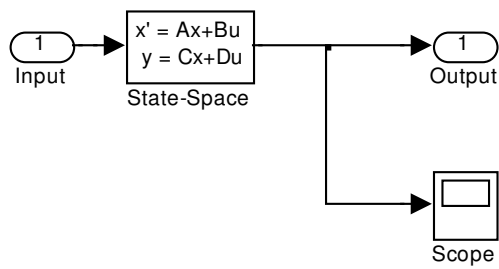
Figure 6.2   ISE Building ZONE model.

The ISE building model allows the user to input a number of parameters describing the building. This includes the volume of the building along with characteristics of the floor, walls, and windows. The parameters that can be input by the user are shown in Figure 6.3.

Figure 6.3   Parameters of the ISE Building ZONE model.

In order to include the performance of the underground heat exchangers it was necessary to integrate these models into the ISE model. The original model's heating and cooling system simply consisted of a proportional system controlled by the temperature difference between the indoor temperature and the desired temperature set point. The heat flow into the building zone was computed in this model.

The model of the heating and cooling system with an integrated underground heat exchanger that had been developed previously and shown in Figure 5.1 required modifications in order to be linked into the ISE model heating/cooling module. The building model in the ISE model requires an input of the heat flow, Q heating/cooling (W). This flow is the amount of heating or cooling determined by the control system required to heat or cool the building. Due to this, a module was developed to simulate a heating and cooling system. The model of the integrated system is shown in Figure 6.4.
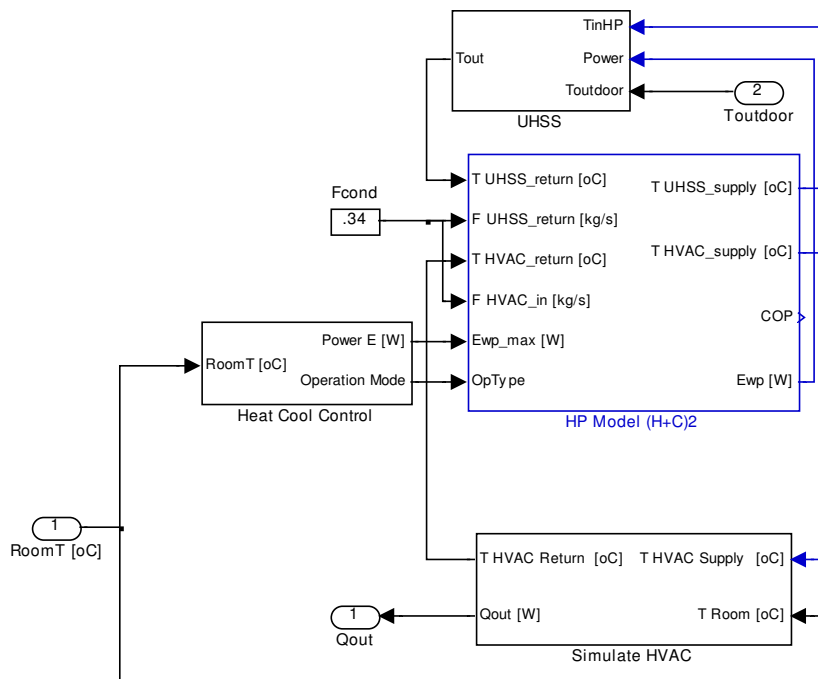
Figure 6.4   Model of the heating/cooling system with an integrated underground heat exchanger.

The model developed to simulate a heating and cooling system is shown in Figure 6.5. This model determines the quantity of heating or cooling, Q_{out}, required by the building zone. The output of the heating and cooling system is simply based on the average temperature of the fluid flowing through the system, the temperature of the room, and a heat transfer coefficient, K [W/K].

$$Q_{out} = K \cdot \left(\frac{T_{h,in} + T_{h,out}}{2} - T_{room} \right) \qquad (6.1)$$

To model the transient performance of the system a lumped model was developed. The model consisted of the heating and cooling system with a capacity of C_{hvac}. The change in energy of the system is simply the difference between the output of the system, the input to the system, and the change in energy. The resulting equation is shown below.

$$C_{hvac} \frac{dT_{h,out}}{dt} = \dot{m}_{hvac} \cdot c_p \cdot \left(T_{h,in} - T_{h,out}\right) - K \cdot \left(\frac{T_{h,in} + T_{h,out}}{2} - T_{room}\right) \qquad (6.2)$$

The temperature output of the heating system is solved for in the equation. From this result, the heat output of the system is determined from Equation *(6.1)*. The resulting model, developed in Simulink is shown in Figure 6.5.
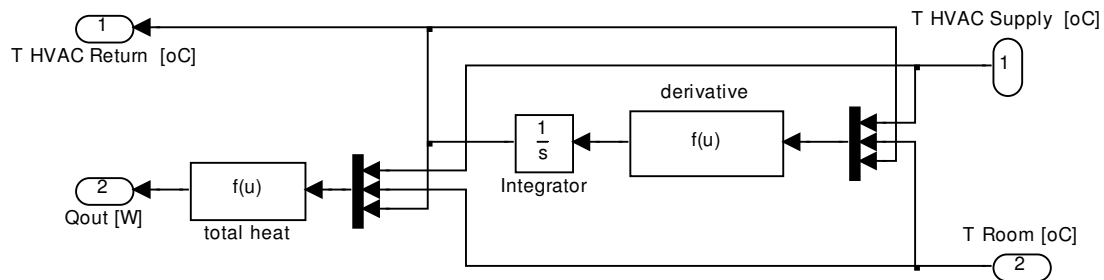
Figure 6.5   Model of the heating/cooling system.

6.1.2   Simulation details

The simulations, using the ISE model with an integrated geothermal heat pump were executed for two locations. The locations chosen were De Bilt, The Netherlands and Gothenburg, Sweden. Weather data was available for both of these locations. Three year simulations were conducted. The buildings were simulated with the same parameters relating to the size and other characteristics of the building. The parameters used for the simulations are shown in Table 6.1

Table 6.1   Building parameters for the building simulations

Description	Value	Units
Building Volume	500	m^3
Ventilation Rate	1	ach
Facade Surface Area	242	m^2
Facade R Value	2.5	m^2K/W
Internal Wall Area	100	m^2
Window Surface Area	16	m^2
Window R Value	0.33	m^2K/W

The operation of the heat pump was controlled using the system detailed in Section 5.1.3. The temperature setting for the heat pump are shown below in Table 6.2

Table 6.2 Temperature control settings for the heat pump.

Temperatures [°C]				
	Heating		Cooling	
Balance	Low	High	Low	High
22.5	20	21.5	24.5	26

In the ISE model, thirty years of weather data for De Bilt was included from the year 1971 to 2000. The simulations were conducted over the years from 1973 to 1975. This was done, since the average outdoor temperature in each of these years was near the average for the entire time period. The outdoor temperature for the selected time period is shown in Figure 6.6. Three full years of weather data were available for Gothenburg from 2004 to 2006. The outdoor temperature for this period is shown in Figure 6.7.

BHEs were used in each of the systems. The model InfSingleUpipe_sfun.m was used for the simulations. The spacing between the pipes simulated at a four and six metre spacing to get an indications on the effects of pipe spacing. A full list of the parameters used for the heat exchanger is listed in Table 6.3 and Table 6.4.

Internal heat loads were also included in the buildings models. This was done to ensure there was a cooling load in the building at certain parts of the year. The amounts are listed in Table 6.5. The relatively high loads for the Gothenburg simulations were chosen to simulate a building with considerable cooling loads in order to assess energy storage capabilities.

Table 6.3   *List of static parameters, mesh variables, and an output variable in the BHE constant files used for the building simulations.*

Input Variable	Value	Units
Static parameters		
U_pipe.pipelength	200	m
U_pipe.pipedepth	2	m
U_pipe.soilspace	20	m
U_pipe.boreholediameter	0.13	m
U_pipe.relativepipelocation	0.8	–
U_pipe. TsoilinitialdegC	9.8	°C
U_pipe. Boreholespace[1]	4 and 6	M
Mesh Variables		
U_pipe.mesh_geom1_hauto	8	–
U_pipe.mesh_geom2_hmax	4	M
Output Variable		
U_pipe.outputdetails	1	–

*Table 6.4   List of simulation variables in the BHE constant files used for building
            simulations.*

Input Variable	Value	Units
Simulation Variables		
U_pipe.L0	0.1257	m
U_pipe.alpha0	164	W/(m^2·K)
U_pipe.Tgin	9.8	°C
U_pipe.Ti	9.8	°C
U_pipe.xarea	0.001257	°C
U_pipe.flow_velocity	0.6	m/s
U_pipe.Tsoiltop	9.8	°C
U_pipe.Tsoilbottom	9.8	°C
U_pipe.k_ground	3.5	W/(m·K)
U_pipe.rho_ground	2000	kg/m^3
U_pipe.cp_ground	1000	J/(kg·K)
U_pipe.k_grout	2	W/(m·K)
U_pipe.rho_grout	2000	kg/m^3
U_pipe.cp_grout	1000	J/(kg·K)
U_pipe.k_fluid	0	W/(m·K)
U_pipe.rho_fluid	1000	kg/m^3
U_pipe.cp_fluid	4200	J/(kg·K)

Table 6.5   Internal heat generation.

	Internal Heat Generation [W]	
Location	Constant	Additional during Day
De Bilt	1500	4500
Gothenburg	2800	5900
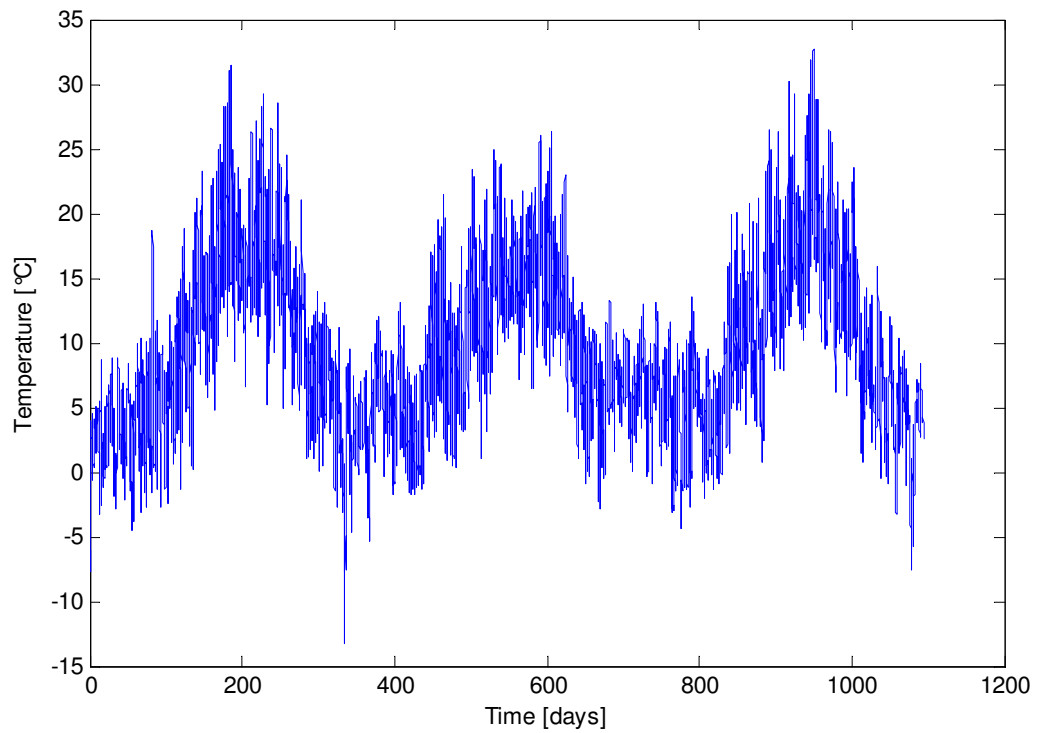
Figure 6.6   Outdoor temperature in De Bilt, Netherlands (1973 to 1975).
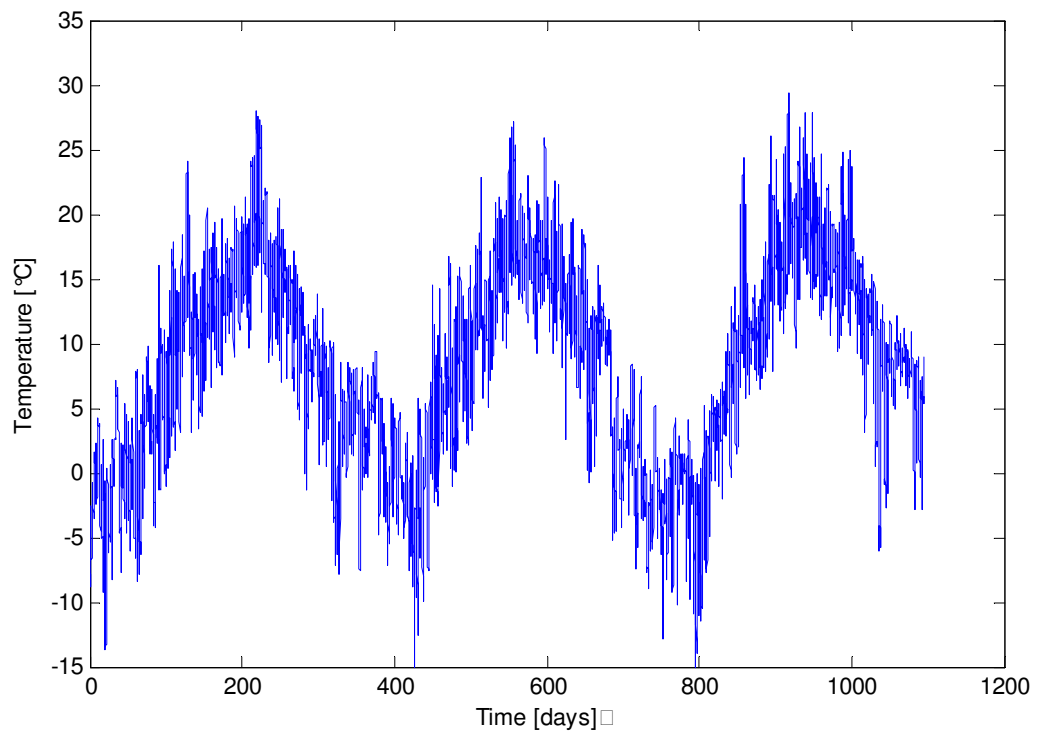
Figure 6.7   Outdoor temperature in Gothenburg, Sweden (2004 to 2006).

6.1.3   De Bilt results

The overall performance of the system was examined by calculating the energy use and output of the buildings over the length of the simulation. Figure 6.8 and Figure 6.9 show the cumulative energy supplied to the building and electrical energy used by the building respectively. The cumulative energy used after three years is tabulated and shown in Table 6.6.

Figure 6.8   Cumulative absolute energy supplied to the building zone for the De Bilt simulations.

Figure 6.9   Cumulative absolute energy supplied as electricity to the heat pump for the De Bilt simulations.

Table 6.6 Cumulative energy use in De Bilt Simulations

| Borehole spacing [m] | Year | Energy [MWh] | | | | COP |
		Heating	Cooling	Cumulative Use	HP Electrical	
4 m	1	10.17	2.50	12.68	2.74	4.63
	2	7.53	1.44	8.97	2.01	4.46
	3	8.82	2.90	11.72	2.58	4.54
	Total	26.53	6.84	33.37	7.33	4.55
6 m	1	10.17	2.50	12.67	2.70	4.69
	2	7.54	1.43	8.97	1.91	4.70
	3	8.82	2.91	11.73	2.54	4.61
	Total	26.53	6.84	33.37	7.16	4.66

The temperature in the ground was also determined during the simulations. The ground temperature at a depth midway along the borehole and two meters away are compared in Figure 6.10.

Figure 6.10   Ground temperature at a distance of 2 m from the borehole for the De Bilt Simulations.

Note that in the De Bilt simulations there was a cooling load, but it was significantly lower than the heating load. This is demonstrated by the fact that the temperature in the ground continued to decline over the length of the simulation. The results also show a somewhat lower electrical energy use with greater pipe spacing.

6.1.4 Gothenburg results

The overall performance of the system was examined by calculating the energy use and output of the buildings over the length of the simulation. Figure 6.11 and Figure 6.12 show the cumulative energy supplied to the building and electrical energy used by the building respectively. The cumulative energy used after three years is tabulated and shown in Table 6.7
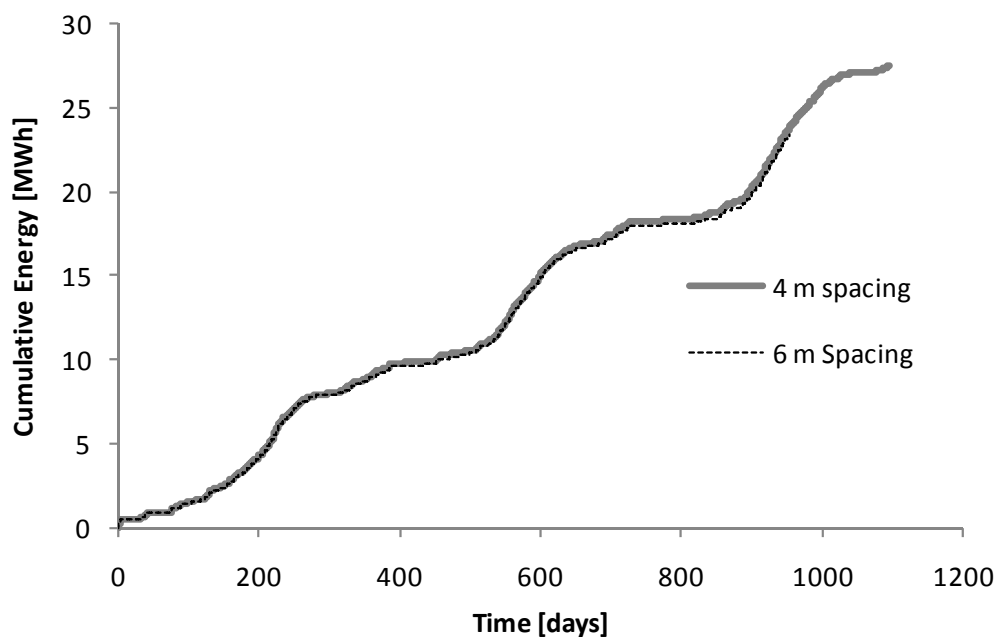
Figure 6.11   Cumulative absolute energy supplied to the building zone for the Gothenburg simulations.
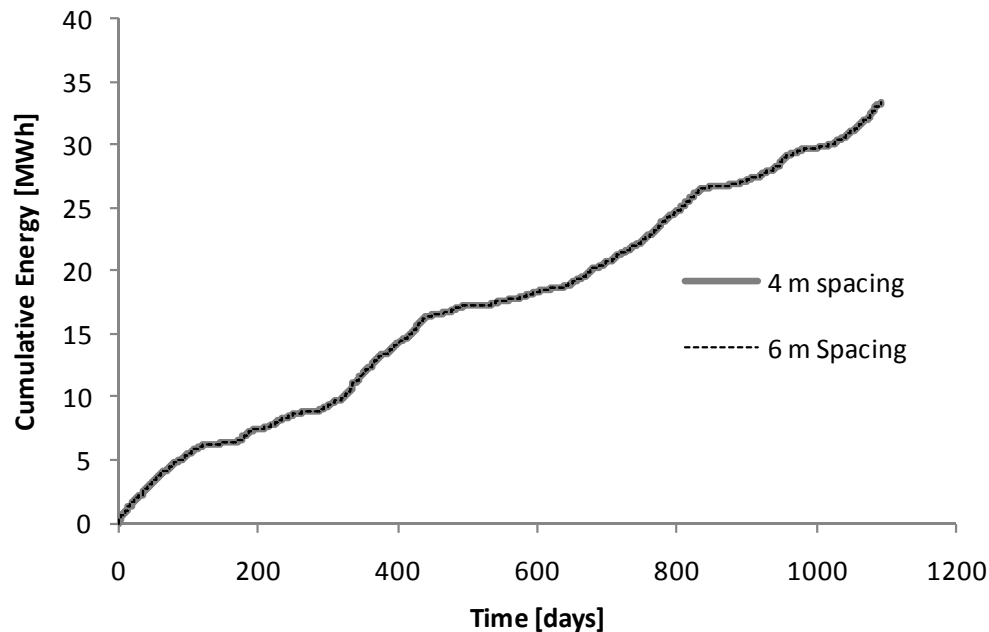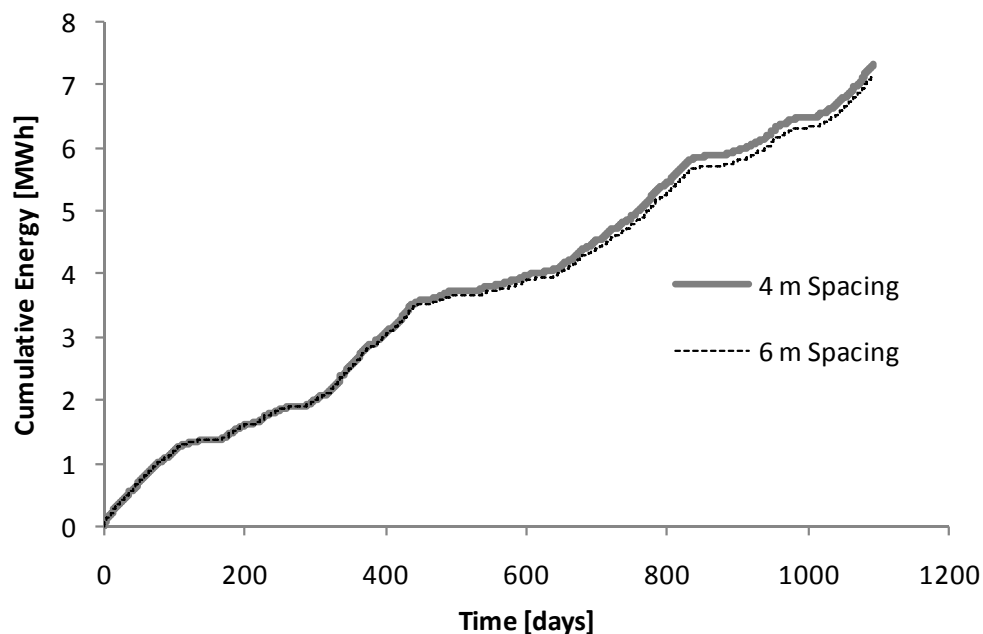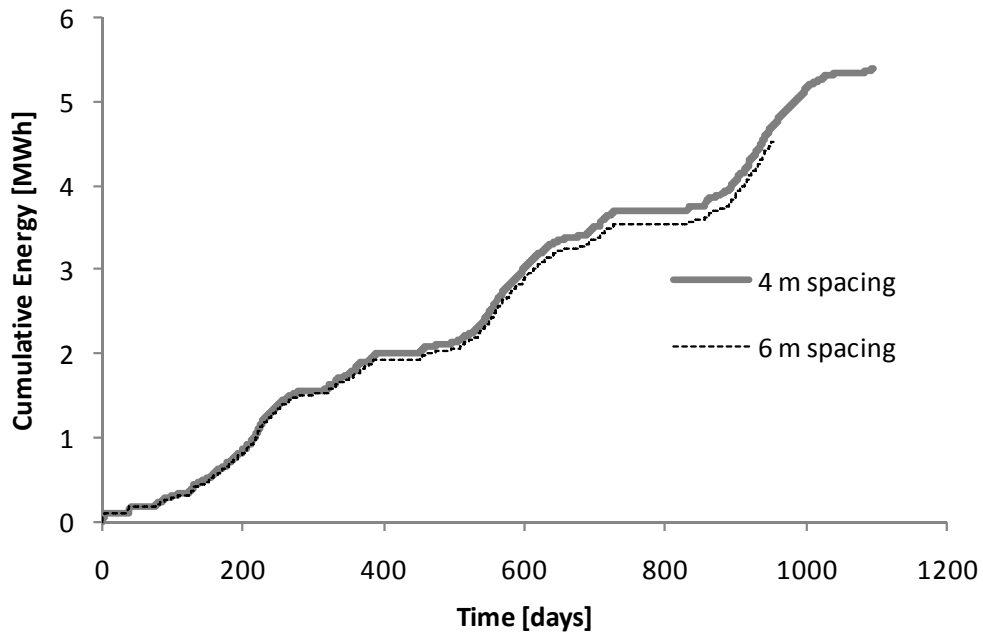
Figure 6.12   Cumulative absolute energy supplied as electricity to the heat pump for the Gothenburg simulations.

Table 6.7 Cumulative energy use in Gothenburg simulations.

| Borehole spacing [m] | Year | Energy [MWh] | | | | COP |
		Heating	Cooling	Cumulative Use	HP Electrical	
4 m	1	2.82	6.49	9.31	1.88	4.95
	2	2.15	6.72	8.87	1.81	4.91
	3	0.44	5.08	5.52	1.02	5.41
	Total	5.41	18.29	23.70	4.71	5.04
6 m	1	2.63	6.51	9.15	1.81	5.05
	2	2.03	6.74	8.77	1.72	5.10
	3	0.36	5.10	5.46	0.99	5.52
	Total	5.02	18.36	23.37	4.52	5.17

The temperature in the ground was also determined during the simulations. The ground temperature at a depth midway along the borehole and two meters away are compared in Figure 6.13.
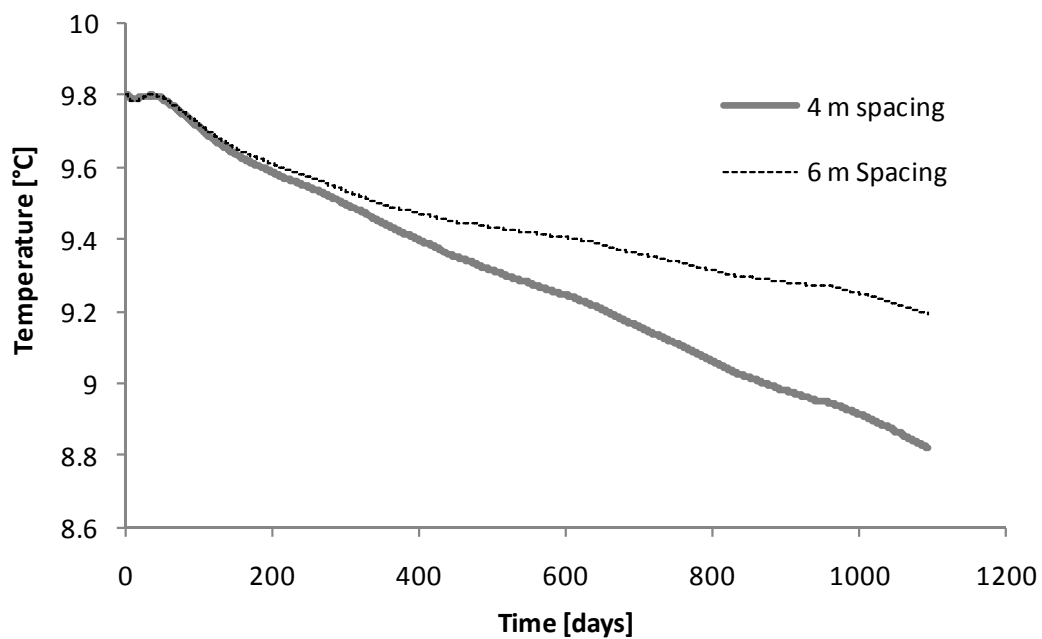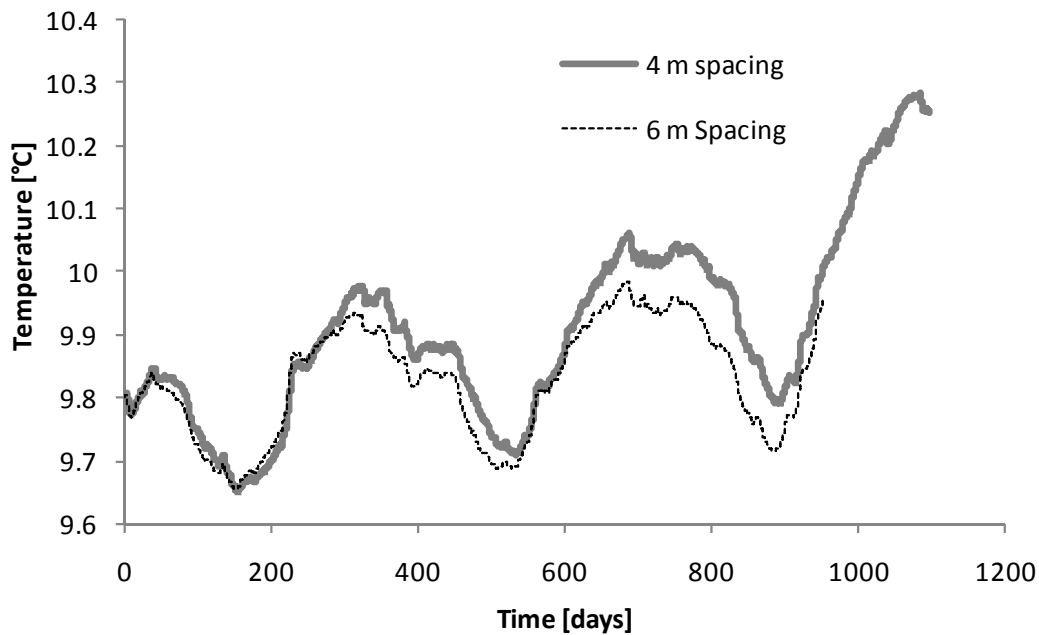
Figure 6.13   Ground temperature at a distance of 2 m from the borehole for the Gothenburg simulations.

In the Gothenburg simulations there was a higher cooling load than a heating load. As a result, the trend in the temperature in the ground rises. There is, however, a periodic fluctuation in the temperature as a result of heating and cooling. The results also show a somewhat lower electrical energy use with higher pipe spacing.

6.1.5   Discussion

The results of the simulations give some preliminary results on the performance of BHEs and their use in energy storage. The results from Gothenburg, particularly, show the fluctuations in the temperature of the ground with underground heat storage. There is a rise in the temperature of the ground during the cooling season and the lowering of temperatures during the heating season can be observed. When designing a system, a balance between the heat removed and the heat stored needs to be maintained.

There also was a small decrease in the total energy use of the heat pump when the pipe spacing was increased from four to six meters for both of the simulations. This could be due, in heating for example, to lower ground temperatures around the underground heat exchanger which could lead to lower temperature differences between the inlet and outlet temperatures of the heat exchanger. This would result in a lower heat output requiring longer operation of the heat pump and greater energy use.

7 Case Study of an Existing Building: Anatomy House

7.1 Development of a lumped thermal model

In order to simulate an existing building a simplified lumped thermal capacity model was used to accomplish this. A lumped model is based on the assumption that the spatial variation of temperature in a body is constant. That is, the temperature variation across the body in space is negligible. The temperature in that body varies with time depending on its thermal capacity and the heat flows into and out of the body (Bejan and Kraus, 2003). For example, consider a body with a volume, V, surface area, A_s, density, ρ, and specific heat, c_p, at an initial temperature of T_i. After time t=0, there is conduction that occurs from an external temperature, $T_{ext}(t)$, through a material with a conductance, U. Using the lumped model, the temperature of the body, T, is determined with the differential equation of the process shown below.

$$\rho \cdot V \cdot c_p \cdot \frac{dT}{dt} = -U \cdot A_s \cdot (T - T_{ext}(t)) \qquad (7.1)$$

With the initial condition:

$$T(t = 0) = T_i \qquad (7.2)$$

This example only shows the conduction into the body. To simulate a building all of the significant heat flows are considered, such as solar radiation, conduction, wind convection, internal heat gains, ventilation, and air leakage. The building can also be divided into multiple bodies with thermal flows between them. The differential equation for the lumped model of a body with only conduction into that body can be solved using Simulink. The model for this is shown in Figure 7.1.
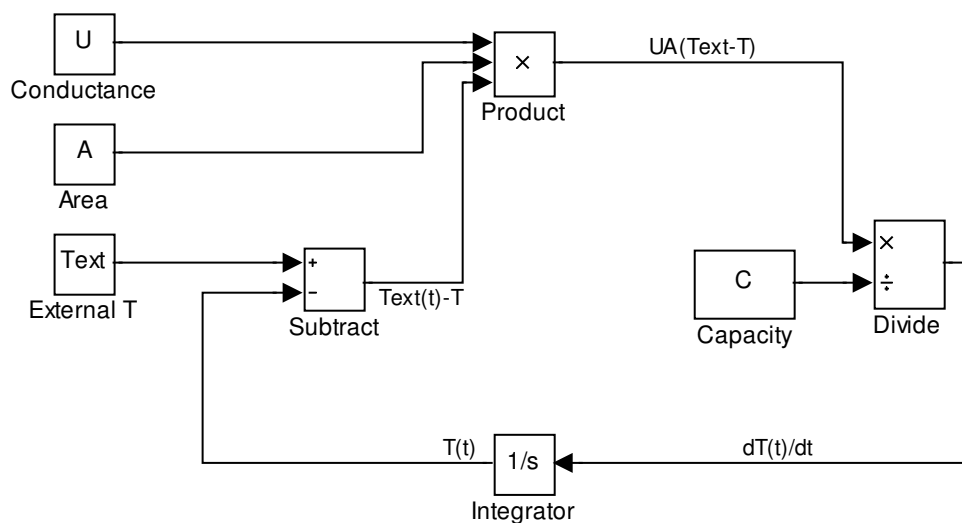
Figure 7.1   A simple lumped model of a building in Simulink with heat flow into the body through conduction.

The solution to Equation *(7.1)* can be solved for as the equation shown below when assuming a constant exterior temperature, T_{ext}.

$$T(t) = T_{ext} + (T_i - T_{ext}) \cdot e^{-t/t_c} \qquad (7.3)$$

Where the time constant, t_c is:

$$t_c = \frac{U \cdot A_s}{C} \qquad (7.4)$$

The lumped capacity of the building, C, is found with the equation below:

$$C = \rho \cdot V \cdot c_p \qquad (7.5)$$

Using a simple numerical example, the solution to the differential equation was compared to the Simulink model results for the temperature of the body. The results, shown in Figure 7.2, show that Simulink calculates a solution that is identical to the analytical solution.
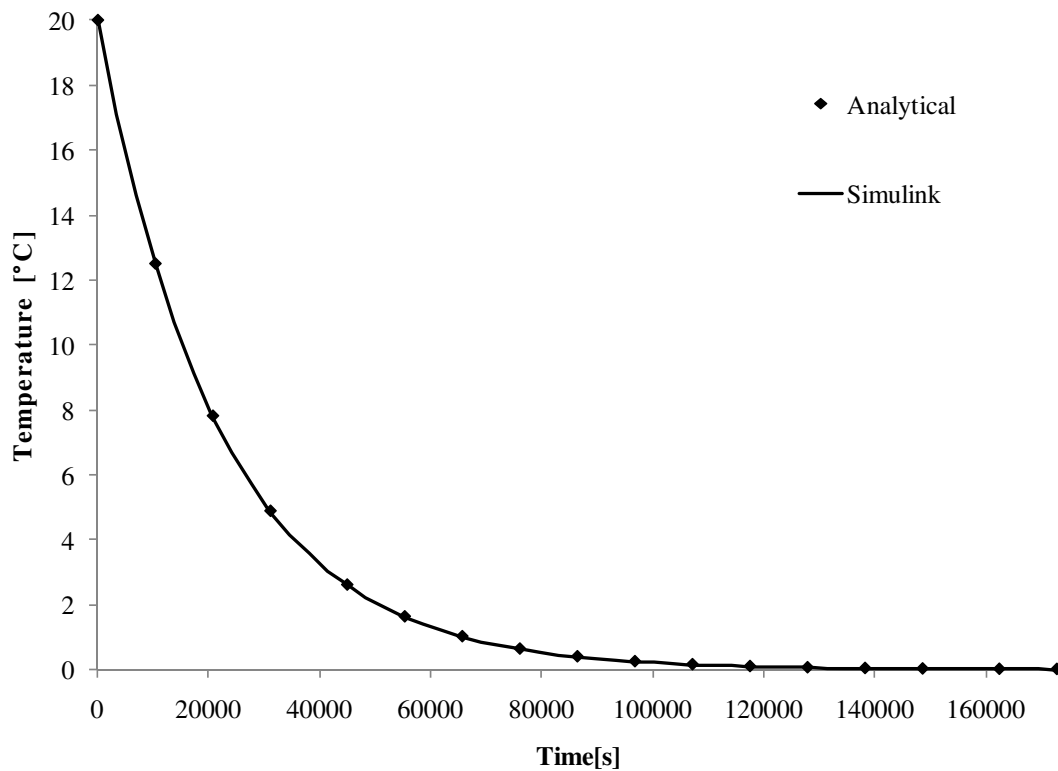
Figure 7.2   Comparison of simple lumped model in Simulink with the analytical solution of the equation.

A lumped model for a whole building can also be developed. For a simple building the interior of the building can be modelled as one body. The temperature in the interior of the building is assumed to be constant. There are also additional thermal flows into and out of the building that need to be considered. These flows include conduction through the walls and windows, air convection at surfaces, long and short wave radiation, indoor heating systems, internal heat gains, ventilation, air conditioning, and air leakage.

In developing more complex models of building, multiple lumped bodies can be modelled together in one simulation. For example a building could be divided into two zones if there are two floors in the building. The model must then consider any flows between the zones. The performance of different flows can also be modelled in detail using Simulink. The thermal flows provided by a heating system, for example, can be modelled to include proportional control and delayed thermal outflows.

7.2 Anatomy House (Anatomi Hus) at Salgrenska Hospital in Gothenburg

In order to evaluate the underground heat exchanger and its integration into buildings, a case study of an existing building was conducted. This case study investigated the performance of the Anatomy House, an office building located in Göteborg, Sweden. It is located near the Sahlgrenska University Hospital. Figure 7.3 shows an image of the building and its surroundings.

The building was originally constructed in the 1960s as an anatomy laboratory, thus the name Anatomy House. The building was later renovated during which the laboratories were removed and replaced with offices. The renovation of the building included the installation of a new ventilation and air conditioning system.

The new ventilation system was a Variable Air Volume (VAV) system. An air conditioning plant was installed in the attic as part of the ventilation system. The purpose of the air conditioning system is to maintain a supply air temperature to the offices of 15 °C during working hours. The ventilation rates to the offices can be adjusted based on the occupancy and when additional cooling is required.

This section presents the model developed in Simulink to model the Anatomy House.
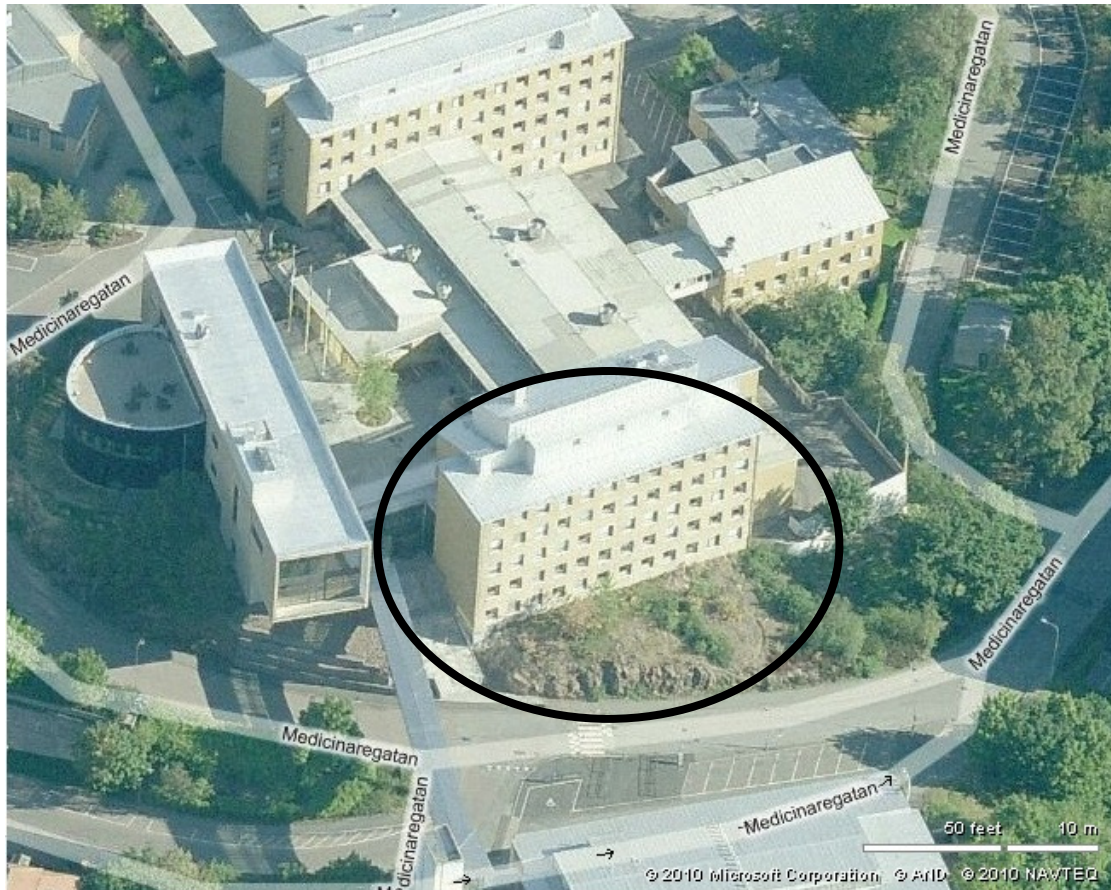
Figure 7.3   Image and location of the Anatomy House in Göteborg (Microsoft, 2010).

7.2.1   Building characteristics

A lumped model of the building was developed to consider the main heat sources, heat sinks, and components with a significant heat capacity. As a first step, a model was developed for the fourth floor of the building. This model considered heat conduction through the walls and windows, thermal radiation through the windows, internal heat generation, ventilation and air leakage, and supplementary heating. The floor was divided into two zones while accounting for cross–ventilation between them. This section describes the assumptions and calculations that have been used to model the Anatomy House. The assumptions and building characteristics are based on a previous study (Sasic Kalagasidis, 2009). In this study complex simulations were conducted of the Anatomy house using the building simulation program HAM–Tools (Sasic Kalagasidis, 2004).

The response of each of the zones to thermal loads has been calculated using a lumped thermal capacity of each zone. Typically, the thermal capacity of the building envelope that contributes to the lumped capacity of the building interior is considered to be only the thermal mass located on the interior side of the insulation within the building envelope. The modeling of the Anatomy House was more challenging because of the lack of insulation. The wall sections in the building consist of a solid 70 cm brick wall. To determine the thickness of the wall that contributed to the lumped capacity, the penetration depth, d_p, of the wall was calculated using Equation

(7.6) with a time period, t_p, of one day. This time period was used to model the daily fluctuation in the indoor environment.

$$d_p = \sqrt{\frac{a \cdot t_p}{\pi}} \qquad (7.6)$$

In addition to the brick wall double pane windows are part of the building envelope. The thermal properties of the building envelope are shown in Table 7.1.

Table 7.1   Thermal properties of the building envelope.

	Thermal Conductivity [W/m·K]	Density [kg/m^3]	Heat Capacity [J/kg·K]	Emissivity [–]	Absorptivity [–]
Brick	0.7	1800	800	0.9	0.4
Glass	1	2500	750	0.9	0.06

A lumped model was also used for determining the outer temperature of the wall which was used to calculate the heat conduction into the building. The same penetration depth was used as described above. The heat flows used in this lumped model include the heat flow through the wall to the interior, heat transfer at the surface including convection, and heat flow due to long and short wave radiation.

The solar radiation absorbed on the wall is calculated using:

$$q_{solar,wall} = a_{abs,i} \cdot \left[I_{direct,i}(t) + I_{diffuse}(t) \right] \cdot A_i \qquad (7.7)$$

The heat transfer at the surface was calculated using:

$$\alpha_c = 5.82 + 3.96 \cdot u \qquad\qquad u \leq 5\ m/s \qquad (7.8)$$

$$\alpha_c = 7.68 \cdot u^{0.75} + 3.96 \cdot u \qquad\qquad u > 5\ m/s \qquad (7.9)$$

The heat conduction from the interior through the wall was then calculated using Equation (7.10).

$$Q_{wall} = UA_{wall} \cdot (T_{interior} - T_{outer}) \qquad (7.10)$$

The lumped model developed for each interior zone also included the conduction. The other heat flows include internal heat sources, conduction and solar radiation through the windows, supply air ventilation, air leakage, supply heat, and cross ventilation between the zones.

The intensity of the internal heat gains was based on the data from.... They are listed in Table 7.2. The internal gains during the working hours from 8:00 to 18:00 are 3890 W. During non–working hours the internal heat gains are assumed to be 500W.

Table 7.2   Intensity of internal heat gains

Gain Type	Description	Total Gain [W]
People	There are 12.6 (60%) out of possible 21, each releasing 80 W.	1010
Computers and Printers	8 W/m² of floor area or 100 W per office	1310
Office Lighting	7 W/m² of floor area with 12.6 out of 21 offices in use at the same time	1150
Corridor Lighting	5 W/m² of floor area	420
	Total	3890

Heat gains through the windows consisted of solar radiation and conduction. Radiation was calculated using:

$$Q_{sol}(t) = a \cdot b \cdot \left[I_{direct,i}(t) \cdot \tau_1(\alpha) + I_{diffuse}(t) \cdot \tau_2 \right] \cdot A_{window} \qquad (7.11)$$

The transmittance τ_1, is a function of the angle of incidence. The solar transmittance was calculated using the values shown below in Table 7.3.

Table 7.3   Solar transmittance, τ_1, versus solar incidence angle.

α	0	10	20	30	40	50	60	70	80	90
τ_1	0.747	0.747	0.745	0.74	0.73	0.707	0.652	0.517	0.263	0

The VAV system provides ventilation and cooling when the air is warm to the offices. The air exchange rate is controlled as shown in Figure 7.4. The minimum air exchange rate of 0.7 ACH is provided at all times. If the temperature rises above 21 °C the exchange rate is increased to 1 ACH. A maximum supply of 3 ACH is provided when the temperature is at 25 °C or above. The air supply rate, as described, is enabled during normal working hours from 6:00 to 19:00. During evenings and on weekends a supply rate of 0.4 ACH is maintained. A supply air temperature of 15 °C was maintained.
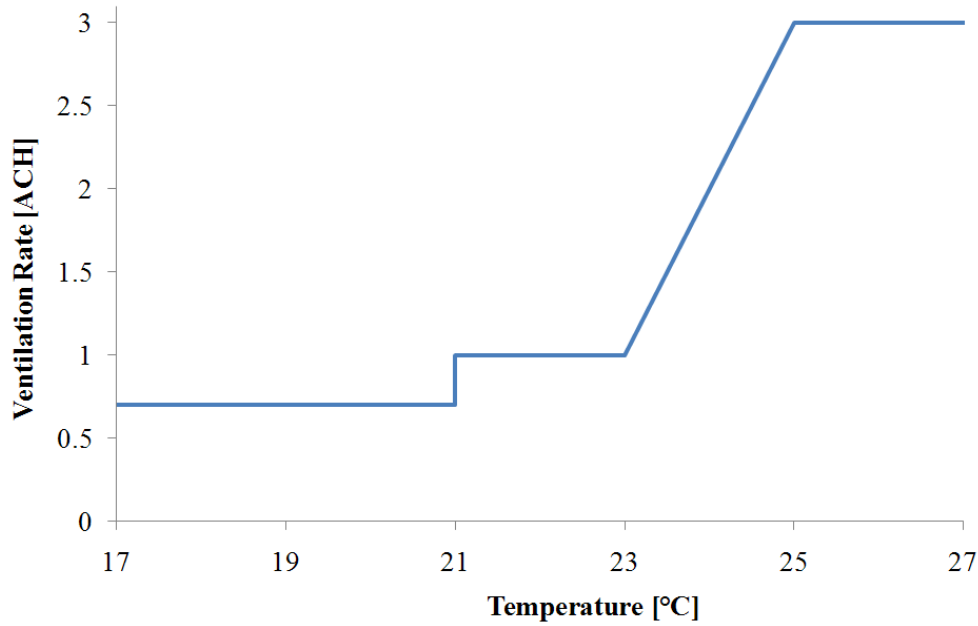
Figure 7.4   Supply air flow rate as controlled by the indoor air temperature during working hours.

The total air supply to each zone was calculated based on the assumption that all of the air supply units were of the same size and provided the same ventilation rate. This means that the total air supply to the floor was based on the number of vents as shown in Equation (7.12). The total ventilation for the floor was 23.7 % of the total ventilation.

$$Q_{a,floor} = \frac{number\ of\ supply\ air\ units\ per\ floor}{total\ number\ of\ supply\ units} \cdot Q_{a,total} \qquad (7.12)$$

The difference between the supply and exhaust flow rate was found to be approximately 35%. The exhaust fan takes more air out than is introduced by the supply fan resulting in an under–pressurization of the building. The air leakage, in the calculations, was assumed to be 35% during working hours and zero during nights and weekends.

There is no solid partition between the north and south zone of the building. During normal conditions it is assumed that many doors are left ajar and there is a significant amount of air movement between the zones. Based on this assumption an exchange rate of 3 ACH of cross ventilation was simulated.

The cooling load on the zone and for the air conditioning system was determined during the simulations. The cooling load in the air conditioning plant was calculated by determining how much cooling was required to maintain a constant supply air temperature. The design temperature, T_{supply}, was 13 °C.

$$Q_{cool,coil} = \begin{cases} 0\ if\ T_{out} \leq T_{supply} \\ \rho \cdot C_p \cdot Q_{a,total}(t) \cdot [T_{out}(t) - T_{supply}]\ if\ T_{out} > T_{supply} \end{cases} \qquad (7.13)$$

The effect of ventilation on the interior zones of the building was based on the ventilation rate as described in Figure 7.4 and the temperature difference between the interior temperature and the supply air temperature as described in Equation (7.14).

$$Q_{vent} = \rho \cdot C_p \cdot Q_{a,total}(t) \cdot \left[T_{supply} - T_{zone}(t)\right] \qquad (7.14)$$

The heating load was calculated based on the heat required to maintain an indoor air temperature of 21 °C during working hours and 20 °C during nights and weekends.

7.2.2   Simulink model of the building

The model of the Anatomy House, developed in Simulink, divided the building into a north zone and a south zone. The loads and temperatures are then simulated for each zone with the cross ventilations between them. The final model of the building is shown in Figure 7.5. The figure shows the subsystems for each of the north and south zone, along with the air conditioning and heating systems for each.
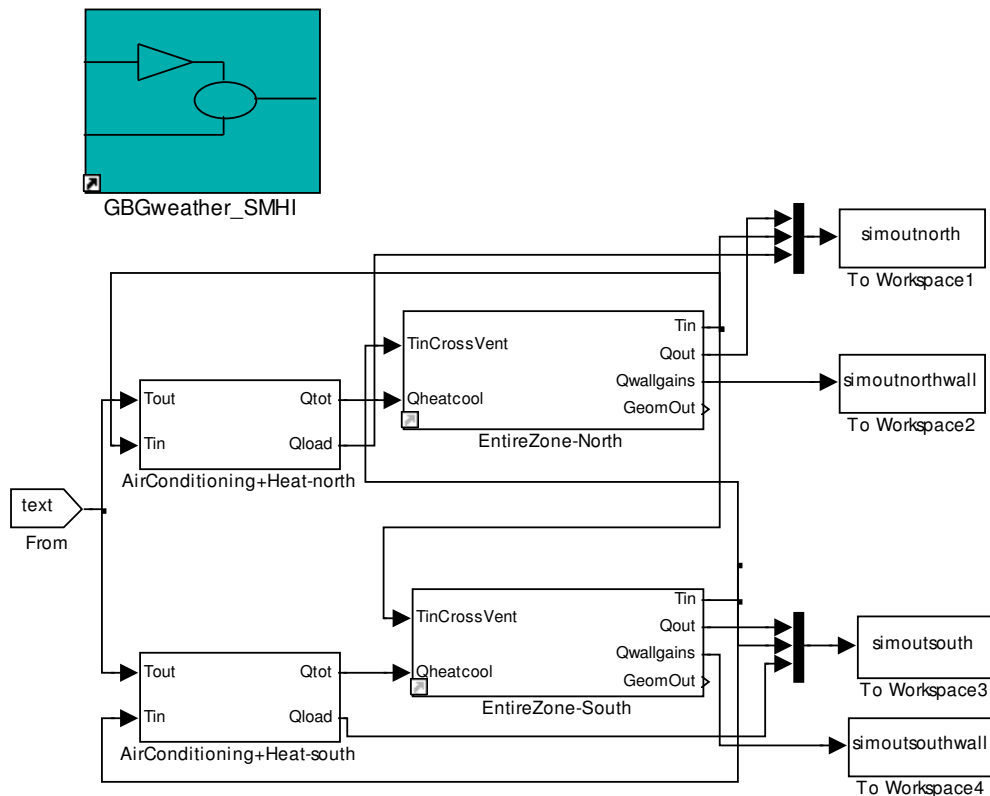
Figure 7.5   Simulink model of the Anatomy House

The model of each of the entire interior zones, as shown in Figure 7.6, determines the thermal loads from the exterior walls and inputs the results to the model of the interior zone. In each of the wall subsystems an equivalent exterior temperature of the wall is calculated along with the heat transfer through the windows. These values are then output to the interior zone.
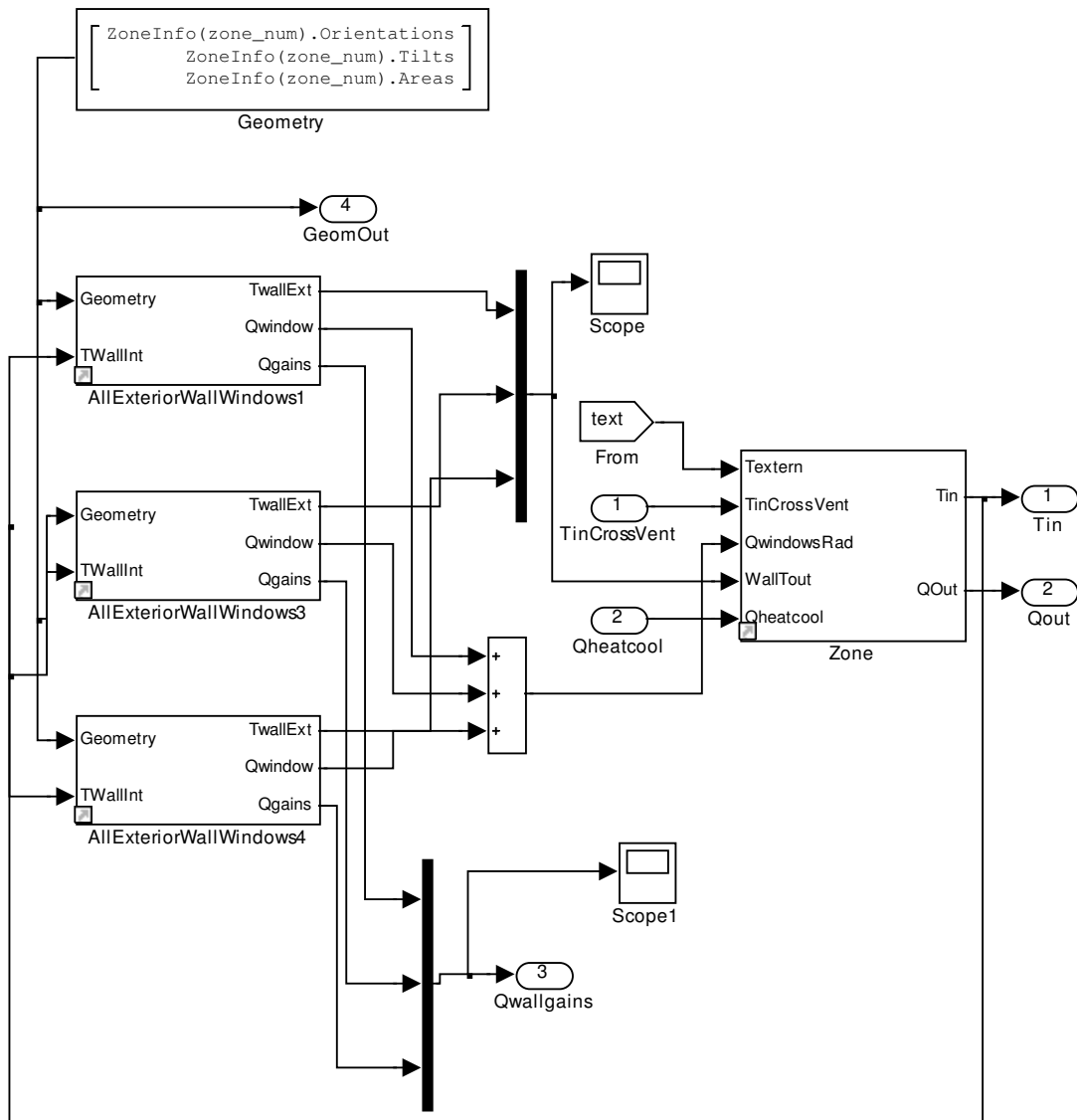
Figure 7.6   Simulink model of entire zone of the Anatomy House

The equivalent exterior temperature of the wall was determined by modelling the exterior portion of the wall as a lumped system. The subsystem model is shown in Figure 7.7. The heat transfer into the section of the wall due to the exterior weather is calculated in the 'Outdoor surface weather conditions, walls and roof' subsystem. The heat transfer from the interior of the building is also calculated based on the temperature difference between the interior temperature and the equivalent exterior temperature. The resulting loads are divided by the lumped capacity of the system and then integrated at each time step to find the equivalent wall temperature. The heat transfer through the windows is also calculated in this subsystem.
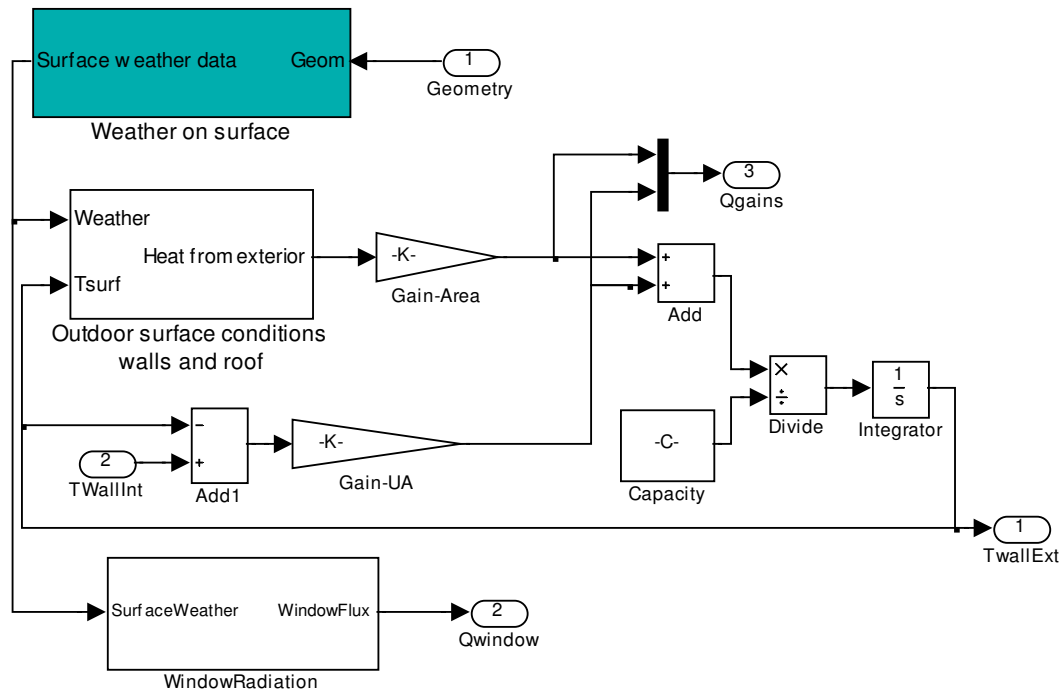
Figure 7.7   Simulink subsystem of the lumped model of the exterior walls and windows

The subsystem of the heating and air conditioning system for the building is shown in Figure 7.8. Seperate systems are modelled for each zone. The output of the heating system is a proportional control calculated from the temperature difference between the interior of the zone and the temperature setpoint. The ventilation system is modelled on the description shown in the previous section with a ventilation rate shown in Figure 7.4. The subsystem output two different heat flows. Qout is the amount of heat transfer directly into the building zone. Qload is the amount of energy used to heat or cool the air in the ventilation system to the desired temperature.
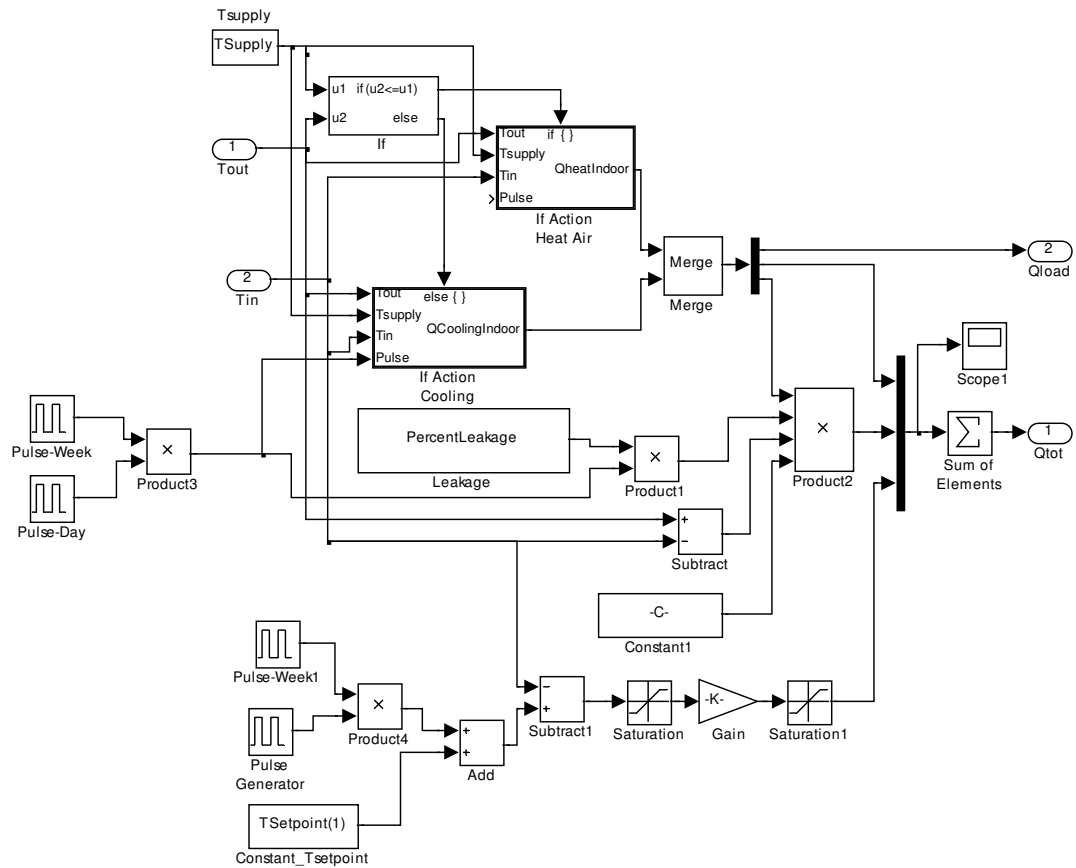
Figure 7.8   Simulink subsystem of the heating and air conditioning model

The final lumped model calculations for the interior zone of the building are done in the subsystem shown in Figure 7.9. This model considers all of the thermal loads that have been described and computes the resulting interior temperature of the zone. The model considers all of the passive loads along with the heating and cooling loads from the mechanical systems.
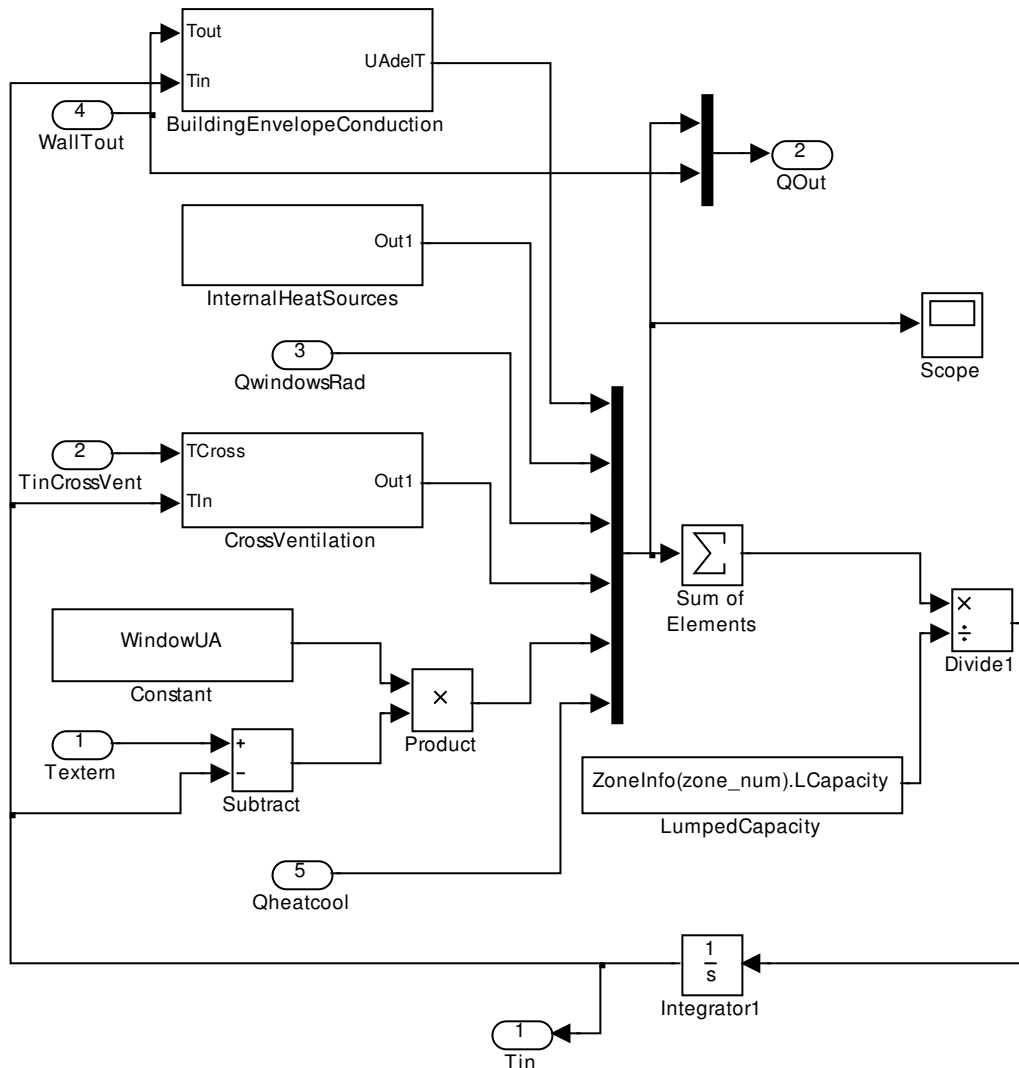
Figure 7.9   Simulink model of single interior zone of the Anatomy House

7.2.3   Preliminary simulations of the Anatomy House

The performance of the Anatomy House model was assessed by comparing the results of the model to the complex model developed in HAM–tools (Sasic Kalagasidis, 2009). This model contained a detailed representation of the building.

In order to verify the performance of the simple model a simulations was run over the period of one year beginning from May 2004. This period was chosen to match the time period of the simulations conducted in the above project.

In order to compare the performance, several graphs of the two models were prepared. Three of the figures are shown below. Note that each of the figures shows a 24 hour moving average. Figure 7.11 shows a comparison of the average indoor temperatures, Figure 7.12 shows the total transmittance through the entire wall, and Figure 7.13 shows a comparison between the total heating and cooling loads.
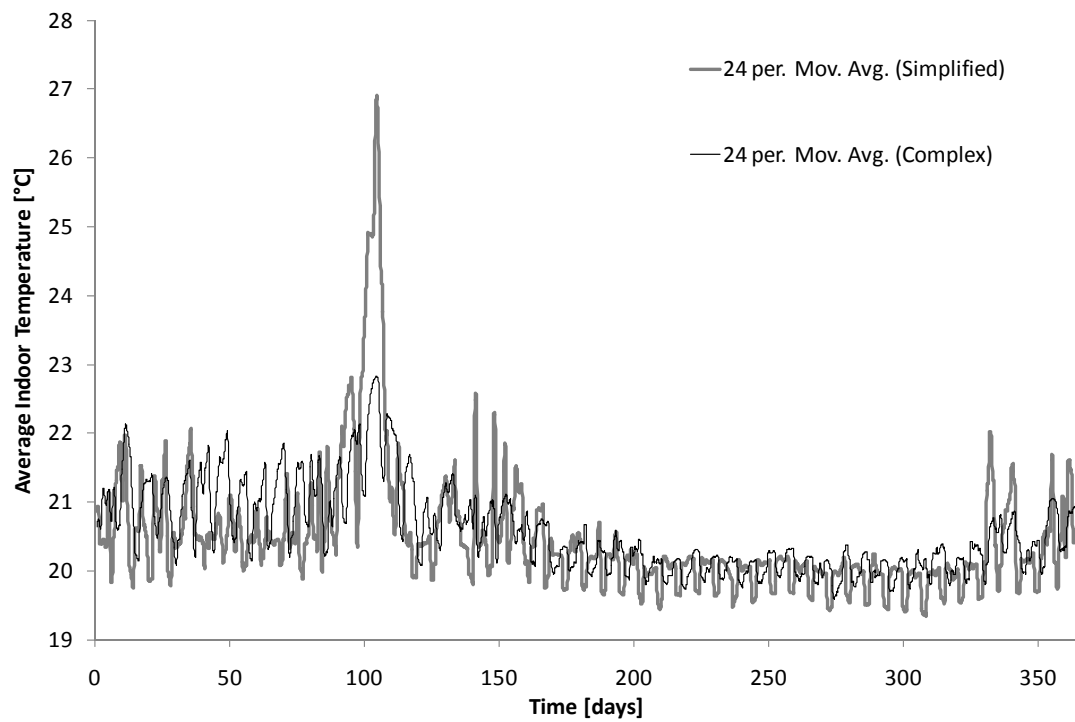
Figure 7.10   Comparison of the 24 hour moving average of indoor temperatures in the simplified and complex models of the Anatomy house
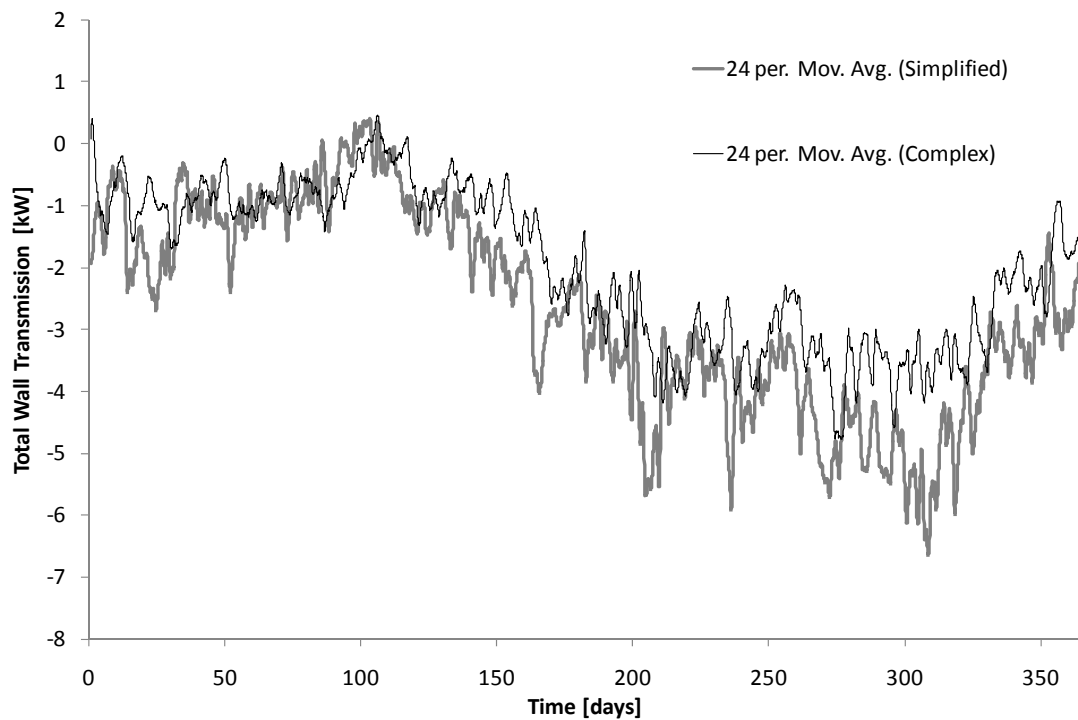
Figure 7.11   Comparison of a 24 hour moving average of the total wall transmission in the simplified and complex models of the Anatomy house
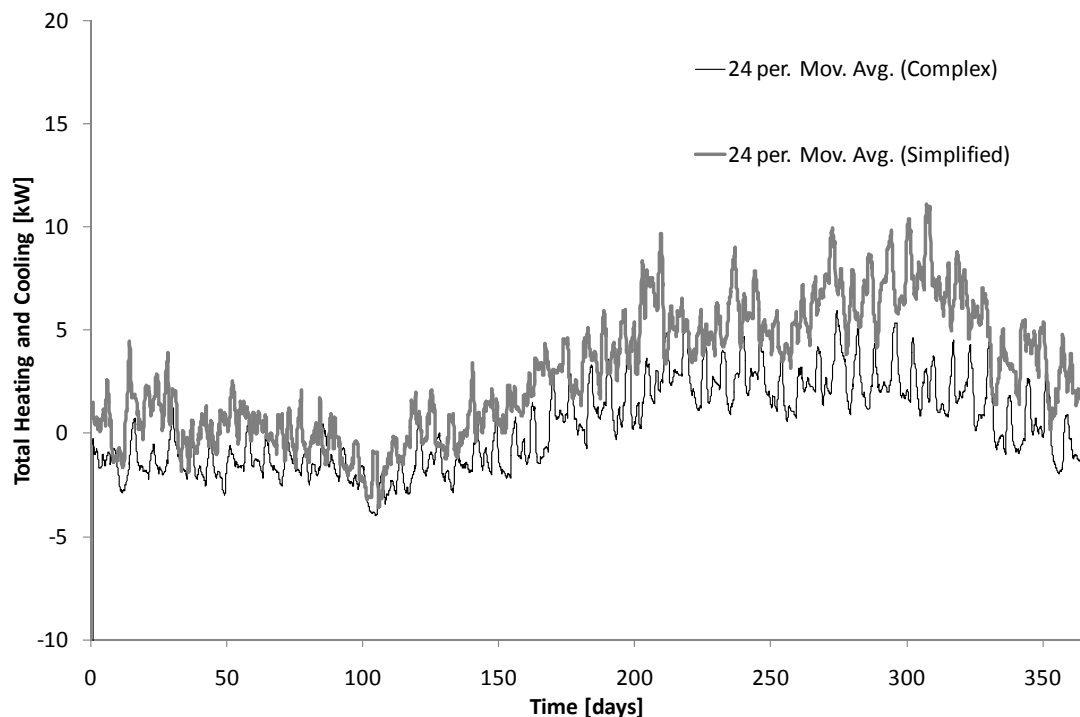
Figure 7.12   Comparison of a 24 hour moving average of the heating and cooling rates in the simplified and complex models of the Anatomy house

The results from the figures do show similar trend in the performance of the building. For example, both models show elevated temperatures in the building around day 100. The simple model however, shows significantly higher temperatures than the complex model. The simple model also shows higher heating loads during the period from approximately day 200 to day 350. These observations could indicate that some of the simplifications made in the model result in some errors in the calculations. It was decided, however, to proceed with the integration of a heat pump model into the simplified model, since the general trends in the performance of the two models were similar.

7.2.4   Development of a model including ground coupled heat pumps

In order to integrate the underground heat pumps into the model of the Anatomy house, the models of the heating systems for each zone were combined into a single model. The air conditioning and ventilation systems were kept as separate objects. In this simulation only the primary heating system of the building was integrated with the ground couple heat pump. The primary heating system is separate from the cooling and air conditioning system and can be simply modified to integrate the underground heat pump system. This was implemented to see the impacts of adding the system to the building. The Simulink model of the building with the combined system is shown in Figure 7.13.
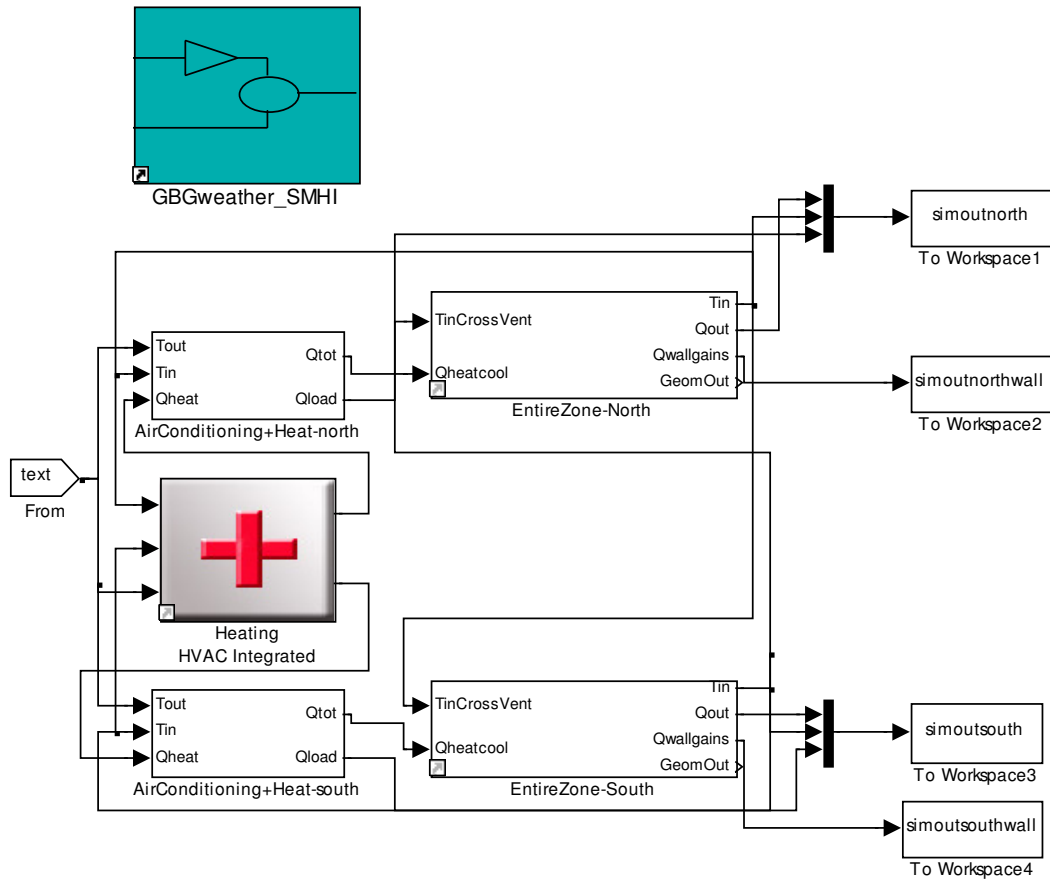
Figure 7.13   Simulink model of the Anatomy House modified to include a heating and cooling system with integrated underground heat exchangers

The subsystem of the integrated heating and cooling system is shown in Figure 7.14. It contains models for the heat pump, underground heat exchanger, the control system, and the heating system. The model then outputs the total heat flows into each zone.

Figure 7.14   Simulink subsystem of the heating and air conditioning model

7.2.5   Simulation of the Anatomy House with BHEs

The model with the integrated BHEs was simulated over the same time period as the previous simulations in order to determine their impact. The overall temperature comparison is shown in Figure 7.15. As expected, the temperature trends compared between the two simulations are quite similar. There are, however, some differences that can be seen in the short term response. Figure 7.16 shows the temperature response over a period of one week.

This figure shows the cyclic nature of the heat pump performance. Due to the slower response and the fact that the heat pump turns off and on due to temperature limits the interior temperature has a more cyclic response. In reality, the cycles shown in the figure could be reduced, but there would most likely be more fluctuations than with the direct heating simulated with the simple model. The simple model simulated direct proportional heating into the interior zone of the building. In reality there would be more fluctuations in this temperature as well, as there would be delays in the response as well.

Table 7.4 shows the total amount of energy output by the heating system integrated with the heat pump along with the total electrical energy used to power the heat pump. The heat pump over the length of the simulation has an average COP of 4.8. This is theoretically quite high, but does give an indication that a large amount of heat is obtained from the ground with a significantly reduced input.

Table 7.4 Cumulative energy use for heating system

Energy [MWh]		COP
Heating Output	HP Electrical	
29.2	6.1	4.8
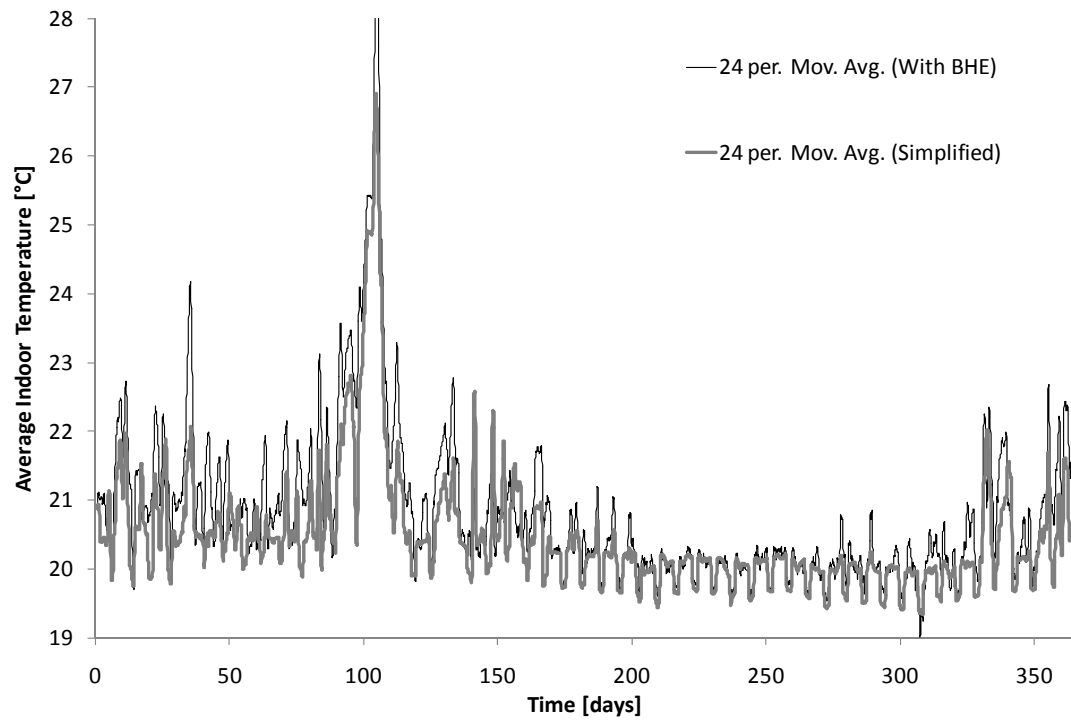
Figure 7.15   Comparison of the average indoor temperatures in the simplified model with and without an underground heat pump system over one year
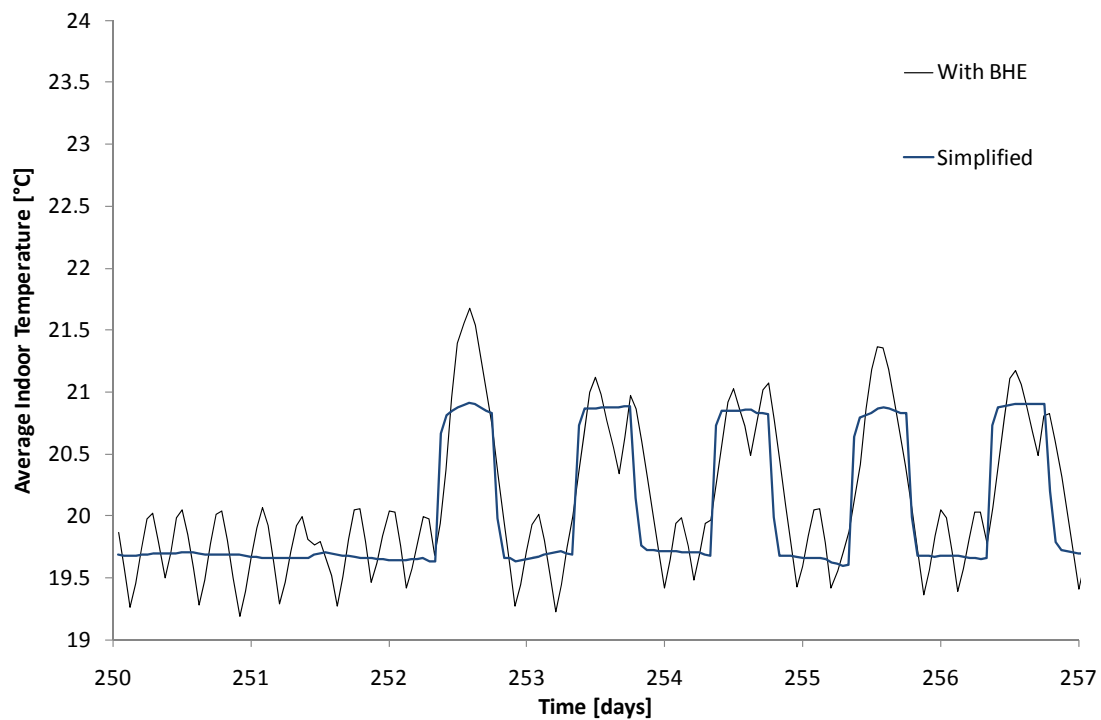
Figure 7.16   Comparison of the average indoor temperatures in the simplified model with and without an underground heat pump system over one week

The temperature of the ground at the centre depth of the pipe at a distance two metres from the pipe over the year of the simulations is shown in Figure 7.17. As expected, since only heating was used, the temperature in the ground decreases over the year once significant use of the heating system has begun.
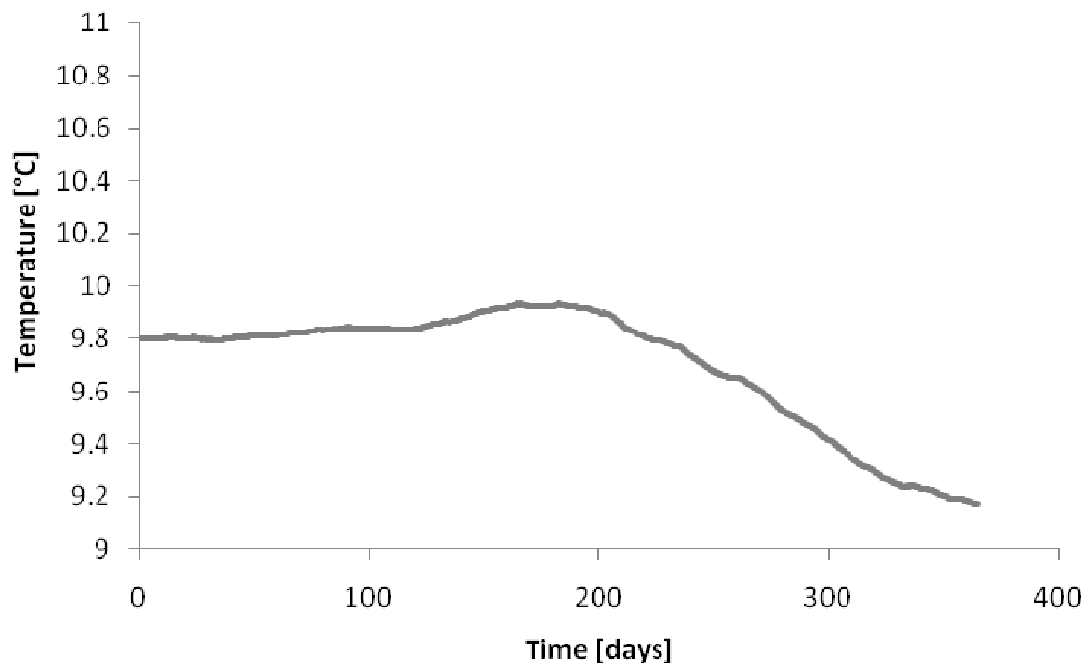
Figure 7.17   Ground temperature at a distance of 2 m from the borehole for the simulation of the Anatomy House.

8 Conclusions

Overall, this project has developed a useful model of underground heat exchangers that can accurately model their performance. In the development of this model, COMSOL has proven to be a useful tool for modelling heat transfer by conduction and convection of the heat exchanger. The use of model coupling allowed the simplification of the three dimensional convection in the pipe into a one dimensional problem. This model has been validated and shown to model the fluid flow through an underground heat exchanger and the resulting change in ground temperatures.

The integration of the model into a building model was accomplished by the use of the model m–file created in COMSOL. An S–function was used to integrate the complete model of the heat exchanger into a model of a building. This has been shown to simulate the performance of a building with an integrated geothermal heat pump system.

The results of the simulations give some preliminary results on the performance of BHEs and their use in energy storage. Fluctuations in the temperature could be seen during the heating and cooling periods. There also was a small decrease in the energy use when the pipe spacing was increased from four to six meters.

The results of the simulations give some preliminary results on the performance of BHEs and their use in energy storage. The results from Gothenburg, particularly, show the fluctuations in the temperature of the ground with underground heat storage. There is a rise in the temperature of the ground during the cooling season and the lowering of temperatures during the heating season can be observed. When designing a system, a balance between the heat removed and the heat stored needs to be maintained

The case study of the Anatomy House shows how the interior temperatures in a building can be affected by the use of a ground couple heat pump can affect the short term temperature response. The results show increased cycles due to the slower response of the heat pump and due to the fact that the heat pump turns off and on due to temperature limits.

9 Recommendations for Further Study

This study has focused on two types of geothermal heat exchanger systems: a horizontal pipe parallel and near the ground surface and BHEs. Other types of systems could also be modelled in a similar manner to model their performance including open loop underground heat exchangers and aquifer systems.

The ground was assumed to be homogeneous with uniform thermal properties. The effect of variation in these properties should be done. The movement of ground water was ignored in this study. It would be valuable to quantify the effects of this on the performance of the systems.

Further study of underground heat storage should also be initiated. This could include the effects of spacing of pipes, different layouts of the pipes , and imbalances between the heating and cooling loads.

10 References

ASHRAE. (2003): *ASHRAE Handbook: HVAC Applications*. SI ed. Atlanta, Ga: American Society of Heating, Refrigerating and Air Conditioning Engineers.

Acuna, J, and B Palm. (2009): Local Conduction Heat Transfer in U–pipe Borehole Heat Exchangers. *Proceedings of the COMSOL Conference 2009 Milan*.

Bejan, A, and A.D Kraus. (2003): *Heat transfer handbook*. Wiley–IEEE.

COMSOL AB. (2008): *COMSOL Multiphysics*.

Claesson, J, and A Dunand. (1983): *Heat extraction from the ground by horizontal pipes: a mathematical analysis*. Swedish Council for Building Research.

Hagentoft, C–E. (2001): *Introduction to building physics*. Lund: Studentlitteratur.

Hellström, G. (1991): *Ground Heat Storage, Thermal Analyses of Duct Storage Systems, Part I: Theory (thesis)*. Lund.

IPCC. (2007): *Climate Change 2007: Synthesis Report. Contribution of Working Groups I, II, and III to the Fourth Assessment Report of the Intergovernmental Panel on Climate Change [Core Writing Team, Pachuari, R.K and Reisinger, A. (eds)]*. IPCC Secretariat, Geneva, Switzerland,

MathWorks, Inc. (1984): *Matlab and Simulink*.

Microsoft. (2010): Bing Maps. *Bing Maps*. http://www.bing.com/maps/.

Rohsenow, W.M, J.P Hartnett, and Young I. Cho. (1998): *Handbook of heat transfer*. McGraw–Hill, May 1.

Sanner, B, C Karytsas, D Mendrinos, and L Rybach. (2003): Current status of ground source heat pumps and underground thermal energy storage in Europe. *Geothermics* 32 (4–6): 579–588.

Sasic Kalagasidis, A. (2004): HAM–Tools–An Integrated Simulation Tool for Heat, Air and Moisture Transfer Analyses in Building Physics.

Sasic Kalagasidis, A. (2009): *Possibilities with weather forecast control of building cooling system in Sweden: a pre–study for the Swedish Meteorological and Hydrological Institute (SMHI)*. Chalmers University of Technology.

van Schijndel, A.W.M. and M.H.M. de Wit. (2003): Advanced simulation of building systems and control with Simulink. In *Eighth International IBPSA Conference, Eindhoven, Netherlands*.

van Schijndel, A.W.M. (2003): ISE (MatLab 6.5 or higher): User Manual. Technische Universiteit Eindhoven.

Appendix A Details of the Model Coupling in COMSOL

COMSOL allows for the coupling of different models to simulate interaction between them. Model Couplings allow a user to map variables from one section in a model to another or also to integrate a variable along curves and map from one entity to another. In COMSOL 3.5a Extrusion coupling variables were used as shown in the figure below. Variables were created for each pipe segment in each model. Using a linear extrusion source vertices and destination vertices were specified.

The heat transfer from the ground to the pipe was specified on the subdomain settings for the pipe model. The heat flow was specified using the coupling variable T_geom1a which would access the temperature in the ground at the same location as the pipe. The heat flow from the pipe to the ground is specified for the equation.
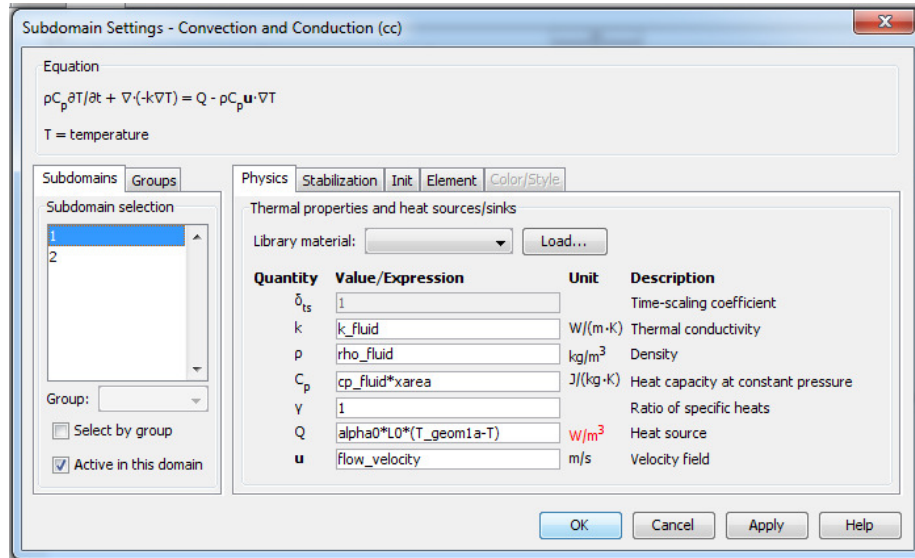
The corresponding flow from the pipe to the ground is done in the model of the ground using the Edge Settings. The weak form of the heat transfer equation is specified along each pipe segment edge in the ground. The equation in this case is:

$$weak = Test(T) \cdot alpha0 \cdot L0 \cdot (T\_geom2a - T)$$

The figure below shows how this is implemented in the COMSOL interface.
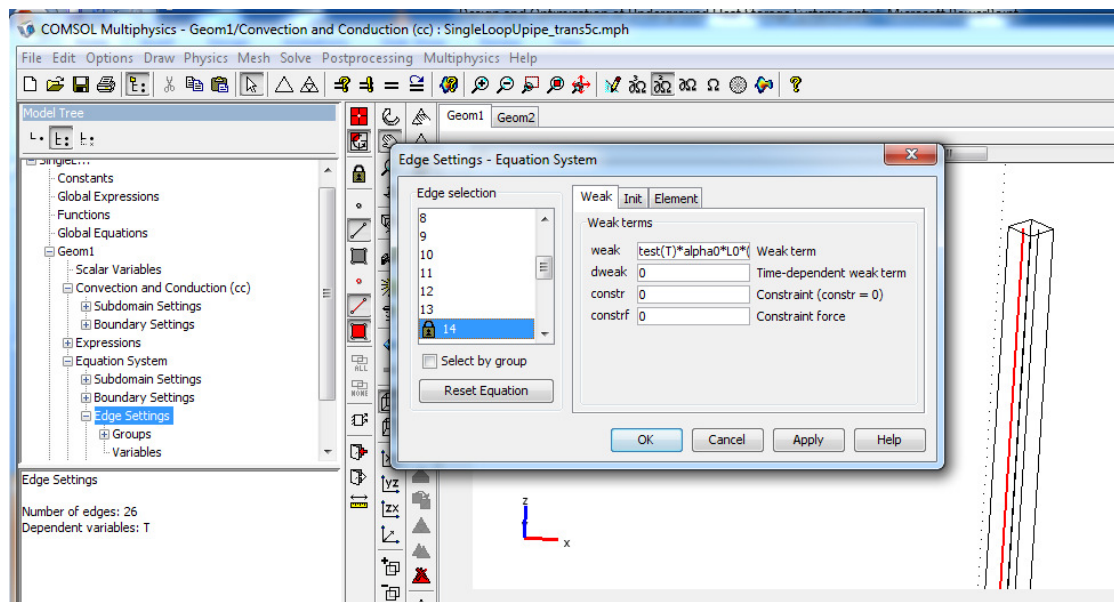
Appendix B   Code for Example S–functions

Code for Initialization of COMSOL Simulation in heat1Dsfun

```
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%  Initialize COMSOL Simulations  %%%
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
% COMSOL version
clear vrsn
vrsn.name = 'COMSOL 3.5';
vrsn.ext = 'a';
vrsn.major = 0;
vrsn.build = 603;
vrsn.rcs = '$Name:  $';
vrsn.date = '$Date: 2008/12/03 17:02:19 $';
fem.version = vrsn;
% Geometry
g1=solid1([0,0.2]);
% Analyzed geometry
clear s
s.objs={g1};
s.name={'I1'};
s.tags={'g1'};
fem.draw=struct('s',s);
fem.geom=geomcsg(fem);
% Initialize mesh
fem.mesh=meshinit(fem);
% Application mode 1
clear appl
appl.mode.class = 'HeatTransfer';
appl.sshape = 2;
appl.assignsuffix = '_ht';
clear bnd
bnd.type = 'T';
bnd.T0 = {288.15,293.15};
bnd.ind = [2,1];
appl.bnd = bnd;
clear equ
equ.C = 1000;
equ.init = 283.15;
equ.k = 0.3;
equ.rho = 2000;
equ.ind = [1];
appl.equ = equ;
fem.appl{1} = appl;
fem.frame = {'ref'};
fem.border = 1;
fem.outform = 'general';
clear units;
units.basesystem = 'SI';
fem.units = units;
% ODE Settings
clear ode
clear units;
units.basesystem = 'SI';
ode.units = units;
fem.ode=ode;
% Multiphysics
fem=multiphysics(fem);
% Extend mesh
fem.xmesh=meshextend(fem);
% Solve problem
fem.sol=femtime(fem, ...
  'solcomp',{'T'}, ...
  'outcomp',{'T'}, ...
  'blocksize','auto', ...
  'tlist',[colon(0,1,2)], ...
  'tout','tlist');
solut=fem.sol.u(:,3);
indout=uint32(size(solut,1)/2);
tout=solut(indout)-273.15;
disp('prevtime tout');
%endfunction
```

Code for the Update of COMSOL Simulation in heat1Dsfun at each time step

```matlab
function Update(block)
  global fem
  global tout
  global solut
  persistent prevtime
  stoptime=block.CurrentTime;
  if stoptime~=0
    steptime=(stoptime-prevtime)/2;
    ext_temp=[num2str(block.InputPort(1).Data+273.15) '[K]'];
    % Set Exterior Temperature
    appl=fem.appl{1};
    clear bnd
    bnd.type = 'T';
    bnd.T0 = {288.15,ext_temp};
    bnd.ind = [2,1];
    appl.bnd = bnd;
    fem.appl{1}=appl;
    % Multiphysics
    fem=multiphysics(fem);
    % Extend mesh
    fem.xmesh=meshextend(fem);
    % Solve problem
    fem.sol=femtime(fem, ...
      'solcomp',{'T'}, ...
      'outcomp',{'T'}, ...
      'blocksize','auto', ...
      'tlist',[colon(prevtime,steptime,stoptime)], ...
      'tout','tlist');
    solut=fem.sol.u(:,3);
    prevtime=stoptime;
  else
    prevtime=0;
  end
  indout=uint32(size(solut,1)/2);
  tout=solut(indout)-273.15;
  disp([prevtime tout]);
```

Appendix C   S–function of Underground Heat Exchangers

Model of Horizontal Pipe in the Ground

Function Name: SinglePipeHoriz_sfun.m

The model developed in Section 4.3.1 was saved as a model m–file and then modified into an S–function.   The function consists of the main model m–file 'SinglePipeHoriz_sfun.m' along with a file containing a list of constants and parameter, 'SinglePipeHoriz_constants.m'.   The constants and parameters file is shown below.

```matlab
% SinglePipeHoriz_constants.m
%Static Parameters that are parameters of s-function
horiz_pipe.pipelength=200; %length of the pipe
horiz_pipe.pipedepth=2;    % depth of pipe from the ground surface
horiz_pipe.soilspace=20;   % space of ground around the pipe in all
                           % horizontal directions and in depth

% Variables for Horizontal Pipe Simulation
horiz_pipe.L0='0.207'; %circumference of the pipe [m]
horiz_pipe.alpha0='140'; %heat transfer coefficient from fluid to soil [W/(m2K)]
horiz_pipe.Tgin='9.8[degC]'; %initial ground temperature [°C]
horiz_pipe.Ti='9.8[degC]'; %initial inlet temperature of the pipe [°C]
horiz_pipe.xarea='0.002827'; %cross sectional area of the pipe [°C]
horiz_pipe.flow_velocity='0.2'; %velocity of the fluid in the pipe [m/s]
horiz_pipe.Tsoiltop='9.8[degC]'; % initial exterior temperature at soil surface [°C]
horiz_pipe.Tsoilbottom='12[degC]'; %initial and long term temperature of ground [°C]
horiz_pipe.k_ground='2[W/(m*K)]';  % conductivity of the ground [W/(m*K)]
horiz_pipe.rho_ground='1500[kg/m^3]'; % density of the ground [kg/m^3]
horiz_pipe.cp_ground='800[J/(kg*K)]'; % heat Capacity of the ground [J/(kg*K)]
horiz_pipe.k_fluid='0[W/(m*K)]'; % conductivity of the pipe fluid [W/(m*K)]
horiz_pipe.rho_fluid='1000[kg/m^3]'; % density of the pipe fluid [kg/m^3]
horiz_pipe.cp_fluid='4200[J/(kg*K)]'; % heat Capacity of the pipe fluid [J/(kg*K)]

% Variables for Mesh
horiz_pipe.mesh_geom1_hauto=7; % Auto meshing scale (1= very fine to 8=very coarse)
horiz_pipe.mesh_geom2_hmax=5; %maximum size of elements [m]

% Variables for Output
horiz_pipe.outputdetails=1; % include output of soil temperatures(1=true)
```

```matlab
function SinglePipeHoriz_sfun(block)
% Level-2 M file S-Function for a model of a single horizontal pipe in
% the ground.
setup(block);
%endfunction

function setup(block)
  %% Register number of dialog parameters
  block.NumDialogPrms = 0;
  %% Register number of input and output ports
  block.NumInputPorts  = 3;
  % 3 dynamic inputs -  1. Pipe inlet fluid Temperature [°C]
  %                     2. Surface temperature of the ground [°C]
  %                     3. Velocity of the fluid in the pipe [m/s]
  block.NumOutputPorts = 1;
  %% Setup functional port properties to dynamically inherited.
  block.SetPreCompInpPortInfoToDynamic;
  block.SetPreCompOutPortInfoToDynamic;
  block.InputPort(1).DirectFeedthrough = true;
  %% Set block sample time to inherited
  block.SampleTimes = [-1 0];
  %% Run accelerator on TLC
  block.SetAccelRunOnTLC(false);
  %% Register methods
```

```matlab
    block.RegBlockMethod('PostPropagationSetup',    @DoPostPropSetup);
    block.RegBlockMethod('Outputs',                 @Output);
    block.RegBlockMethod('Update',                  @Update);
    % Global variables
    global tout
    global solut
    global xfem
    global flagoutputdetails
    % call file with constant parameters of model
    SinglePipeHoriz_constants
    flagoutputdetails=horiz_pipe.outputdetails;
      %% Control Parameters
    pipelength= horiz_pipe.pipelength;
    pipedepth= horiz_pipe.pipedepth;
    soilspace= horiz_pipe.soilspace;
    strdomainwidth=num2str(soilspace*2);
    strsoildepth=num2str(pipedepth+soilspace);
    stryloc=num2str(-soilspace);
    strdepthloc=num2str(-pipedepth-soilspace);
    strdomainlength=num2str(pipelength+soilspace*2);
    %% Beginning of COMSOL generated code
    %% COMSOL version
    clear vrsn
    vrsn.name = 'COMSOL 3.5';
    vrsn.ext = 'a';
    vrsn.major = 0;
    vrsn.build = 608;
    vrsn.rcs = '$Name: v35ap $';
    vrsn.date = '$Date: 2009/05/11 07:38:49 $';
    xfem.version = vrsn;
    % turn off COMSOL Simulation progress window
    flreport('off')
    %% Constants - modified from COMSOL generated code
    xfem.const = {'L0',horiz_pipe.L0, ...
      'alpha0',horiz_pipe.alpha0, ...
      'Tgin',horiz_pipe.Tgin, ...
      'Ti',horiz_pipe.Ti, ...
      'xarea',horiz_pipe.xarea, ...
      'flow_velocity',horiz_pipe.flow_velocity, ...
      'Tsoiltop',horiz_pipe.Tsoiltop, ...
      'Tsoilbottom',horiz_pipe.Tsoilbottom, ...
      'Tsoiltop2',horiz_pipe.Tsoiltop, ...
      'k_ground',horiz_pipe.k_ground, ...
      'rho_ground',horiz_pipe.rho_ground, ...
      'cp_ground',horiz_pipe.cp_ground, ...
      'k_fluid',horiz_pipe.k_fluid, ...
      'rho_fluid',horiz_pipe.rho_fluid, ...
      'cp_fluid',horiz_pipe.cp_fluid};
    %% Geometry and Mesh
    % Geometry 1
    flclear fem
    clear draw
g1=block3(strdomainlength,strdomainwidth,strsoildepth,'base','corner','pos',{'0',stryl
oc,strdepthloc},'axis',{'0','0','1'},'rot','0');
    g2=curve3([soilspace,pipelength+soilspace],[0,0],[-pipedepth,-pipedepth]);
    draw.s.objs = {g1};
    draw.s.name = {'BLK1'};
    draw.s.tags = {'g1'};
    draw.c.objs = {g2};
    draw.c.name = {'B1'};
    draw.c.tags = {'g2'};
    fem.draw = draw;
    fem.geom = geomcsg(fem);
    fem.mesh=meshinit(fem, ...
      'hauto',horiz_pipe.mesh_geom1_hauto);
    xfem.fem{1}=fem;
    % Geometry 2
    flclear fem
    clear draw
    g3=solid1([0,pipelength]);
    draw.s.objs = {g3};
    draw.s.name = {'I1'};
    draw.s.tags = {'g3'};
    fem.draw = draw;
    fem.geom = geomcsg(fem);
    fem.mesh=meshinit(fem, ...
```

```
    'hmax',horiz_pipe.mesh_geom2_hmax, ...
    'hgrad',1.2);
xfem.fem{2}=fem;
%% Set up Physics for Geom1
fem=xfem.fem{1};
% Application mode 1
clear appl
appl.mode.class = 'FlConvCond';
appl.sshape = 2;
appl.assignsuffix = '_cc';
clear bnd
bnd.type = {'q0','T','T'};
bnd.T0 = {273.15,'Tsoilbottom','Tsoiltop'};
bnd.ind = [1,1,2,3,1,1];
appl.bnd = bnd;
clear equ
equ.C = 'cp_ground';
equ.rho = 'rho_ground';
equ.init = 'Tgin';
equ.k = 'k_ground';
equ.ind = [1];
appl.equ = equ;
fem.appl{1} = appl;
fem.frame = {'ref'};
% Shape functions
fem.shape = {'shlag(2,''T'')'};
fem.border = 1;
fem.outform = 'general';
% Equation form
fem.form = 'general';
clear units;
units.basesystem = 'SI';
fem.units = units;
% Edge settings
clear edg
edg.weak = {0,'test(T)*alpha0*L0*(T_geom2-T)'};
edg.shape = 1;
edg.ind = [1,1,1,1,1,1,1,1,2,1,1,1,1];
edg.dim = {'T'};
fem.edg = edg;
% Coupling variable elements
clear elemcpl
% Extrusion coupling variables
clear elem
elem.elem = 'elcplextr';
elem.g = {'1','2'};
src = cell(1,2);
clear edg
edg.expr = {{{},'T'}};
edg.map = {{'1','1'}};
edg.ind = {{'1','2','3','4','5','6','7','8','10','11','12','13'},{'9'}};
src{1} = {{},edg,{},{}};
src{2} = {};
elem.src = src;
geomdim = cell(1,2);
geomdim{1} = {};
clear equ
equ.map = {{'2'}};
equ.ind = {{'1'}};
geomdim{2} = {{},equ};
elem.geomdim = geomdim;
elem.var = {'T_geom1'};
map = cell(1,2);
clear submap
submap.type = 'unit';
map{1} = submap;
clear submap
submap.type = 'linear';
submap.sg = '2';
submap.sv = {'1','2'};
submap.dg = '1';
submap.dv = {'5','6'};
map{2} = submap;
elem.map = map;
elemcpl{1} = elem;
fem.elemcpl = elemcpl;
xfem.fem{1} = fem;
```

```
%% Set up Physics for Geom2
fem=xfem.fem{2};
% Application mode 1
clear appl
appl.mode.class = 'FlConvCond';
appl.sshape = 2;
appl.assignsuffix = '_cc';
clear bnd
bnd.type = {'qc','T'};
bnd.T0 = {273.15,'Ti'};
bnd.ind = [2,1];
appl.bnd = bnd;
clear equ
equ.C = 'cp_fluid*xarea';
equ.init = 'Tgin';
equ.k = 'k_fluid';
equ.u = 'flow_velocity';
equ.Q = 'alpha0*L0*(T_geom1-T)';
equ.rho = 'rho_fluid';
equ.ind = [1];
appl.equ = equ;
fem.appl{1} = appl;
fem.frame = {'ref'};
fem.border = 1;
fem.outform = 'general';
clear units;
units.basesystem = 'SI';
fem.units = units;
% Coupling variable elements
clear elemcpl
% Extrusion coupling variables
clear elem
elem.elem = 'elcplextr';
elem.g = {'1','2'};
src = cell(1,2);
src{1} = {};
clear equ
equ.expr = {{'T'}};
equ.map = {{'1'}};
equ.ind = {{'1'}};
src{2} = {{},equ};
elem.src = src;
geomdim = cell(1,2);
clear edg
edg.map = {{{},'2'}};
edg.ind = {{'1','2','3','4','5','6','7','8','10','11','12','13'},{'9'}};
geomdim{1} = {{},edg,{},{}};
geomdim{2} = {};
elem.geomdim = geomdim;
elem.var = {'T_geom2'};
map = cell(1,2);
clear submap
submap.type = 'unit';
map{1} = submap;
clear submap
submap.type = 'linear';
submap.sg = '1';
submap.sv = {'5','6'};
submap.dg = '2';
submap.dv = {'1','2'};
map{2} = submap;
elem.map = map;
elemcpl{1} = elem;
fem.elemcpl = elemcpl;
xfem.fem{2} = fem;
%% Final Settings and Solve
% ODE Settings
clear ode
clear units;
units.basesystem = 'SI';
ode.units = units;
xfem.ode=ode;
% Multiphysics
xfem=multiphysics(xfem, ...
  'edl',{[1,2,3,4,5,6,7,8,10,11,12,13]});
% Extend mesh
xfem.xmesh=meshextend(xfem);
```

```matlab
  % Solve problem
  xfem.sol=femtime(xfem, ...
    'symmetric','off', ...
    'solcomp',{'T'}, ...
    'outcomp',{'T'}, ...
    'blocksize','auto', ...
    'tlist',[colon(0,0.1,0.2)], ...
    'rtol',0.001, ...
    'tout','tlist', ...
    'atol',{'0.00010'}, ...
    'linsolver','gmres');
  %%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
  solut=xfem.sol.u(:,3);
  tout=solut(size(solut,1));
  disp('Output of SinglePipeHoriz_sfun');
  disp('Time[days]      Pipe Return Temperature[°C]');
%endfunction

function DoPostPropSetup(block)
  %% Setup Dwork
  block.NumDworks = 1;
  block.Dwork(1).Name = 'model';
  block.Dwork(1).Dimensions      = 1;
  block.Dwork(1).DatatypeID      = 0;
  block.Dwork(1).Complexity      = 'Real';
  block.Dwork(1).UsedAsDiscState = true;
%endfunction

%% Update function updates S-function at each time step
function Update(block)
    %global model
    global tout
    global solut
    global xfem
    global flagoutputdetails

    persistent prevtime
    persistent outputpoints
    persistent outputpointsx
    stoptime=block.CurrentTime;
    if stoptime~=0
        steptime=(stoptime-prevtime);
        pipe_inlet_temp=[num2str(block.InputPort(1).Data+273.15) '[K]'];
        ground_surface_temp=[num2str(block.InputPort(2).Data+273.15) '[K]'];
        pipe_flow_velcity=[num2str(block.InputPort(3).Data)];
        % Set Pipe Inlet Temperature
        fem=xfem.fem{2};
        appl=fem.appl{1};
        clear bnd
        bnd.type = {'qc','T'};
        bnd.T0 = {273.15,pipe_inlet_temp};
        bnd.ind = [2,1];
        appl.bnd = bnd;
        % Set Pipe Flow Velocity
        equ=appl.equ;
        equ.u = pipe_flow_velcity;
        appl.equ = equ;
        fem.appl{1} = appl;
        xfem.fem{2} = fem;
        % Set Ground Surface Temperature
        fem=xfem.fem{1};
        appl=fem.appl{1};
        clear bnd
        bnd.type = {'q0','T','T'};
        bnd.T0 = {273.15,'Tsoilbottom',ground_surface_temp};
        bnd.ind = [1,1,2,3,1,1];
        appl.bnd = bnd;
        fem.appl{1} = appl;
        xfem.fem{1}=fem;
        % Multiphysics
        xfem=multiphysics(xfem, ...
            'edl',{[1,2,3,4,5,7,8,9,10,11,12,13]});
        % Extend mesh
        xfem.xmesh=meshextend(xfem);
        xfem.sol=femtime(xfem, ...
                'symmetric','off', ...
                'solcomp',{'T'}, ...
```

```matlab
                    'outcomp',{'T'}, ...
                    'blocksize','auto', ...
                    'tlist',[colon(prevtime,steptime,stoptime)], ...
                    'init',solut,...
                    'rtol',0.0001, ...
                    'tout','tlist', ...
                    'atol',{'0.000010'}, ...
                    'linsolver','gmres');
            solut=xfem.sol.u(:,2);
            prevtime=stoptime;
            if flagoutputdetails==1
                [outputdata,pe]=postinterp(xfem,'T',outputpoints);
                outputdata=outputdata-273.15;
                fid =fopen('SingleHorizPipe_ground_output.txt', 'a+');
                fprintf(fid,'%e\t',stoptime);
                fprintf(fid, '%e\t', outputdata);
                fprintf(fid, '\r');
                fclose(fid);
                [outputdata,pe]=postinterp(xfem,'T',outputpointsx,'Geomnum',2);
                outputdata=outputdata-273.15;
                fid =fopen('InfSingleUPipe_pipe_output.txt', 'a+');
                fprintf(fid,'%e\t',stoptime);
                fprintf(fid, '%e\t', outputdata);
                fprintf(fid, '\r');
                fclose(fid);
            end

        else
            prevtime=0;
            if flagoutputdetails==1
                SinglePipeHoriz_constants
                pipelength= horiz_pipe.pipelength;
                pipedepth= horiz_pipe.pipedepth;
                soilspace= horiz_pipe.soilspace;
                [outputcoord.x,outputcoord.y,outputcoord.z]=meshgrid(  ...
                    soilspace:(pipelength/7):(soilspace+pipelength), ...
                    0:(soilspace/6):(soilspace/2), ...
                    ((-soilspace/2)-pipedepth):soilspace/6:-pipedepth);
                outputpoints=[outputcoord.x(:)';outputcoord.y(:)';outputcoord.z(:)'];
                outputpointsx=[soilspace:(pipelength/7):(soilspace+pipelength)];
                fid =fopen('SingleHorizPipe_ground_output.txt', 'a+');
                fprintf(fid, 'Sfunction Output: Ground Temperature \r')
                for iline=1:3
                    fprintf(fid, '\t')
                    fprintf(fid, '%e\t', outputpoints(iline,:));
                    fprintf(fid, '\r');
                end
                fclose(fid);
                fid =fopen('SingleHorizPipe_pipe_output.txt', 'a+');
                fprintf(fid, 'Sfunction Output: Pipe Temperature [°C] \r')
                fprintf(fid, '\t')
                fprintf(fid, '%e\t', outputpointsx(:));
                fprintf(fid, '\r');
                fclose(fid);
            end
        end
        tout=solut(size(solut,1));
        disp([num2str(prevtime/3600/24) '     ' num2str(tout)]);
%endfunction

function Output(block)
    global tout
    block.OutputPort(1).Data = tout-273.15;
%endfunction
```

Model of BHEs

The models developed in Section 4.3.2 were saved as a model m–file and then modified into an S–function. The functions each consists of the main model m–file along with a file containing a list of constants and parameters as shown in Table 4.6. An example of the constants and parameters file is shown below. The main model S–functions are not shown, as they are similar to the model of the horizontal pipe shown above.

File with Constants: InfSingleUpipe_constants.m

```
%General Parameters
U_pipe.upipelength=200; % length of the bore hole [m]
U_pipe.upipecover=2;    % depth of pipe from the ground surface [m]
U_pipe.soilspace=20;    % space of ground around the pipe in all
                        % horizontal directions and in depth [m]
U_pipe.boreholediameter=0.13; %diameter of the borehole [m]
U_pipe.relativepipelocation=0.8; % relative location of
    % pipes in borehole [-] ((relativepipelocation=distance of pipe from
    % centre of bore hole)/(radius of borehole))
U_pipe.Tsoilinitialdegc=9.8; % initial ground temperature [°C]
U_pipe.pipesspace=6;    % space between pipes [m]

% Variables for Simulation
U_pipe.L0='0.1257'; %circumference of the pipe [m]
U_pipe.alpha0='164'; %heat transfer coefficient from fluid to soil [W/(m2K)]
U_pipe.Tgin='10[degC]'; %initial ground temperature [°C]
U_pipe.Ti='10[degC]'; %initial inlet temperature of the pipe [°C]
U_pipe.xarea='0.0012566'; %cross sectional area of the pipe [°C]
U_pipe.flow_velocity='0.6'; %velocity of the fluid in the pipe [m/s]
U_pipe.Tsoiltop='9.8[degC]'; % initial exterior temperature at soil surface [°C]
U_pipe.Tsoilbottom='9.8[degC]'; %initial and long term temperature of ground [°C]
U_pipe.k_ground='3.5[W/(m*K)]';   % conductivity of the ground [W/(m*K)]
U_pipe.rho_ground='2000[kg/m^3]'; % density of the ground [kg/m^3]
U_pipe.cp_ground='1000[J/(kg*K)]'; % heat Capacity of the ground [J/(kg*K)]
U_pipe.k_grout='2.0[W/(m*K)]';     % conductivity of the grout [W/(m*K)]
U_pipe.rho_grout='2000[kg/m^3]';   % density of the grout [kg/m^3]
U_pipe.cp_grout='1000[J/(kg*K)]';  % heat Capacity of the grout [J/(kg*K)]
U_pipe.k_fluid='0[W/(m*K)]';       % conductivity of the pipe fluid [W/(m*K)]
U_pipe.rho_fluid='1000[kg/m^3]';   % density of the pipe fluid [kg/m^3]
U_pipe.cp_fluid='4200[J/(kg*K)]'; % heat Capacity of the pipe fluid [J/(kg*K)]

% Variables for Mesh
% Variables for 3D ground mesh
U_pipe.mesh_geom1_hauto=8; % Auto meshing scale (1= very fine to 8=very coarse)
U_pipe.mesh_geom1_xscale=20; % relative scaling in x direction
U_pipe.mesh_geom1_yscale=20; % relative scaling in y direction
% Variables for 1D pipe elements
U_pipe.mesh_geom2_hmax=4; %maximum size of elements [m]

% Variables for Ground Temperature Output
U_pipe.outputdetails=1; % include output of soil temperatures(1=true)
```