

# Numerical methods for modeling geothermal energy extraction

Master of Science Thesis in Applied Mathematics

Anders Solstrand Fosen

Department of Mathematics  
University of Bergen



November 20, 2009



# Preface

First of all I want to thank my supervisors Jan Martin Nordbotten and Inga Berre. I have enjoyed many hours at Knøderen with Jan and his great enthusiasm. I have also had the great pleasure of going to a work shop at Svalbard and to visit universities in Europe with him. My friends at the math department have made my study time great, and I wish to thank you all for that. Good luck with your master thesis next semester! (I did beat you!) I am truly grateful for my mother and father, they are fantastic parents, so a huge thanks goes to them. To have two brothers that I enjoy to have as company, is also something I appreciate highly. Finally I especially want to thank my girlfriend Helene.



# Contents

<b>Preface</b>	<b>I</b>
<b>Contents</b>	<b>III</b>
<b>Outline</b>	<b>1</b>
<b>1 Geothermal energy extraction</b>	<b>3</b>
1.1 Geothermal energy . . . . .	3
1.1.1 Environmental advantages . . . . .	4
1.2 Heat sources . . . . .	5
1.3 Hydrothermal systems . . . . .	5
1.3.1 Enhanced geothermal systems . . . . .	5
<b>2 Reservoir mechanics</b>	<b>9</b>
2.1 Porous media . . . . .	9
2.1.1 Porosity . . . . .	10
2.1.2 Representative elementary volume . . . . .	10
2.1.3 Darcy's law . . . . .	11
2.1.4 Permeability . . . . .	12
2.2 Conservation laws . . . . .	13
2.3 Temperature and enthalpy . . . . .	14
<b>3 Mathematical model</b>	<b>17</b>
3.1 Fine-scale model . . . . .	18
3.1.1 Blocks and fractures . . . . .	18
3.1.2 Local conservation law . . . . .	20
3.2 Reservoir scale model . . . . .	22
3.2.1 Reservoir scale conservation law . . . . .	22
3.2.2 Flux expressions . . . . .	22
3.2.3 Combining the flux terms to model equations . . . . .	25
3.3 Initial and boundary conditions . . . . .	25

<b>4</b>	<b>Numerical model</b>	<b>29</b>
4.1	Two dimensional representation . . . . .	30
4.2	Grid . . . . .	30
4.3	Numerical implementation of the block layers . . . . .	30
4.4	Finite element methods . . . . .	31
4.4.1	Variational form for the heat equation . . . . .	34
4.5	Finite difference methods . . . . .	35
4.6	Initial and boundary conditions . . . . .	36
4.7	Numerical solution of the model equations . . . . .	36
4.7.1	All terms implicit . . . . .	37
4.7.2	Heat exchange term explicit . . . . .	38
4.7.3	Diffusion term explicit . . . . .	38
4.7.4	Transport term explicit . . . . .	38
4.7.5	Special cases with only one flux term . . . . .	38
4.7.6	Courant–Friedrichs–Lewy condition . . . . .	41
<b>5</b>	<b>Results</b>	<b>43</b>
5.1	Coefficients and the subsequent CFL conditions . . . . .	44
5.1.1	Physical interpretation of the grid and layers . . . . .	46
5.2	Adaptive strategy . . . . .	46
5.3	Evaluation of results . . . . .	47
5.3.1	Norms . . . . .	48
5.3.2	Interpolation . . . . .	48
5.3.3	Convergence . . . . .	48
5.3.4	Solution plots . . . . .	51
5.3.5	Comparison of the solvers . . . . .	53
<b>6</b>	<b>Summary and conclusion</b>	<b>61</b>
6.1	Summary . . . . .	61
6.2	Conclusion . . . . .	62
6.3	Possible additions . . . . .	63
	<b>Bibliography</b>	<b>65</b>

# Outline

Modeling of flow in porous media is an important scientific research area, and has been so for decades. It is also one of the major topics within applied mathematics. Models for flow in porous media are for example important in the oil industry, in groundwater hydrology and in geothermal energy extraction. In this thesis we are building both a mathematical and a numerical geothermal model. To understand the processes that happens in geothermal reservoirs far below the earth's surface, good models are needed. The long term reservoir behavior is important when the economical feasibility of a geothermal project is determined. Good models are needed to determine the long term behavior.

To model flow in porous media, there are several steps that needs to be done. The first step is to obtain and understand the background knowledge, such as theory from reservoir mechanics, that is needed to build a model. Some of this knowledge is common for all the different topics that use flow in porous media models. Other parts of the theory are more specific and connected to an application. When sufficient knowledge has been obtained, the next step is to use it to create a mathematical model for flow in porous media. When this has been done, it is time to implement a numerical model that is based on the mathematical model.

To obtain a numerical model, it is common to discretize the continuous model expressions in the mathematical model. We try to retain the essential properties of the continuous model expressions when we discretize them. Discretizing model expressions often leads to a linear system that can be solved by numerical equation solvers. The main focus in this thesis is the discretization of the equation terms, both spatial and in time. We will use a finite element method to spatially discretize the diffusion term in our model equations. A finite difference method will be used to discretize the advection term in space. An equation term can be solved with either explicit or implicit time discretization. When a term is solved explicitly it is solved at the start of each time step, using the the previous equation values. Solving the term implicitly, the term is calculated at the end of each time step. We will try to create an adaptive strategy that decides which terms that should

be solved with explicit time discretization.

The thesis is split into 6 chapters. Chapter 1 will work as a background for the rest of the thesis, and is dedicated to geothermal energy extraction. As we build a model for geothermal energy extraction, it is important to have some knowledge of how a geothermal reservoir works. In Chapter 2 we will go through the theory from reservoir mechanics that is relevant for this thesis. We will explain the terms porous media, porosity, representative elementary volume, permeability, homogeneity, and isotropy. We will also explain Darcy's law and the general conservation law. At the end of the chapter we will look at the similarities and differences between the physical properties enthalpy and temperature.

The mathematical model is built on a local and a reservoir scale conservation law for enthalpy, and we create this model in Chapter 3. We see our reservoir as blocks of rock, with fractures that are filled with water between them. The local conservation law will model the heat transfer in one block and the fractures near it. To do this we will split the block up into layers, and there will be an enthalpy flux between them. The reservoir scale conservation law models enthalpy transfer in the entire reservoir, between the different blocks and fractures.

The flux term in the reservoir scale conservation law consists of three different fluxes, the diffusive flux, the advective flux and the heat exchange flux. The diffusive flux is diffusion driven and transfers enthalpy between the different blocks and between the different fractures. The heat exchange flux transfers enthalpy inside each block of porous media, and this reservoir scale flux is based on the local flux inside each block. The heat exchange flux is also a diffusive flux, but it only transfers enthalpy between the block layers, not between the different blocks as the diffusive flux does. The advective flux, the result of water flowing in the fractures, transfers enthalpy between the different fractures in the reservoir.

In Chapter 3 we establish expressions for these fluxes, and combine them into model equations. We also establish initial and boundary conditions for our model. In Chapter 4 our numerical model is created. Here we discretize our model equations from Chapter 3. When we have discretized the equations, we implement them in our numerical model. We then explain how we solve the different terms of the equations numerically. The solvers discretize the equation terms either with explicit or implicit time discretization.

In Chapter 5 we go through the simulation results. First we will go through the methods we use to compare the results, then we do a comparison. We create an adaptive strategy, that decides which terms that should be solved with explicit time discretization. We then show why using explicit time discretization in some of the terms in the modeling equations make sense. In Chapter 6 we summarize what we have done in our research, go through possible improvements to our model, and come to a conclusion of the research we have done.

# Chapter 1

## Geothermal energy extraction

This chapter, dedicated to different aspects of geothermal energy extraction, will work as a background for the rest of the thesis. There will be a short explanation of what geothermal energy is and how it is used. Next, there will be a short look at the benefits of geothermal energy and an explanation of what a Hydrothermal System and an Enhanced Geothermal System is.

### 1.1 Geothermal energy

The word geothermal is derived from the two Greek words geo (earth) and therme (heat), and subsequently geothermal energy means energy from the earth's inner heat. Since before the beginning of recorded history people have used natural geothermal fluids for cooking and bathing. Geothermal resources has been used to extract heat and power for centuries. In the 14th century the first district heating system started up in France and it is still running [1]. Early on in the 20th century electric power production from geothermal energy began, and geothermal energy was used for industrial purposes. In 1904 geothermal steam produced electricity for the first time in Lardello, Italy.

Today there is extended use of geothermal energy. It is used for direct heating and to produce electricity worldwide. In many cases the geothermal energy is used in a power plant to produce electricity and then used for direct heating. In 2005, when the World Geothermal Congress (WGC2005) were held in Turkey, 68 countries submitted papers about their use of geothermal resources. There were 72 countries that reported use of geothermal energy for either direct use (for heating), for electricity production, or both, between WGC2000 and WGC2005 [2], [3], [4]. The combined figures from the papers submitted to WGC2005 are reported in table

1.1 below. The table is reproduced from [1]. The World Geothermal Congress is arranged every fifth year, so the next will be held next year, in 2010.

Use	Installed power (MW)	Annual energy Use (MW yr)	Capacity factor	Countries reporting
Electric power	8,933	6,482	0.73	24
Direct use	28,268	8,669	0.31	72

Table 1.1: The worlds total geothermal use in 2005, table from [1].

27 countries have produced electric power from geothermal energy using power plants. Greece, Taiwan and Argentina have closed down their power plants driven by geothermal energy for environmental and economic reasons [1].

### 1.1.1 Environmental advantages

Geothermal energy is one of the renewable and green energy resources. There are only minor emissions when producing geothermal energy, substantially less than the emissions from fossil fuels. Geothermal energy extraction causes some noise pollution (mostly from fans). These are at such a low level that they not are regarded as a problem. In addition there is some usage of water and land, but this is at a smaller scale than when fossil fuels are used. The environment damages are much less when using geothermal energy instead of oil, gas and coal as the energy source. In addition there are small impacts on natural phenomena, wildlife and vegetation [5]. In table 1.2 there is a comparison of the emissions from a geothermal plant and a coal plant. Looking at the table we see that the emissions from a geothermal plant are very low. The only emission that is higher from a geothermal plant than from a coal plant is hydrogen sulphide. It is routine at geothermal power plants to treat hydrogen sulfide and convert it to elemental sulfur.

	CO <sub>2</sub> (kg/MW h)	SO <sub>2</sub> (kg/MW h)	NO <sub>x</sub> (kg/MW h)	H <sub>2</sub> S (kg/MW h)	Particulate matter(kg/MW h)
Coal plant	994	4.71	1.95	0	1.01
Geothermal plant	40	0.16	0	0.08	0

Table 1.2: Emission of carbon dioxide, sulfur dioxide, nitrogen oxides, hydrogen sulfide and particulate matter from a coal plant compared to a geothermal plant in kilograms per MW h produced. Data collected from [1].

## 1.2 Heat sources

Geothermal energy is unique, because of the source and transport mechanisms behind it. The average heat flow through the earth crust is  $59 \text{ mW/m}^2$ . The two primary processes which causes heat flow through the crust of the Earth are [7]:

- Heat from the Earths mantle and core, through upward convection and conduction.
- Radioactive elements in the crust causing heat, particularly isotopes of uranium, thorium and potassium.

Regional geologic and tectonic phenomena can cause higher heat flow locally. For example tectonic plate boundaries and volcanic events are associated with higher heat flow. In Iceland, the heat flow is much higher than the average heat flow. The reason behind the high heat flow is that Iceland lies on a tectonic plate boundary. A consequence of lying on such a boundary is often recent volcanic events. Such events create an ideal environment for geothermal energy extraction. This may be the background for geothermal energy extraction to be associated only with areas where the heat flow is high because of such conditions. However, recently Enhanced Geothermal Systems (EGS) have made it possible to extract geothermal energy from areas that were unsuitable for Hydrothermal Systems (HS).

## 1.3 Hydrothermal systems

A hydrothermal system (HS) is a system where there is a natural fractured reservoir ready to be used to extract geothermal energy from. A HS consists of a drill hole into the groundwater and a pump, which is pumping up the hot water before injecting it into the ground again. When targeting a new geothermal reservoir, the ideal reservoir consists of rock that is hot, tectonically stressed and fractured with groundwater in the fractures. Most areas where a HS is economical sustainable, are today used.

### 1.3.1 Enhanced geothermal systems

After some time the fractures in the ground seal, due to secondary mineralization processes. This results in low permeability and little or no fluids present. When this is the case other methods such as hydraulic, thermal and chemical processes must be used to extract energy. With such processes an enhanced geothermal system (EGS) is created. The methods stimulate the reservoir. This causes the

fractures to open, extend and interconnect and results in the creation of a conductive fracture network [6].

An EGS can extend the margins of an existing hydrothermal system (HS) or create new ones where it is not possible to create a HS for different reasons, like low heat flow, low permeability, below ground water levels, etc. The U.S. Department of Energy has earlier defined an EGS “as engineered reservoirs that have been created to extract economical amounts of heat flow from low permeability and/or porosity geothermal resources” [7].

In the assessment done in the MIT report *The Future of Geothermal Energy* [7], where they evaluate geothermal energy as a major supplier of energy in the United States, they have chosen to adapt this definition to include “all geothermal resources that are currently not in commercial production and require stimulation or enhancement”. This definition excludes high-grade hydrothermal reservoirs, but it includes the conduction dominated, low permeability resources in sedimentary and basement formations.

Deep geothermal systems are a kind of EGS. To create a deep geothermal system, drilling far below the groundwater is needed. Because the amount of inherent ground water is limited, water has to be injected to create sufficient heat flow. Therefore there will be a start up time (injection time) where there is no production. In figure 1.1 we see an EGS with an injection and a production well. Here follows the different steps in an EGS [6]:

- The first step in an EGS is to drill an injection well into hot basement rock that has limited permeability and/or fluid contents. This drilling continues considerably below water levels.
- The second step is to inject water at sufficient pressure to ensure that the reservoir is fractured or to open existing fractures, similar to what is commonly done in the oil industry.
- The continued pumping of water ensures extended fractures and reopening of old fractures at some distance from the injection well.
- The next step is to drill a production well with an intent to intersect the fracture system. This creates a circulation of water in the basement rock due to improved porosity. To extract heat from large volumes it may be suitable to drill additional production wells. This is very much an economical question.

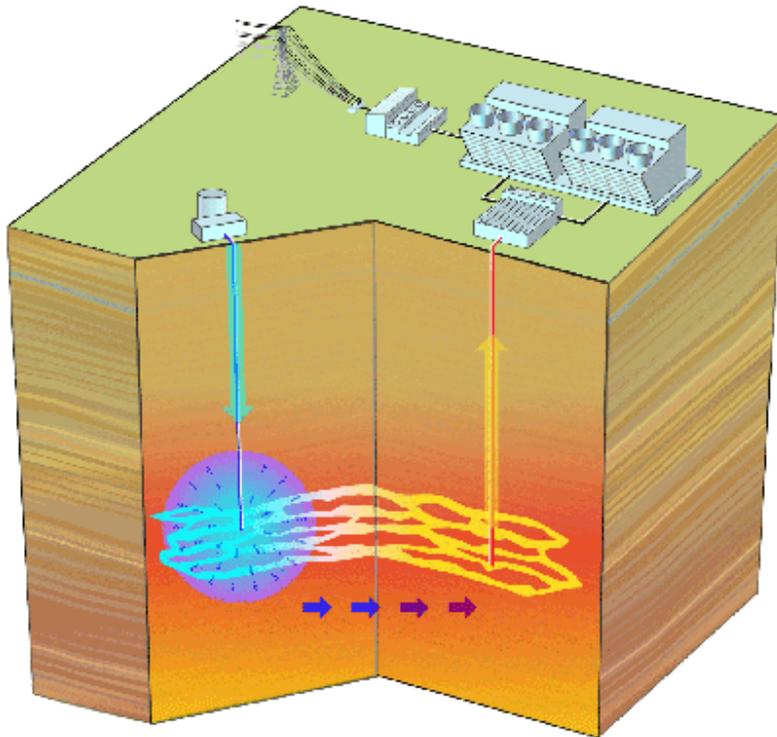


Figure 1.1: Here we see an EGS with an injection and a production well. The water flows from the injection well to the production well. The picture is taken from [8].

### Benefits With EGS

There are many benefits with an EGS, here follows some of them, described in the article [6]. In an EGS there will be more fractures and higher heat flow than in a HS, and therefore the productivity will increase. EGSes can be established in areas where it is too low heat flow and/or too few fractures to establish HSES. The size of the geothermal system becomes more flexible. There are economic advantages, as more energy can be produced. Because of the economic advantages a system can have an extended lifetime. With extended lifetime there follows environmental advantages. This is because there can be extracted more energy from the same site/area without using energy to establish new sites. If energy from EGS is used instead of energy from fossil fuels, the environmental advantages becomes huge.

### EGS reservoir simulation

To understand hydraulic, thermal, mechanical, and chemical processes in geothermal reservoirs, good simulations are needed. An EGS is characterized by low

permeable rocks and a system of hydraulically stimulated fractures. While the energy transfer is dominated by the fluid flow in the fractures, the long-term heat extraction rate is determined by the small scale energy transfer in the blocks. A challenge in EGS reservoir simulation is therefore to capture both the large scale fluid flow in the fractures, and to capture the small scale energy transfer in the blocks with rock. In the following chapters we will create a model that capture both the large and small scale energy transfer.

# Chapter 2

## Reservoir mechanics

In this chapter, we will look into the theory of reservoir mechanics relevant to this master thesis. The focus will be on single phase flow as we only consider single phase flow later in the thesis. The main concepts used for fluid flow in porous media, such as the terms porosity, permeability, homogeneity and isotropy will be explained. We will also describe some of the equations used for fluid flow, Darcy's law and the general conservation law. The term representative elementary volume will be explained. At the end of the chapter we will look into the relation between temperature and enthalpy. The presentation is mainly based on the books of Pettersen [9] and Bear [10].

### 2.1 Porous media

A porous media is a media of firm substance with channels of void space where fluids can flow. Almost every substance in nature is a porous media. It is usually difficult to describe the geometry of the internal solid surfaces that constrict the flow inside a porous medium in an exact way. In the same way it is impossible to precisely describe the fluid that is flowing. Therefore a statistical approach is more suitable, where we can determine the average values of successive measurements. The void space is called the pores of the porous media. In a flow perspective the pores are the most important part of the media. This is where there can flow fluids, such as water, oil and gas. To measure how much pores and firm substance there are in a porous media, the term porosity is needed.

### 2.1.1 Porosity

The volume of the firm substance  $V_{FS}$  and of the pores  $V_P$  together composes the total volume, such that

$$V_B = V_{FS} + V_P , \quad (2.1)$$

where  $V_B$  stands for the bulk (total) volume. The porosity of the media is

$$\phi = \frac{V_P}{V_B} \quad (2.2)$$

and will always be between 0 and 1. The higher porosity a porous media has, the more pores that liquid can flow through. Some of the pores can be isolated, and there will be no flow from these pores to the rest of the pore network. Other pores are dead ends, where the flow is very restricted. In figure 2.1 we see connected, isolated and dead end pores. As a consequence of isolated and dead end pores the term effective porosity is defined as

$$\phi_E = \frac{V_P - V_{DE} - V_I}{V_B} , \quad (2.3)$$

where  $\phi_E$  is the effective porosity,  $V_{DE}$  is the volume of the dead end pores and  $V_I$  is the volume of the isolated pores. In reservoir mechanics it is mainly  $\phi_E$  which is interesting, as it tells how much of the pores the fluid can flow through. Although  $\phi$  should be used in some contexts, for example when determining the specific heat capacity, we will only use  $\phi_E$ . We have chosen to neglect the difference between  $\phi$  and  $\phi_E$ , as this is not the priority in the thesis.

The pore diameter and especially the pore throat diameter are important parameters when modeling how much liquid that can flow through a pore. We see the pore diameter and a pore throat in the figure 2.1. The pore throat is where the pore is at its thinnest. The pore throat diameter will together with other parameters decide how much fluid that can flow trough the pore.

### 2.1.2 Representative elementary volume

When setting up mathematical models for a continuous dynamic system, it is usual to study the variables in a representative elementary volume, from now on referred to as a REV. A REV is a volume that is large enough to have the properties of a porous media. An infinitesimal volume,  $dV$  is the smallest possible volume that can be defined. When  $V_{ref}$  is a typical volume for a whole model, and we want to create a differential equation or a system of differential equations describing the properties of  $V_{ref}$ , it should be divided into  $dVs$ . If we use  $dVs$  to describe the properties of a porous media, it will create difficulties if we use a continuum

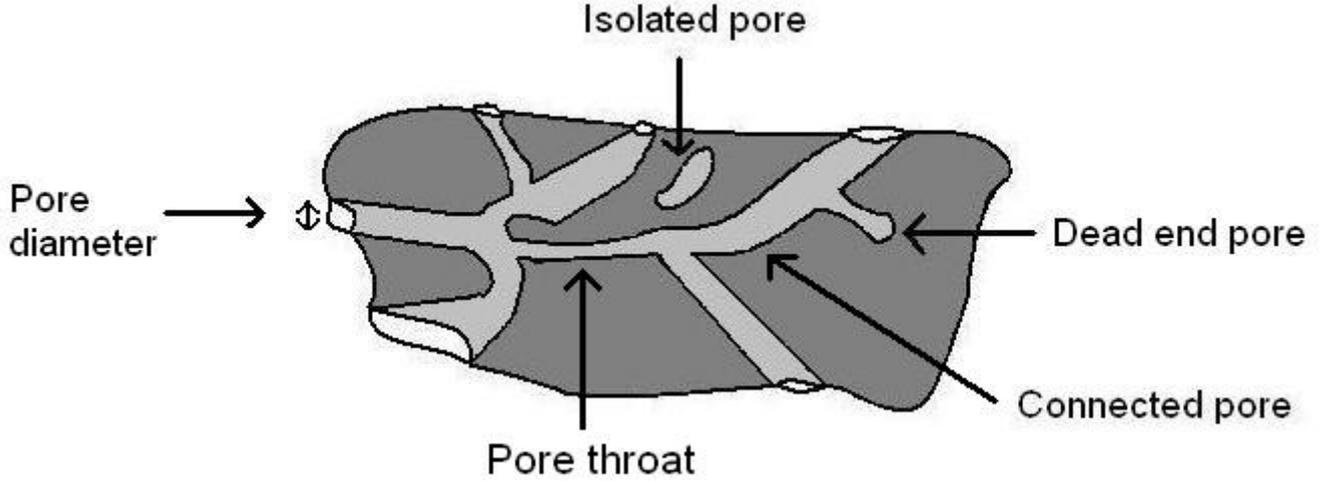


Figure 2.1: A porous media with a pore throat and connected, isolated and dead end pores.

approach to porous media. The volume is too small to have a well defined middle value for the porosity. This has to do with the pore throat diameter. If the pore diameter is larger than  $dV$ , the porosity in  $dV$  will be either 1 or 0, dependent on  $dV$  being inside a pore or outside a pore.  $REV_{crit}$  is the critical value for the size of the volume, the limit for the volume to have porous media properties.  $REV_{crit}$  is the smallest possible volume, where a middle value for the porosity can be defined. It is therefore usual to use a REV that fulfills the inequality

$$dV \ll REV_{crit} < REV \ll V_{ref} ,$$

in models describing porous media.

### 2.1.3 Darcy's law

The French engineer Henry Darcy's researched flow through sand filters, in connection with the dams in Dijon, France. All of his experiments were with vertical flow. Darcy concluded that the rate of flow ( $u$ ) is proportional to the constant cross sectional area ( $A$ ), to the difference in height of the water ( $\Delta h$ ), and inversely proportional to the length ( $L$ ). We then end up with the famous Darcy formula:

$$u = \alpha A \frac{(\Delta h)}{L} , \quad (2.4)$$

The constant  $\alpha$  varies with the type of sand, and is negative. The two heights ( $h_1$  and  $h_2$ ) are measured with a piezometer. A piezometer measures the piezometric

head  $h$ , which describes the sum of pressure and potential energies of the fluid per unit weight. Therefore  $h_1 - h_2$  is the difference in piezometric head across the sand filter of length  $L$ , and  $\left(\frac{h_1 - h_2}{L}\right)$  is the hydraulic gradient. The "Darcy velocity"  $u$  is not the actual velocity the water have in the pores, but can be seen as a kind of flux and is therefore also called the Darcy flux. This flux is the volume that goes through an area per time, and therefore the entity is m/s. The entity is the reason why the Darcy flux often is referred to as a velocity. The Darcy flux is related to the porosity of the porous medium, by

$$u = \frac{v}{\phi}, \quad (2.5)$$

where  $u$  is the Darcy flux,  $v$  is the average velocity in the pores and  $\phi$  is the porosity. With the constant  $\alpha$  being negative, the flow direction is from the position with high potential energy to the position with low potential energy. Darcy did not try other fluids than water in his experiment, and therefore he did not find any variations with different fluids.

### The differential form of Darcy's law

After Darcy's experiments fluid flow through a porous media has been subject of many experiments with different fluids and different flow directions. For flow in all directions, there has been established a differential form of Darcy's law,

$$\underline{u} = -\frac{\mathbf{K}}{\mu} \left[ \frac{\partial p}{\partial x_1} + \theta_1 \rho g, \frac{\partial p}{\partial x_2} + \theta_2 \rho g, \frac{\partial p}{\partial x_3} + \theta_3 \rho g \right], \quad (2.6)$$

where  $\mathbf{K}$  is the permeability,  $\mu$  is the viscosity,  $p$  is the pressure, and  $\rho$  is the density of the fluid. The gravity acceleration ( $g$ ) works in the horizontal direction, and  $\theta_1, \theta_2, \theta_3$  is the angle between the  $x, y, z$ -directions in the coordinate system and the horizontal direction. An important part of the Darcy flux, is the permeability.

#### 2.1.4 Permeability

The permeability is a portion of the conductivity, which is a measure of how well a fluid flow through a porous media. While the conductivity is dependent on both the fluid that flows and the rock it flows through, the permeability is only dependent on the rock. Permeability is measured in Darcy, which is approximately  $10^{-12} \text{ m}^2$ . The permeability can be seen as inverse proportional with the resistance from the rock for the liquid flowing through it. In general the permeability is expressed as

$$\mathbf{K} = \begin{bmatrix} K_{11} & K_{12} & K_{13} \\ K_{21} & K_{22} & K_{23} \\ K_{31} & K_{32} & K_{33} \end{bmatrix},$$

### Homogeneity

If the permeability is constant in a media, it is homogenous. If  $\mathbf{K} = \mathbf{K}(\underline{x})$ , where the permeability varies with its position in the media, we say that it is inhomogeneous or heterogeneous. The permeability can be different in different directions, but still be homogenous. If the permeability is constant in each direction in some coordinate system, it is homogenous. If it is independent of  $x_1$ ,  $x_2$  and  $x_3$  in some coordinate system, and we have  $\mathbf{K} = (K_{x_1}, K_{x_2}, K_{x_3})$  in that specific coordinate system, it is homogenous.

### Isotropy

If the permeability is equal in all directions, where  $K_{x_1} = K_{x_2} = K_{x_3}$  in one specific coordinate system, then it is isotropic. If it is different in all directions, independent of coordinate system, then it is anisotropic. For a system to have positive entropy, the tensor  $\mathbf{K}$  must be positive definite. As the entropy has to be positive, we can assume that  $\mathbf{K}$  is positive definite. The tensor  $\mathbf{K}$  is symmetric, and there exists a orthogonal transformation, such that  $\mathbf{K}$  can be transformed into the diagonal tensor

$$\mathbf{K}^* = \begin{bmatrix} K_{11}^* & 0 & 0 \\ 0 & K_{22}^* & 0 \\ 0 & 0 & K_{33}^* \end{bmatrix} .$$

With other words, the permeability can be represented by flow directions.

## 2.2 Conservation laws

A law that describes a conservation of a particular measurable property in a isolated physical system is called a conservation law. A conservation law is central in almost every model of a dynamic system. There are conservation laws for conservation of mass, energy, linear momentum, electric charge and many other measurable properties. The general conservation law is:

$$\frac{d}{dt} \int_{\Omega} \Gamma d\underline{x} + \int_{\partial\Omega} \underline{n} \cdot J d\underline{x} = \int_{\Omega} q d\underline{x} , \quad (2.7)$$

where  $\Gamma$  is the conserved quantity within the volume  $\Omega$ , the flux of  $\Gamma$  through the boundaries of  $\Omega$  is  $\underline{n} \cdot J$  and  $q$  is either a source or a sink inside  $\Omega$ . If a conservation law holds for each REV, it is also possible to create a conservation law that holds for a arbitrary sized volume that is greater than a REV. Later in this thesis we will use conservation laws for enthalpy to create a mathematical model for geothermal heat extraction.

## 2.3 Temperature and enthalpy

To measure heat flow in a reservoir, one natural choice would be to use temperature with Kelvin as the entity. It is also possible to look at the internal energy in the reservoir, or more precisely the change in the internal energy over some time. Enthalpy is a thermodynamic potential, that describe the state of a thermodynamic system [21].

When the pressure is constant enthalpy is defined as

$$H = U + pV ,$$

where  $H$  is the enthalpy,  $U$  is the internal energy,  $p$  is the pressure of the system and  $V$  is the volume. We have the following relation between enthalpy and temperature

$$dH = d(U + pV) = TdS + Vdp ,$$

where  $dS$  is the change of entropy in the system. With constant pressure we have

$$C_p = \left( \frac{\partial H}{\partial T} \right)_p , \quad (2.8)$$

where  $C_p$  is a constant. As we model the pressure in other parts of the equations than the enthalpy, the pressure will be constant in the enthalpy terms. Due to water flow there is a energy flux in geothermal reservoirs. The reason that we do use enthalpy instead of internal energy is that while the inner energy represents the energy that is present, enthalpy represents the part of the energy flux in the reservoir that is a consequence of water flow. We model energy fluxes later in the thesis, and therefore we have chosen to use enthalpy as the conserved quantity.

As the enthalpy is a energy quantity, it has joules as its entity. When the temperature of the water is below its boiling point, the enthalpy and the temperature will be proportional entities. (If the pressure is constant.) When the waters temperature is near the boiling point, this is not true. In the plot in figure 2.2, the temperature is plotted versus enthalpy. Before the liquid is boiling, the temperature is proportional to the enthalpy. When the liquid starts to boil, the temperature will not rise, but the enthalpy will. The rise in the enthalpy is a consequence of the pressure of the water increasing when its boiling. In this thesis we are only looking into the first part of the curve, where the water is not boiling. In our model we have

$$T \propto h, \quad T < \text{boiling point} ,$$

and the temperature and the enthalpy will be proportional. Note that in a reservoir far below the ground, the pressure will ensure that the water has to be much

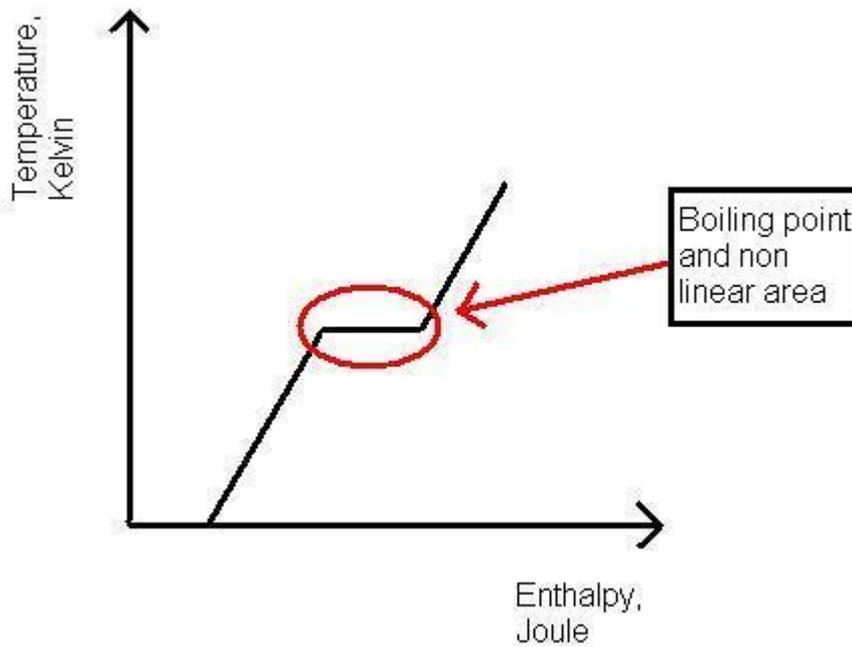


Figure 2.2: The temperature plotted versus enthalpy, with one non linear area.

warmer than at ground level to boil. Water and rock have different entropy. As we in this thesis mainly focus on the discretization aspects of the models we use, we have chosen to neglect these differences. Therefore we use the same proportionality constant between temperature and enthalpy in both water and rock. A consequence of this is that joules no longer are the entity of enthalpy. We will still use the term enthalpy for this quantity, despite the fact that the proportionality constants are inaccurate.



# Chapter 3

## Mathematical model

In this chapter, we exploit the theory from the reservoir chapter to make a model for geothermal energy extraction. The model will be based on the conservation law for enthalpy. To define a local (fine-scale) conservation law, we look into the composition of a reservoir. There are two scenarios that we will like to use our model for. The first scenario is that the reservoir consists of blocks with dry rock and fractures between the blocks, where the fractures are filled with flowing water. The other scenario is that the blocks consist of porous media, with small fractures, but that there is no flow of water inside the blocks. As in the first scenario, the water flows in the fractures between the blocks. The model is inspired by the articles [18] and [16].

In both scenarios the global flow of water will only take place in the fractures, but there is an enthalpy flux in both the blocks and in the fractures between the blocks. Parameters like the thermal conductivity, specific heat conduction and density inside each block will be averages. In the second scenario, the parameters in the blocks are averages where both the firm substance's and the water's parameters are part of the average. We split the blocks into layers. There will be a enthalpy transfer from one layer to the two neighboring layers. In the utmost layer there will be an enthalpy transfer to the water phase in the fractures between the blocks.

The local conservation law will be valid inside each block, and the flux term in the conservation law will transfer enthalpy between the layers. In this chapter we explain why we divide the blocks into layers and how the local conservation law models an enthalpy flux between the layers. We then see how a reservoir scale conservation law for enthalpy leads to three different flux expressions, one for the diffusive flux, one for the advective flux and one for the heat exchange flux. The heat exchange flux is based on the local conservation law. We will go through how

the expressions for these fluxes are, and how they transfer enthalpy. We will also discuss the initial and boundary conditions that will be used. The model created in this chapter will make the foundations for the numerical experiments in the next chapter, where we see the effect of different numerical choices.

## 3.1 Fine-scale model

As mentioned, the model consists of blocks and water filled fractures between the blocks. In the fine-scale model we will model one block and the fractures near it. Enthalpy is exchanged between the block layers and between the water and the utmost block layer. The blocks in our model are incompressible, homogenous and isotropic.

### 3.1.1 Blocks and fractures

The blocks can be of different sizes and structures. If a block is small, all of it will be near a fracture, and the model will reach equilibrium fast through a large enthalpy transfer. In a larger block the enthalpy will have to go through rock for some distance before it reaches the water phase. To model this we divide each block into different layers, as illustrated in figure 3.1. Each layer interacts with the two closest layers, there is an enthalpy transfer between them, and each layer is defined as the domain  $\Omega_i$ .

The first block layer will interact with the water phase and the block layer inside itself. The last layer will only interact with the layer outside itself. This idea is inspired by the articles [19] and [20]. The longer the distance between a block layer and the water are, the further the enthalpy has to be transferred to reach the water phase. This will give a more realistic model than if we only had one layer in each block. If we only use one layer, all of the block would interact with the water independently of how far it is away from it. We see the reservoir as filled with hexagon shaped blocks, like the block in figure 3.1. In our model we choose to have all the blocks shaped as hexagons. This is mainly done for simplicity, but it also has some root in nature. There are some fracture systems that create hexagons in nature, for example in the Death Valley in California, USA, as seen in figure 3.2.

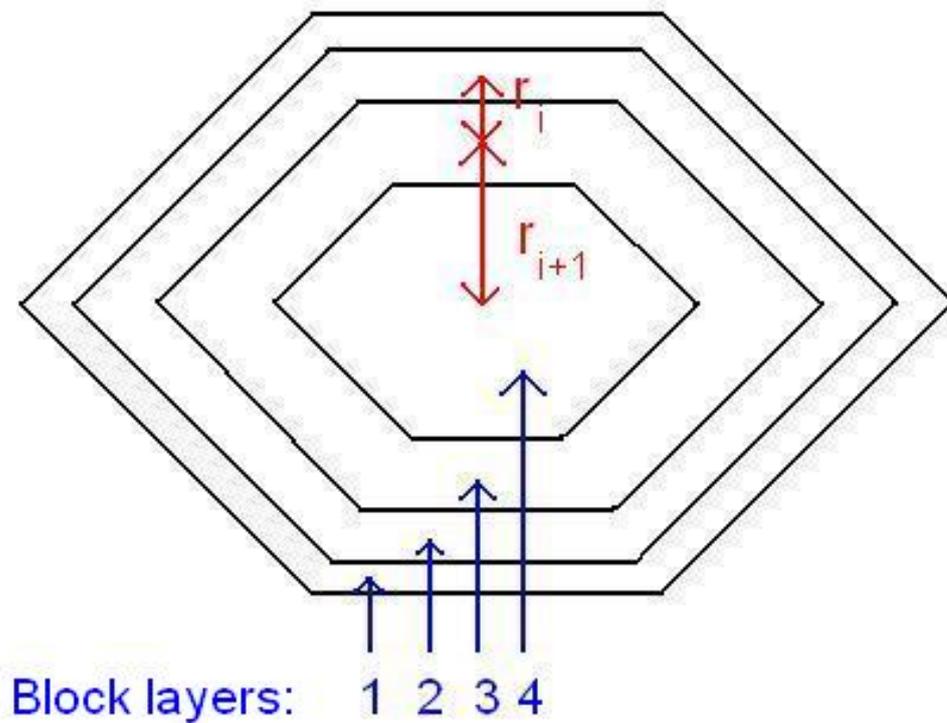


Figure 3.1: A hexagon shaped block, its block layers and the distances between the layers,  $r_i$  and  $r_{i+1}$ .



Figure 3.2: The picture is taken from [17], and shows how some fracture systems in nature are hexagon shaped. The picture is from Death Valley, USA.

### 3.1.2 Local conservation law

Recall from section 2.3 in the reservoir mechanics chapter that we use the same proportionality constant in both water and in the blocks, when enthalpy as a concept is defined in our thesis. To model the enthalpy transfer in the fine-scale model, we need to establish a local conservation law. Such a law for the enthalpy in a block and the fractures near it is defined as:

$$\frac{1}{V} \left[ \frac{d}{dt} \int_{\Omega_i} h \, d\mathbf{x} + \int_{\partial\Omega_{i-1}} \underline{n} \cdot \underline{J} dS + \int_{\partial\Omega_i} \underline{n} \cdot \underline{J} dS \right] = q, \quad (3.1)$$

where  $\Omega_i$  is the volume of layer  $i$ ,  $\partial\Omega_i$  is the surface between layer  $i$  and  $i + 1$ ,  $\underline{n}$  is the normal vector at the boundary in the direction of the flux,  $V$  is the volume that we model,  $\underline{J}$  is the flux vector, and  $q$  is the sink/sources inside  $V$ .

To establish a local conservation law in the fine-scale model, we need to find approximations to the integrands in equation (3.1). To approximate the integrand in the surface integrals (the enthalpy flux), an approximation to the enthalpy gradient over the surface is needed. The distance between two block layers,  $r_i$  is the distance between layer  $i$  and  $i + 1$ , defined as in figure 3.1. If  $r_i$  is the same in all layers we will have

$$r_i \propto \frac{1}{k}, \quad (3.2)$$

where  $k$  is the number of layers we have. How thick the layers are and how many layers there are, together decides the distance the enthalpy has to be transferred to reach the water.

We approximate the enthalpy gradient between two layers with

$$\underline{n} \cdot \nabla h|_{\Omega_i} \approx \frac{h_{i+1} - h_i}{r_i}, \quad (3.3)$$

where  $\nabla h$  is the gradient of the enthalpy,  $\Omega_i$  is the surface between layer  $i$  and  $i + 1$ ,  $h_i$  is the enthalpy of layer  $i$ , and  $\underline{n}$  is the normal vector of the enthalpy transportation over the surface  $\Omega_i$ .

The integrand in the surface integral is therefore approximated with

$$\underline{n} \cdot \underline{J}|_{\partial\Omega_i} = \underline{n} \cdot (-\kappa \nabla h)|_{\partial\Omega_i} \approx \kappa_{hm} \frac{h_{i+1} - h_i}{r_i}, \quad (3.4)$$

where  $\kappa_{hm}$  is the weighted harmonic mean of the thermal conductivity in the layers  $i$  and  $i + 1$ . Therefore the last surface integral in equation (3.1) becomes

$$\int_{\partial\Omega_i} \underline{n} \cdot \underline{J} dS \approx -\kappa_{hm} \frac{h_{i+1} - h_i}{r_i} \cdot |\partial\Omega_i|, \quad (3.5)$$

where  $|\partial\Omega_i|$  is the area of the surface  $\partial\Omega$ . The first integral in (3.1) becomes

$$\frac{1}{V} \frac{d}{dt} \int_{\Omega_i} h \, d\underline{x} = \frac{1}{|\Omega_i|} \Omega_i \frac{d\bar{h}_i}{dt} = \frac{d\bar{h}_i}{dt} , \quad (3.6)$$

where  $\bar{h}_i$  is the average of the enthalpy in layer  $i$ ,  $|\Omega_i|$  is the volume of block layer  $\Omega_i$ , the volume that is modeled is  $V = \sum_i |\Omega_i|$ , and  $\phi_i$  is the volume density of layer  $i$ .

Each block layer will have a different thermal conductivity,  $\kappa_i$ , which is defined as the mean of the thermal conductivity in the layer. We are using the harmonic mean of the thermal conductivities  $\kappa_i$  and  $\kappa_{i+1}$  in the equations (3.4) and (3.5). The weighted harmonic mean of the thermal conductivities are defined as

$$\kappa_{hm} = \frac{\frac{r_i}{\kappa_i} + \frac{r_{i+1}}{\kappa_{i+1}}}{\frac{r_i}{\kappa_i} + \frac{r_{i+1}}{\kappa_{i+1}}} . \quad (3.7)$$

In equation (3.5), the area of the surface between block layer  $i$  and  $i+1$  is used. If the surface between two layers,  $|\partial\Omega_i|$ , is large, there will be a large enthalpy interaction between them. In equation (3.6) we use the term  $|\Omega_i|$  for the volume of block layer  $\Omega_i$ .

Later in the thesis we will use the terms volume density or macro porosity, that means that each block layer covers a fraction of the volume of the entire block. This fraction is

$$\varphi_i = \frac{|\Omega_i|}{\sum_{j=0}^{j=N} |\Omega_j|} . \quad (3.8)$$

With  $k$  layers we will have

$$\sum_{i=1}^{i=k} \varphi_i = \varphi_{pm} , \quad (3.9)$$

where  $\varphi_{pm}$  is the volume of the entire block. The water phase has volume density  $\varphi_0$ , and we therefore have

$$\sum_{i=0}^{i=k} \varphi_i = 1 . \quad (3.10)$$

We now have simple expressions for the the integrals in the local conservation law (3.1), that can be used to model the enthalpy transfer locally. The next step is to establish a model for the reservoir scale.

## 3.2 Reservoir scale model

A reservoir scale model, is a global model for the whole reservoir. The reservoir is divided into REVs. We make a model that is valid from the REV size and up.

### 3.2.1 Reservoir scale conservation law

In model a reservoir scale conservation law for enthalpy,

$$\frac{d}{dt} \int_{\Omega} h \, dx + \int_{\partial\Omega} \underline{n} \cdot J \, d\underline{S} = q . \quad (3.11)$$

The difference from the local conservation law is that the enthalpy now is conserved for each REV. In the reservoir scale we model the flux between the different blocks, but there will in addition be a flux between the different layers in the blocks. The reservoir scale flux between the layers, which we call the heat exchange flux, will be modeled much in the same way as in the fine scale model. The flux between the different blocks will consist of two different fluxes, a diffusive flux and a advective flux. The flux  $J$  consist of the three fluxes,

$$J = J_{diff} + J_{adv} + J_{exch} , \quad (3.12)$$

where  $J_{diff}$  is the diffusive flux,  $J_{adv}$  is the advective flux, and  $J_{exch}$  is the heat exchange flux.

The diffusive flux takes place in both the fluid and the blocks. The advective flux takes place only in the water which is flowing in the fractured parts of the reservoir. There will also be a flux locally inside each block and locally in the water phase, that is similar to the diffusive flux between the blocks. Expressions for the three different fluxes are needed to model it.

### 3.2.2 Flux expressions

The expressions for the global enthalpy fluxes are based upon the theory from the reservoir mechanics chapter.

#### The diffusive flux

In both the blocks and in the water there will be a diffusive flux given by heat conduction. From reservoir mechanics theory we know that the heat conduction depends on the thermal conductivity and the gradient of the temperature. Therefore the diffusive flux will be

$$J_{diff} = -\nabla(\kappa T(h)) ,$$

where  $\kappa$  is the thermal conductivity,  $T$  is the temperature and  $h$  is the enthalpy as before. Since we in our model sees enthalpy and temperature as proportional quantities, with the same proportionally constant for all materials, the diffusive flux can also be written

$$J_{diff} = -\nabla(\kappa ph) = -\nabla(\kappa_{mod}h) ,$$

where  $p$  is the proportional constant between enthalpy and temperature. This constant is part of  $\kappa_{mod}$ . Later in this thesis we will for simplicity use  $\kappa$  instead of  $\kappa_{mod}$ .

In the special case where the diffusive flux is the only term that is transferring enthalpy in the reservoir, this leads us to the heat equation,

$$\phi_i \frac{\partial h_i}{\partial t} - \nabla \cdot (\kappa \nabla h_i) = 0 . \quad (3.13)$$

The larger  $\kappa$  is, the larger the enthalpy transfer driven by the diffusive flux will be.

The heat equation is an important Partial Differential Equation (PDE). It has been and is object for intensive research and modeling in many different areas. The heat equation describes the density of some quantity such as heat, chemical concentration, etc, and how this density evolves in time [14]. In a geothermal perspective the heat equation spreads heat out from warm regions to colder regions in the reservoir. If the heat equation is the only equation that is transferring the enthalpy in the model, with no enthalpy added or subtracted, equilibrium will eventually be established. The diffusive term will model the diffusive flux between the different blocks in our model, and similar between the different fractures with water. It will transfer enthalpy from one layer in one block to the corresponding layer in other blocks.

### The advective flux

The water phase is moving through the fractures between the blocks. This creates an advective enthalpy flux, that is of the form

$$J_{adv} = (vh) ,$$

where  $v$  is the flow field. The flow field specify the direction and speed of the water movement in the reservoir. In the special case where there only is an advective flux, this leads to a transport equation

$$\phi_0 \frac{\partial h_0}{\partial t} + \nabla \cdot (vh_0) = 0 . \quad (3.14)$$

The blocks will in general not move, and if they do our reference system moves with it. We have chosen the blocks to be incompressible. Therefore the advection term only contributes in the modeling equation for the water. The higher value of  $v$ , the faster the water will move, and as a consequence also the enthalpy.

A transport equation as (3.14) is fundamentally different from the heat equation. The enthalpy is simply carried along, or advected, with the water flow [15], while the heat equation is driven by molecular diffusion.

The flow field is normally determined by a pressure equation, based on Darcy's law (2.6) and mass conservation. As we do not focus on that here, we have instead used an analytical flow field. As we later will have a production well in the middle of the reservoir, we have chosen a radial flow field, ensuring water flow to the well. Water will be taken out in the production well, the pressure will sink near the well and water will flow to it.

### The heat exchange flux

The heat exchange flux that works globally, transfers enthalpy from one block layer to another. We have called it the heat exchange flux as it is heat that is transferred, but we model it with a enthalpy flux. In the fine-scale model, each block is modeled separately. A REV potentially contains many blocks, and we model all these blocks as one in the reservoir scale model. A block layer in the REV, covers the same layer in all of these blocks.

In the special case where the heat exchange flux is the only flux in the reservoir, our model equation will be of the form

$$\phi_i \frac{\partial h_i}{\partial t} = c_{i-1}(h_{i-1} - h_i) + c_i(h_{i+1} - h_i) , \quad (3.15)$$

where  $i$  is the index for the layer,  $c_i$  is the heat exchange coefficient for the heat exchange between block layer  $i$  and  $i + 1$ ,  $c_0$  is the heat exchange coefficient for the exchange between the water phase and the utmost block layer, and  $h_0$  is the enthalpy of the water phase. Calculating the heat exchange coefficients is done with basis upon the fine-scale model. The local heat exchange flux only has to be modified slightly to express a global heat exchange flux. From the equations (3.5) and (3.15) we get the coefficients

$$c_i = -\frac{1}{V} k \kappa_{hm} |\partial\Omega_i| , \quad (3.16)$$

where  $k$  is the sum of the number of blocks inside  $|\Omega_i|$  and  $|\Omega_{i+1}|$ ,  $|\partial\Omega_i|$  is the surface between layer  $i$  and  $i + 1$  and  $V$  is the volume we model. If there are small sand grains in the reservoir,  $k$  will be larger and the heat exchange will be larger than if there are large blocks.

The heat exchange flux has many similarities with the diffusive flux. When there are many block layers in the model, this term will be of the same form as the diffusive flux term, with a derivative on the right hand side of equation (3.15). The difference from the diffusive flux term is that instead of transferring enthalpy between the different blocks, it will transfer heat inside each block.

### 3.2.3 Combining the flux terms to model equations

To obtain a model for the enthalpy in a reservoir, the flux terms have to be combined to model equations. All the fluxes mentioned work in the water phase. To model the flux of the enthalpy in the water we combine the flux terms into one model equation,

$$\phi_0 \frac{\partial h_0}{\partial t} - \nabla \cdot (\kappa_0 \nabla h_0) + \nabla \cdot (v h_0) = c_0 (h_1 - h_0) - q , \quad (3.17)$$

where  $q$  is the sinks and sources in the reservoir. Note on the right hand side of equation (3.17), that the enthalpy of the water,  $h_0$ , only will interact with the enthalpy of the utmost block layer,  $h_1$ , as mentioned earlier.

As the blocks stays in the same location, there will be no transport term in the combined equations for the block layers. We therefore have the following equation modeling the enthalpy in the block layers,

$$\phi_i \frac{\partial h_i}{\partial t} - \nabla \cdot (\kappa_i \nabla h_i) = c_{i-1} (h_{i-1} - h_i) + c_i (h_{i+1} - h_i) - q , \quad (3.18)$$

where  $i \geq 1$ .

#### Sinks and sources

In our model we only have sinks and sources at the boundaries, a sink in the production wells and sources on the surrounding boundary. The same amount of enthalpy is taken out at the production wells for the whole production time. How we model this sink is described in Chapter 4.

## 3.3 Initial and boundary conditions

When there are water and blocks in a reservoir it can be assumed that the temperature will reach equilibrium after some time. The temperature will be the same in the blocks and in the water. As a consequence the enthalpy (in the way we have defined enthalpy) will also be the same in the entire reservoir at the start of our modeling. Then the enthalpy is

$$h = C ,$$

where  $C$  is a set constant.

Our model reservoir has two boundaries, one surrounding the reservoir, and one in the production wells. In the production wells water, and subsequently enthalpy, is taken out. When water and enthalpy is subtracted in the wells, there will be a transfer of heat and enthalpy from the rest of the reservoir to the wells, due to temperature and pressure differences.

In real life there is no boundary surrounding the reservoir, as the model reservoir will be a part of a much larger reservoir. In fact the whole of the earth's crust is a geothermal reservoir. Therefore there will be transferred enthalpy into the reservoir that is modeled. To model this effect we need boundary conditions. There are several to choose from, and some examples are Dirichlet, Neumann or mixed boundary conditions.

### **Dirichlet, Neumann, Mixed and Robin boundary conditions**

With Dirichlet boundary conditions, the value of enthalpy at the boundary is kept constant. Using this condition on our boundary surrounding the reservoir yields

$$h_{sb} = C , \quad (3.19)$$

where  $C$  is a given constant. No matter how much enthalpy the wells are producing, the enthalpy value on the boundary stays the same. If enthalpy is taken out in the wells, the temperature on the boundary will stay the same no matter how much enthalpy that is taken out.

If the model reservoir is part of a much larger reservoir, this makes physical sense. When this is the case there will be so much heat surrounding the reservoir, that the "boundary" almost would be constant also in real life.

The assumption that the enthalpy at the boundary will stay constant is not entirely true, it will get colder. Anyhow, this is a good assumption to make as it would stay almost constant, when there is a large reservoir surrounding the model reservoir.

With a Neumann boundary condition, the normal derivative of the enthalpy is kept constant at the boundary,

$$\frac{\partial}{\partial n} h_{sb}(x) = C , \quad (3.20)$$

where  $\frac{\partial}{\partial n}$  is the normal derivative and  $C$  is a set constant. As a consequence of this boundary condition, the boundary values are dependent on the rest of the reservoir.

Mixed boundary conditions means using the Dirichlet boundary condition on one boundary and Neumann boundary conditions on another boundary. This gives advantages when the boundaries have different properties.

The Robin boundary condition is seen as a compromise between the Dirichlet and the Neumann boundary conditions. The Robin boundary condition will have the form

$$-\kappa \frac{\partial}{\partial x} h(x) + v_x(x)h(x) = C , \quad (3.21)$$

where  $\kappa$  is the diffusion constant, and  $v$  is the advection velocity. With Robin boundary condition, boundary conditions on both the diffusion term and the advection term have to be implemented.

We have chosen Dirichlet boundary conditions for the boundary surrounding the model reservoir, as they are simple to implement and make physical sense in the setup we have chosen. The surrounding boundary will have the same enthalpy as the reservoir had initially, for as long as the model runs.

The wells also need boundary conditions. There is a sink/source in each well. How we choose to model this is described in the next chapter.



# Chapter 4

## Numerical model

In this chapter, we will use our mathematical model from the previous chapter to create a corresponding numerical model. First we will explain why we have chosen a two dimensional representation, then we will describe the grid that we use in our model. We discretize the flux terms separately, with both explicit and implicit time discretizing for each term. To discretize the diffusion term we use a finite element method (FEM). A finite difference method (FDM) is used to resolve the transport term in our model equations. Therefore the mentioned methods will be described briefly in this chapter. A description of how we implement the boundary conditions we established in Chapter 3 will also be given.

We have created one solver that solves all equation terms with implicit time discretization, and three that solve one term with implicit discretization. First we will describe how the model equations in the different solvers will be.

A term in a solver can either have implicit or explicit time discretization. That is, the term can be calculated either at the start or at the end of a time step. All the terms that are solved at the end of the time step, must be solved in the same system. We use a matrix system to solve the implicit terms. The terms that are solved explicitly are solved at the start of the time step, using the previous values when they are calculated. Therefore they can be solved one at a time, in less complex systems. When only one term is transferring enthalpy in the model, the equations turn into either a heat equation, a heat transport equation or a heat exchange equation.

There is a CFL condition for each term that is solved explicitly. This condition gives a upper limit for the time step used in the explicit solvers, and we will state the conditions for the different solvers we have developed. Other numerical choices we have made, such as using no diffusion term in the inner block layers and using

a divergence free velocity field, will also be mentioned.

## 4.1 Two dimensional representation

We are modeling a three dimensional (3D) reservoir, but for simplicity we have chosen to model the reservoir using a two dimensional (2D) representation. We assume that the fractures are vertical, and therefore it makes sense to see the reservoir from above. In our model we use the averages of the variables in the vertical direction. The results from our 2D representation can be extended to a 3D representation after it is calculated. For a reservoir that is homogenous and isotropic, a 2D representation that we later extend to a 3D representation will give the same results as if we choose to have a 3D representation to begin with. If the reservoir is inhomogeneous and anisotropic, using such a 2D representation will give an approximate solution, but it can still be a good estimation. Using a 2D representation also give us many advantages numerically. There will be less grid points, thus reducing the size of our numerical problem, and the computations will go much faster.

## 4.2 Grid

In our model we use a uniform triangular grid, with no local refinements. When our grid has dimension  $N$ , we have  $N$  internal grid points plus two on the boundary in the x-direction and the same in the y-direction, as in figure 4.1. Therefore there will be  $(N + 2)^2$  grid points. If we had used a 3D representation there would have been  $(N + 2)^3$  grid points, and the computational cost would have been much higher. When  $N = 97$  is used in a grid, the grid will have almost 10,000 grid points.

Using a triangular grid makes it easier to use a finite element method for spatial discretization. We will use a finite element method for spatial discretization in the diffusion term in the model equations. A triangular grid will also make it easier to develop a irregular grid, than with square grid cells. An irregular grid will in most cases give a better accuracy than a regular grid, as there can be more grid points in certain areas where it is needed, but the back draw is that it is harder to implement.

## 4.3 Numerical implementation of the block layers

As we have seen in Chapter 3, we have divided the blocks of rock into block layers, and chosen the blocks to be shaped like hexagonal cylinders. We can model all the

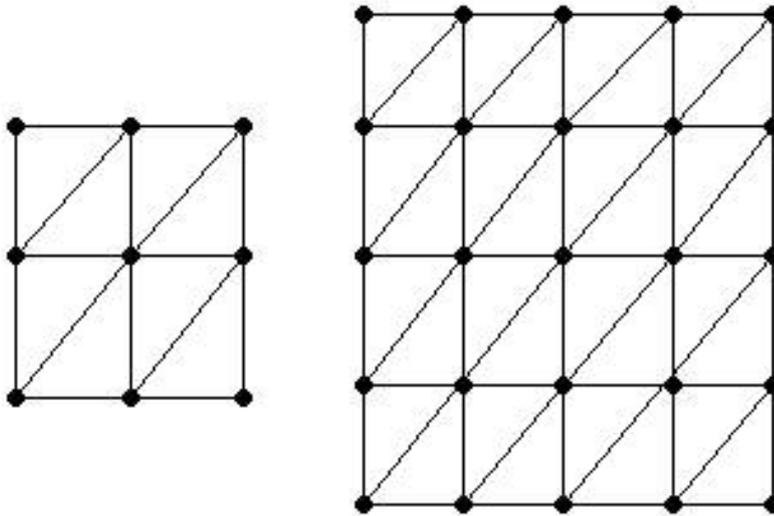


Figure 4.1: A triangular grid with dimension  $N=1$  and  $N=3$ .

blocks inside each REV with one grid point, but there can also be more than one grid point inside each REV. We will establish a numerical model that can model a whole REV with only one grid point.

In figure 4.2 we see a REV with four blocks from above. The rest of the REV will be calculated from the values in the grid points. We need a grid point in each layer, and the grid points in each layer will have the same  $x$  and  $y$  location in our model.

## 4.4 Finite element methods

To discretize the diffusion term in our model equations in space, we use a finite element method (FEM). A FEM approximates PDEs, using a linear approximation. This description of FEM's is based upon [11] and [12].

### Boundary value problem in the variational formulation

A standard problem to be solved by a FEM is a boundary value problem. Such a problem has a given value on the boundary  $\partial\Omega$ , and then solves a function defined on the rest of the domain  $\Omega$ . A FEM discretizes in space using nodes or grid points where the solution of the equation is calculated, and then the solution on the triangles or squares between the nodes are calculated from these node values.

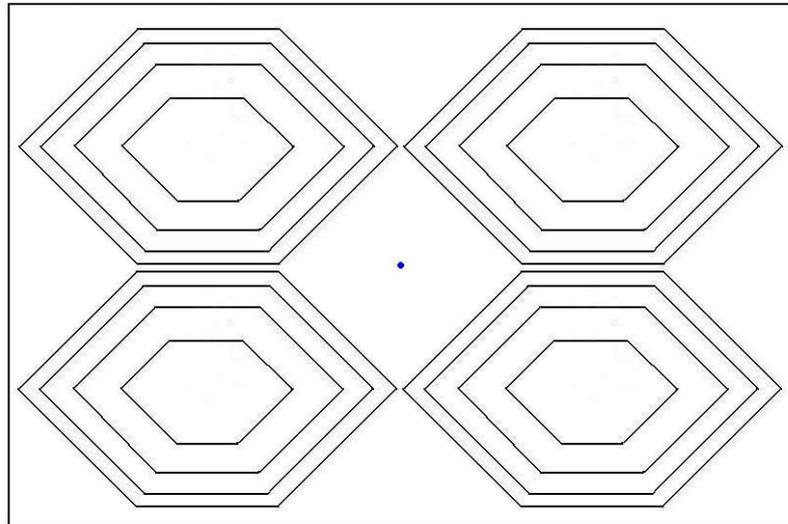


Figure 4.2: A REV with blocks of rock seen from above, with a grid point in the middle.

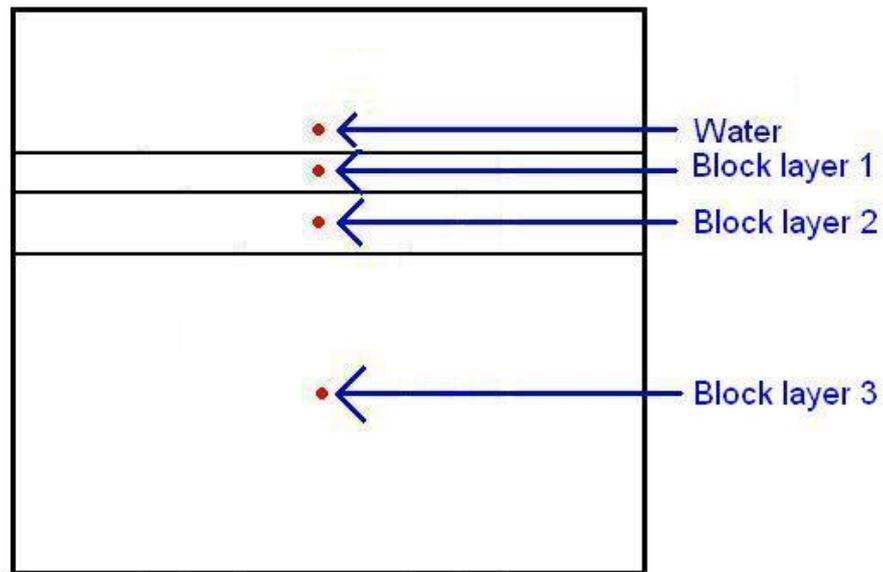


Figure 4.3: A block seen from the side, with grid points in the different block layers and in the water.

For a one dimensional problem, it will be phrased like this: Let  $V$  be the space of all functions  $v$  on  $[a, b]$  that have piecewise continuous first derivative. We have the inner product

$$(u, v) = \int_a^b u(x)v(x)dx$$

and norm

$$\|u\|_V = (u, u)^{\frac{1}{2}} = \left( \int_a^b u(x)^2 dx \right)^{\frac{1}{2}}.$$

The variational formulation of the problem is then posed. Find

$$u \in V$$

such that

$$a(u, v) = L(v) \quad \forall v \in V,$$

where  $L(v)$  is linear to  $v$  and  $a(u, v)$  is linear both to  $u$  and  $v$

### A general problem solved by a FEM

The typical FEM problem is solved in this sequence:

1. Find  $u \in V$  such that  $a(u, v) = L(v) \forall v \in V$ .
2. Pick  $V_h$  such that it is a finite dimensional subspace of  $V$ .
3. Define a discrete problem, that is find  $u_h \in V_h$  such that  $a(u_h, v) = L(v) \forall v$  in  $V_h$ .

When defining a discrete problem, a grid has to be defined. The  $h$ -index indicates that the problem is discretized with length  $h$  between each grid point, such that  $u_h$  is discrete function with distance  $h$  between each value. In this thesis we are working on a regular triangular grid, as mentioned earlier in this chapter. The number of nodes inside each grid cell has to be defined. We have chosen to have three nodes at each triangle, one in each corner. A FEM is built around basis functions, and we need to define one basis function for each node. The basis functions,  $\phi_{i=1}^3$ , define a basis for  $V_h$ .

### Basis functions

Instead of creating separate basis functions for every triangle it is possible to transform it to a elementary triangle, with nodes in  $(0,0)$ ,  $(1,0)$  and  $(0,1)$ . This simplifies the calculations of the contribution from every triangle. The basis functions in this elementary triangle are

$$\begin{aligned}\phi_1 &= 1 - x - y \\ \phi_2 &= x \\ \phi_3 &= y .\end{aligned}$$

The value of the basis functions will be  $\phi_i = 1$ , when we are in the  $i$ th corner, and  $\phi_i = 0$  when we are in one of the other corners. We then transform it back to the original triangle in the grid. We use these to define  $a(u, v)$  and  $L(v)$ . Let  $u_h = \sum_{i=1}^N \zeta_i \phi_i$  and  $v_h = \sum_{j=1}^N \eta_j \phi_j$ , where  $\zeta_i$  and  $\eta_j$  are weight functions. We get

$$a(u_h, v_h) = a(\sum_i \zeta_i \phi_i, \sum_j \eta_j \phi_j) = \sum_i \sum_j \zeta_i a(\phi_j, \phi_i) \eta_j = \underline{x}^T \mathbf{K} \underline{y} ,$$

where  $\sum_i \sum_j a(\phi_j, \phi_i) = \mathbf{K}$ ,  $\underline{u}_h = \underline{x}$ , and  $\underline{v}_h = \underline{y}$ .  $\mathbf{K}$  will be a  $n \cdot n$  matrix, while the vectors  $\underline{x}$  and  $\underline{y}$  will have  $n$  elements. We also get

$$L(v_h) = L(\sum_j \eta_j \phi_j) = \sum_j \eta_j L(\phi_j) = \underline{b}^T \underline{y} ,$$

where  $L(v_h) = \underline{b}^T \underline{y}$ . The vectors  $\underline{y}$  and  $\underline{b}^T$  will also have  $n$  elements. Then we will have

$$a(u, v) = L(v) \Leftrightarrow \underline{x}^T \mathbf{K} \underline{y} = \underline{b}^T \underline{y} \Leftrightarrow \mathbf{K} \underline{x} = \underline{b} ,$$

and we have the possibility to solve the discrete problem as a linear system. The linear system is solved by a numerical equation solver.

#### 4.4.1 Variational form for the heat equation

We start out with the heat equation,

$$\begin{aligned}\nabla \cdot (k \nabla u) &= C, & u &\in \Omega \\ u &= C, & u &\in \partial \Omega ,\end{aligned}$$

with  $k$  constant. We then use a test function  $v \in C_0$ , which gives us

$$\begin{aligned}\int_{\Omega} [\nabla \cdot (k \nabla u) - C] v dx &= \\ \int_{\Omega} \nabla \cdot (k \nabla (u)) v dx - \int_{\Omega} C v dx &\end{aligned}$$

⇓ Green's

$$\int_{\Omega} k \nabla u \nabla v dx - \int_{\partial\Omega} \frac{\partial u}{\partial n} v dS - \int_{\Omega} C v dx = \int_{\Omega} k \nabla u \nabla v dx - C \int_{\Omega} v dx \quad ,$$

as  $v$  is zero on the boundary  $\partial\Omega$ .

We choose

$$a(u, v) = \int_{\Omega} k \nabla u \nabla v dx \quad (4.1)$$

$$L(v) = C \int_{\Omega} v dx \quad , \quad (4.2)$$

which gives us a discrete problem that leads to a linear system, which can be solved numerically.

## 4.5 Finite difference methods

The integral finite difference methods (FDM), are numerical methods for approximating solutions to differential equations (DE). It is used both to approximate solutions in ordinary differential equations (ODEs) and partial differential equations (PDEs). A FDM uses finite difference equations, to approximate the derivatives. A finite difference equation is of the form

$$\Delta f(x) = f(x + b) - f(x - b) \quad .$$

Finite difference equations are divided into forward, backward and central differences. In our thesis only forward and backward difference equations is used. A forward difference is of the form

$$\Delta_h f(x) = f(x + h) - f(x) \quad ,$$

whereas a backward difference has the form

$$\Delta_h f(x) = f(x) - f(x - h) \quad .$$

We use a FDM to resolve the transport equation, equation (3.14). We use an upstream FDM, which switches between using forward and backward difference, depending on the direction of the flow field. When calculating  $f(x)$  it will use  $f(x + h)$  if the direction of the flow field at  $x$  is in the negative  $x$ -direction. If the flow field is in the positive  $x$ -direction,  $f(x - h)$  will be used. This is called the upstream principle, and is often used when modeling flow in porous media.

## 4.6 Initial and boundary conditions

In Chapter 3 we choose the initial conditions for the enthalpy to be equal everywhere in the reservoir, starting with the same enthalpy in all grid points. We have

$$h_i = C \quad \forall i \leq (N + 2)^2 ,$$

where  $C$  is a set constant.

We saw in the previous chapter that the surrounding boundary of the reservoir was constant. As we use a FEM to discretize the diffusive term, the boundary condition will be on the boundary. We have chosen

$$h_{sb} = C ,$$

where  $C$  is the same constant as in the initial condition and the  $sb$  index means all the grid points on the boundary surrounding the reservoir. We also stated in Section 3.3 that there would be taken out enthalpy in the producing well, at the same rate for the whole of the production time. To model this numerically, we have chosen to take out the same amount of enthalpy in each time step. This is realistic, as it is likely that there is a requirement of a set amount of energy to be produced for the owner of the producing well. To model a production well, we use a Dirac delta distribution. This will only make sense when we divide the reservoir up into *REV*s and integrate over all of them. We will then have

$$q = \int s\delta(x - x_0) = s , \quad (4.3)$$

where  $x_0$  is the location of the production well, and  $s$  is a constant. The higher the constant  $s$ , the higher the enthalpy produced in the well at each time step, and the colder the reservoir gets. How much a well in a geothermal reservoir in nature can produce, are very much dependent on the flow field (that again is dependent on the pressure) in the reservoir. It is common to model find this flow field with a pressure solver. As this is not the part of the simulations that we are focusing on, we have chosen to neglect this dependency. Therefore the enthalpy that is taken out is independent of the flow field.

## 4.7 Numerical solution of the model equations

In Chapter 3 we saw that our mathematical model is based on conservation of enthalpy. Our model equations are based upon a local and a reservoir scale conservation law, equations (3.1) and (3.11). We still use enthalpy as defined in Section 2.3 in the reservoir mechanics chapter, with a similar proportionality constant between water and enthalpy, in both the water and in the block layers.

We have a numerical solver that solves all the flux terms with explicit time discretization, and three solvers that solve one flux term with explicit time discretization. When we use a discrete partial derivative of time, the reservoir scale conservation law in the water phase takes the form

$$\phi_0 \frac{h^{n+1} - h^n}{\Delta t} - \nabla \cdot (\kappa_0 \nabla h_0^\alpha) + \nabla \cdot (\underline{v} h_0^\beta) = c_0 (h_1^\gamma - h_0^\gamma) - q, \quad (4.4)$$

where  $\alpha, \beta, \gamma = n, n + 1$  depending on the terms being solved with implicit or explicit time discretization. Recall that  $\phi_0$  is a volume density, a fraction between 0 and 1, that express how much of the total volume that are in this layer and that  $\underline{v}$  is the flow velocity of the water phase.  $q$  is the sinks/sources that are inside the volume we model.

In the block layers the global conservation law will take the form

$$\phi_i \frac{h^{n+1} - h^n}{\Delta t} - \nabla \cdot (\kappa_i \nabla h_i^\alpha) = c_{i-1} (h_{i-1}^\gamma - h_i^\gamma) + c_i (h_{i+1}^\gamma - h_i^\gamma) - q, \quad (4.5)$$

with  $\alpha, \gamma = n, n + 1$ .

#### 4.7.1 All terms implicit

If every term is solved with implicit time discretization, our combined equation for the water phase is

$$\begin{aligned} & [\varphi_0 \mathbf{I} - \Delta t \mathbf{D} + \Delta t c_0 \mathbf{I} - 2 \frac{\Delta t}{\Delta x} v \mathbf{I}] h_{i=0,k}^{n+1} \\ & - \Delta t c_0 h_{i=1,k}^{n+1} + \frac{\Delta t}{\Delta x} v [h_{i=0,k \pm 1}^{n+1} + h_{i=0,k \pm (N+2)}^{n+1}] = \\ & \varphi_0 h_{i=0,k}^n, \end{aligned}$$

where  $k$  is the index for the grid point,  $i$  is the index for the layer,  $n$  is the index for the time step and  $\Delta t$  is the time step. We use a approximation matrix  $\mathbf{D}$  instead of the diffusion term, and  $\mathbf{I}$  is the identity matrix. We have altered all terms with time step  $n + 1$  on the left hand side of the equation, and with  $n$  on the right hand side. For the block layers, our combined equation is

$$\begin{aligned} & [\varphi_i \mathbf{I} - \Delta t \mathbf{D} + \Delta t c_{i-1} \mathbf{I} + \Delta t c_i \mathbf{I}] h_{i,k}^{n+1} \\ & - \Delta t c_{i-1} h_{i-1,k}^{n+1} - \Delta t c_i h_{i+1,k}^{n+1} = \\ & \varphi_i h_{i,k}^n, \end{aligned}$$

where  $i \geq 1$ .

### 4.7.2 Heat exchange term explicit

If we solve the heat exchange term explicitly, we get this equation for the block layers

$$\begin{aligned} & [\varphi_i \mathbf{I} - \Delta t \mathbf{D}] h_{i,k}^{n+1} = \\ & [\varphi_i - \Delta t c_{i-1} - \Delta t c_i] h_{i,k}^n + \Delta t c_{i-1} h_{i-1,k}^n + \Delta t c_i h_{i+1,k}^n \quad , \end{aligned}$$

and correspondingly for the water layer.

### 4.7.3 Diffusion term explicit

If we solve the diffusion term explicitly, we get this equation for the block layers

$$\begin{aligned} & [\varphi_i \mathbf{I} + \Delta t c_{i-1} \mathbf{I} + \Delta t c_i \mathbf{I}] h_{i,k}^{n+1} \\ & - \Delta t c_{i-1} h_{i-1,k}^{n+1} - \Delta t c_i h_{i+1,k}^{n+1} = \\ & [\varphi_i - \Delta t \mathbf{D}] h_{i,k}^n \quad , \end{aligned}$$

and correspondingly for the water layer.

### 4.7.4 Transport term explicit

If we solve the transport term explicitly, the equation for the water phase is

$$\begin{aligned} & [\varphi_0 \mathbf{I} - \Delta t \mathbf{D} + \Delta t c_0 \mathbf{I}] h_{i=0,k}^{n+1} \\ & - \Delta t c_0 h_{i=1,k}^{n+1} = \\ & [\varphi_0 + 2 \frac{\Delta t}{\Delta x} \underline{v} \mathbf{I}] h_{i=0,k}^n - \frac{\Delta t}{\Delta x} \underline{v} [h_{i=0,k\pm 1}^n + h_{i=0,k\pm(N+2)}^n] \end{aligned}$$

where  $\underline{v}$  is the flow velocity of the water phase. For the block layers we get the same equations as with all terms implicit.

### 4.7.5 Special cases with only one flux term

When only one flux term is transferring enthalpy in the model, we end up with three equations. Which equation we end up with is dependent on which flux term.

#### Heat equation

In the special case that only the diffusion flux is transferring enthalpy in the model, we end up with the heat equation,

$$\phi_i \frac{h_i^{n+1} - h_i^n}{\Delta t} - \nabla \cdot (\kappa_i \nabla h^\alpha) = 0 \quad . \quad (4.6)$$

Using a FEM, we use an approximation matrix  $\mathbf{D}$  instead of the diffusion term. We then get

$$\varphi_i \frac{h_{i,k}^{n+1} - h_{i,k}^n}{\Delta t} - \mathbf{D}h_{i,k}^\alpha = 0, \quad (4.7)$$

with  $\alpha$  either equal to  $n + 1$  or  $n$ .

If we solve the heat equation with implicit time discretization in the diffusion term, equation (4.7) becomes

$$\varphi_i \mathbf{I} h_k^{n+1} - \Delta t \mathbf{D} h_k^{n+1} = \varphi_i h_k^n,$$

where we have multiplied throughout with  $\Delta t$ , and  $\mathbf{I}$  is the identity matrix. Adding the matrices together we get

$$[\varphi_i \mathbf{I} - \Delta t \mathbf{D}] h_k^{n+1} = \varphi_i h_k^n. \quad (4.8)$$

We can solve the whole equation numerically, only using one matrix. If we solve the heat equation with explicit time discretization in the diffusion term, equation (4.7) we get

$$\varphi_i h_k^{n+1} = [\varphi_i \mathbf{I} + \Delta t \mathbf{D}] h_k^n. \quad (4.9)$$

### Heat exchange equation

In the special case, where only the heat exchange flux is transferring enthalpy in our model, we get a heat exchange equation. If the heat exchange equation (3.15), is solved with implicit time discretization in the heat exchange term we get

$$\varphi_i \frac{h_{i,k}^{n+1} - h_{i,k}^n}{\Delta t} = c_{i-1}(h_{i-1,k}^{n+1} - h_{i,k}^{n+1}) + c_i(h_{i+1,k}^{n+1} - h_{i,k}^{n+1}). \quad (4.10)$$

Multiplying by  $\Delta t$  and gathering the terms with time step  $n + 1$  on the left hand side, and the term with time step  $n$  on the right hand side we get

$$[\varphi_i + \Delta t c_{i-1} + \Delta t c_i] h_{i,k}^{n+1} - \Delta t c_{i-1} h_{i-1,k}^{n+1} - \Delta t c_i h_{i+1,k}^{n+1} = \varphi_i h_{i,k}^n. \quad (4.11)$$

Equation (4.11) is solved using a block matrix system, as seen in figure 4.4. Having  $h_{i-1}$ ,  $h_i$ , and  $h_{i+1}$  in the upper, middle and final third of the enthalpy vectors, will ensure that the matrix system calculate the enthalpy transfer in from the layers  $i - 1$  and  $i + 1$  to layer  $i$  implicitly. We need to place weighted diagonals on both sides of  $A_i$  and weights on the diagonal in  $A_i$  to calculate the enthalpy transfers. The weights on the diagonals will be decided by the coefficients  $\pm \Delta t c_k$ , where  $k = i, i - 1$  or  $i + 1$ , which comes from equation (4.11).

If the equation is solved with explicit time discretization in the heat exchange term, we get

$$\varphi_i h_{i,k}^{n+1} = \Delta t [c_{i-1}(h_{i-1,k}^n) + c_i h_{i+1,k}^n] + [\varphi_i - 2\Delta t c_{i-1}] h_{i,k}^n. \quad (4.12)$$

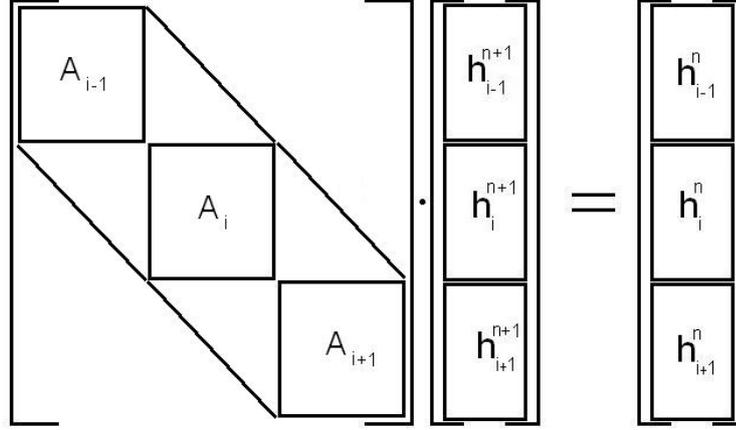


Figure 4.4: A block matrix, used to solve the heat exchange equation.

### Transport equation

In the special case where only the advection term is transferring enthalpy in our solver we get a heat transport equation. When solving the heat transport equation numerically, we use a upstream FDM. The equation has the form

$$\frac{\partial h}{\partial t} - \nabla \cdot (\underline{v}h) = 0 ,$$

where we can move the flow field vector  $\underline{v}$ , outside the divergence sign, since a divergence free flow field is used. This gives us

$$h^{n+1} - \Delta t \underline{v} \left[ \frac{\partial h_x^\beta}{\partial x} + \frac{\partial h_y^\beta}{\partial y} \right] = h^n , \quad (4.13)$$

with no  $z$  direction, as we have a 2D numerical model. We have either  $\beta = n$  or  $\beta = n + 1$ , dependent on the time discretization in the transport term.

Using finite differences for the partial derivatives in equation (4.13), gives us

$$\varphi_0 h_{i,k}^{n+1} - \Delta t \underline{v} \left[ \frac{h_{i,k}^\beta - h_{i,k\pm 1}^\beta}{\Delta x} + \frac{h_{i,k}^\beta - h_{i,k\pm(N+2)}^\beta}{\Delta y} \right] = \varphi_0 h_{i,k}^n .$$

But  $\Delta x = \Delta y$ , as we have a uniform grid, and we get

$$\varphi_0 h_k^{n+1} - \frac{\Delta t}{\Delta x} \underline{v} [2h_k^\beta - h_{k\pm 1}^\beta - h_{k\pm(N+2)}^\beta] = \varphi_0 h_k^n .$$

Solving the equation with implicit time discretization in the transport term gets us

$$[\varphi_0 - 2\frac{\Delta t}{\Delta x}v]h_k^{n+1} + \frac{\Delta t}{\Delta x}v[h_{k\pm 1}^{n+1} + h_{k\pm(N+2)}^{n+1}] = \varphi_0 h_k^n ,$$

whereas explicit time discretization gives us

$$\varphi_0 h_k^{n+1} = -\frac{\Delta t}{\Delta x}v[h_{k\pm 1}^n + h_{k\pm(N+2)}^n] + [\varphi_0 + 2\frac{\Delta t}{\Delta x}v]h_k^n .$$

### Flow field

We have chosen a radial flow field, where the water flow transports the enthalpy with the water flow to the production well. When a production well is drilled into a geothermal reservoir in nature, some water will be taken out and the pressure will sink near the well. Then the rest of the water will flow to the well, and the hot water will transport enthalpy. This is what we approximate with a radial flow field. The closer a grid point are to the well, the higher is the velocity in the flow field. The flow velocity is inverse proportional to the distance from the well, and we have

$$v_k \propto \frac{1}{|k - p|} ,$$

where  $k$  is the index of a grid point in the layer,  $p$  is the index of the grid point where the production well lies, and  $|k - p|$  is the distance between those two grid points. It is common to find this flow field with a pressure solver based on Darcy's equation and mass conservation, but we have chosen to have a analytical flow field instead as the focus is on the numerical methods and discretization of the equations.

#### 4.7.6 Courant–Friedrichs–Lewy condition

The Courant–Friedrichs–Lewy condition, from now on referred to as the CFL condition, is a necessary stability condition for convergence while solving partial differential equations with explicit time discretization, as we have done above. A consequence of solving it explicitly, is that the time step must be less than a certain time to get a stable solution. When solving the equations, there will be different CFL conditions, depending on which of the terms that are solved explicitly. In table 4.1, the CFL conditions when solving one of our equation terms explicitly is listed.

Solved explicitly	CFL condition
Diffusion term	$\frac{\max(\kappa_i)\Delta t}{\phi_i(\Delta x)^2} < C_{\text{diff}}, \quad 0 \leq i \leq k$
Heat exchange term	$\frac{\max(c_i + c_{i+1})\Delta t}{\phi_i} < C_{\text{exch}}, \quad 0 \leq i < k$
Transport term	$\frac{\max(\underline{v})\Delta t}{\phi_0} < C_{\text{trans}}$

Table 4.1: The CFL conditions if a term is solved explicitly. In the table the index  $k$  stands for the number of layers, and  $i$  stands for the  $i$ th layer. The volume density of layer  $i$  is  $\phi_i$ , the diffusion constant in layer  $i$  is  $\kappa_i$ , and  $\underline{v}$  is the flow velocity of the water phase. The constants  $C_{\text{exch}}$ ,  $C_{\text{diff}}$ , and  $C_{\text{trans}}$  depends on how the terms are solved numerically.

### No diffusion in the inner block layers

In our local conservation law for enthalpy there is no enthalpy transfer between the inner block layers and the water phase in the fractures between the blocks. The enthalpy first has to be transferred to the utmost layer, and then to the water. The diffusion flux is similar to the heat exchange flux. The difference is that the diffusion flux transfer enthalpy between the different blocks, while the heat exchange flux transfer enthalpy between the different layers. It is a logical assumption that the enthalpy transfer driven by the diffusion flux, needs to go through the utmost block layer before it is transferred to the other blocks. We have therefore chosen to have no diffusion term in the inner block layers in our simulations. There will still be a heat exchange flux between these layers.

# Chapter 5

## Results

In this chapter, we will present our simulation results with the different solution methods. First we set the coefficients that we use in the simulations, and the CFL conditions that are set as a consequence of these. The CFL conditions give a restriction on the time step when one equation term is solved with explicit time discretization. When we do this, a term is solved at the start of each time step. Using implicit discretization, a term is solved at the end of each time step. When resolving the equations with explicit discretization, operator splitting is used.

The term that is solved explicitly is solved on its own, not in the same matrix as the rest of the equation, and the memory use and complexity of the system is reduced. In the simulations we use a grid that has closer grid points inside each block layer than the grid points in the different layers are. This will give a stricter CFL condition in the diffusion term than in the heat exchange term, and it is reasonable to try to solve the heat exchange term explicitly. The flow field we use, does not give a strict CFL condition in the transport term, and therefore it is reasonable to solve the transport term explicitly as well. We develop a general adaptive strategy, which decides which terms that should be solved explicitly under different CFL conditions.

Next, an evaluation of our results with different solvers is conducted. To evaluate the results we will use norms and plots with the relative errors of the enthalpy in the simulation results. We will show that our different methods converge, and that the results from simulations with the same solver using different amount of grid points also converge. We compare the results with the the  $L_1$  norm. To have something to compare with, we use the heat explicit solver with many grid points as a reference solution. In addition to comparing the norms of the results and the plots with relative error of the enthalpy, we also compare the simulation time of

the solvers.

The objective of this chapter is to show that using the adaptive strategy, and thus run some of the terms explicitly, will be a good choice. When we follow the strategy, the heat exchange and the transport terms should be solved explicitly. We compare the solution from the all terms implicit solver, with the solution from solvers that does one term explicitly.

## 5.1 Coefficients and the subsequent CFL conditions

To model geothermal heat extraction, we need parameters and coefficients that reflects a specific depth and rock type. As we focus on the discretization of the equations, realistic parameters are not the priority in our model.

Coefficients that are used in the simulations has to be set to a certain value, such that the results from the different numerical solvers can be compared. In table 5.1 the coefficients that are used in the simulations are listed. In table 5.2, we see how large the coefficients ( $c_i$ ) in the heat exchange term will be in our model, when two block layers and different block sizes are used. The coefficients in table 5.2 is used to determine the heat exchange flux  $J_i$  from layer  $i + 1$  to layer  $i$  in equation (3.16). Table 5.3 lists the CFL conditions for the different terms with the coefficients chosen as in the table 5.1.

Recall that the enthalpy is defined as in section 2.3, with the same proportional constant between temperature and enthalpy in the water and in the block layers. With the initial level of enthalpy the same in the water and in the layers, the initial temperature is also the same. This enthalpy level corresponds to approximately 300°C. When deep geothermal reservoirs is modeled this initial temperature will be below the boiling point for water, because of the pressure at such depths (several kilometers down).

<b>Coefficient</b>	<b>Value</b>
Diffusion const. ( $\kappa_0$ ) water	0.0573
Diffusion const. ( $\kappa_1$ ) block layer 1	0.1585
Diffusion const. ( $\kappa_2$ ) block layer 1	0
Enthalpy initially	$1.26 \cdot 10^6$
Enthalpy taken out each time step in the well in the water phase	$750 \cdot \Delta t$
Enthalpy taken out each time step in the well in rock layer 1	$2463 \cdot \Delta t$
Enthalpy taken out each time step in the well in rock layer 2	$2463 \cdot \Delta t$
Volume density water	0.015
Volume density block layer 1 and 2	0.4925
Number of block layers	2

Table 5.1: Coefficients used in the simulations.

<b>Block diameter / i</b>	<b>Water and block layer 1</b>	<b>Block layer 1 and 2</b>	<b>Block layer 2 and 3</b>
Gravel	$0.9601 \cdot 10^{-10}$	$0.9822 \cdot 10^{-10}$	$0.2770 \cdot 10^{-10}$
1 meter	$0.3662 \cdot 10^{-6}$	$0.3747 \cdot 10^{-6}$	$0.1057 \cdot 10^{-6}$
10 meter	$0.3662 \cdot 10^{-3}$	$0.3747 \cdot 10^{-3}$	$0.1057 \cdot 10^{-3}$
100 meter	0.3662	0.3747	0.1057

Table 5.2: The heat exchange coefficients when the blocks in the reservoir is gravel with diameter 0.064 m, blocks with diameter 1 meter, 10 meter and 100 meter. We have used 10 meter in the simulations. There are three block layers in the table, and the coefficients determine the enthalpy flux between the layers. In our simulations we have only used two block layers.

<b>Solved explicitly</b>	<b>CFL condition</b>
Diffusion term	0.0025
Heat exchange term	0.6667
Transport term	0.8

Table 5.3: The time step that is used in the simulations when a term is solved with explicit time discretization, due to the CFL conditions. The coefficients in the table 5.1 is used, and  $N = 23$ . (Recall that  $N$  is the number of internal nodes in our grid, as we defined in Section 4.2.)

### 5.1.1 Physical interpretation of the grid and layers

The solvers gives the user the choice to use as many layers and as many grid points in each layer as desired. The number of grid points in one grid layer is higher than the number of layers in our simulations. The physical interpretation of more grid points inside each layer than number of layers, is that the resolution inside each layer is higher than the resolution of layers. Because of this it could be argued that the diffusion term could be solved for each block and each fracture, because there are enough grid points to do this. If it is not known exactly how the reservoir that is modeled look, this is impossible. (In most cases it is not known exactly how it looks.) Therefore we use a statistical approach, where the variables for each layer are averages. As the gradients near a fracture will be large, there is a need of high resolution in and near the fractures.

We have chosen to set the volume densities of all the block layers to be the same, with

$$\varphi_1 = \varphi_2 = \dots = \varphi_k. \quad (5.1)$$

As we have chosen the block layers to have the same volume density, the area they cover will be equal in our 2D representation. As a consequence of this, the grid points will lay closer near the surface of the block layers and we see this in figure 4.3. The surface of a layer will be larger the nearer is to the water phase. This make physical sense, as there will be more heat flow and interaction with the water phase at the surface of the blocks.

## 5.2 Adaptive strategy

Our aim is to show that using explicit discretization for some of the flux terms, will be a better strategy than using implicit discretization for all the terms. The main arguments for solving some of the terms explicit are:

- Because of complexity and memory use, there has to be a good reason to choose implicit discretization for all the equation terms.
- The transport term is the hyperbolic part of our equation. Hyperbolic terms is often solved explicitly in modeling, although there are scientists that prefer to solve it implicitly. In this thesis we have used a linear transport term, and it is trivial to solve the term either explicitly or implicitly. If a non linear term is used, it will be more complex to solve the term implicitly. To solve it explicitly will still be trivial. Therefore it is more common to solve the term with explicit time discretization. If the transport term is solved explicitly, then it is a good strategy to also solve the terms that have less strict CFL conditions explicitly. The time step limit due to the CFL conditions for the

terms in the model equations are  $C_{\text{diff}}$ ,  $C_{\text{trans}}$ , and  $C_{\text{exch}}$  for the diffusion term, transport term, and heat exchange term respectively. For example if  $C_{\text{diff}} \ll C_{\text{trans}} \ll C_{\text{exch}}$ , it will make sense to resolve the heat exchange explicitly. This will not make the CFL conditions on the time step any stricter, and the complexity of the system and memory use will be reduced.

- It is a natural next step to implement a domain decomposition (DD) solver. The strategy behind DD is to divide the layers into different domains, and solve them in parallel. This would be hard with a implicit discretization, so if there are no loss of regularity, solving some of the terms with explicit discretization would make sense.

If the transport term is solved explicitly, our adaptive strategy is to solve the other terms that have less strict CFL conditions explicitly. The transport term will as mentioned, more often than not be solved explicitly when the transport term is non linear. This is also often done when it is linear. The complexity of the problem, the memory use, and thus the simulation time will go down when we follow this strategy. The CFL condition will stay the same, and the same time step can be used. There may also be simulation cases where the condition is stricter for another term, but it still would be better to solve the term in question explicitly. The evaluation of the discretization to use in such a case, is the same evaluation that would have been done if the transport term is solved implicitly.

If the transport term is solved implicitly, our adaptive strategy will also be to solve the term(s) with CFL condition(s) that are not strict explicitly. In this case it is not as straight forward as if the transport term is solved explicitly. When the time step that is used does not break the CFL condition for one term, the strategy will be to solve the term explicitly. If it would require a slightly smaller time step, the same holds. When the CFL condition demands a much smaller time step, a individual evaluation would have to be done.

### 5.3 Evaluation of results

To evaluate the results, we use norms and relative error. To compare the norms the results have to be interpolated, such that they have the same amount of grid points. Here follows a description of the norms that we have used and how we have interpolated. Then we will see how the different methods converge and we will compare the results from three of our solvers, the heat exchange explicit solver, the transport term explicit solver, and the all implicit solver.

### 5.3.1 Norms

A norm is a function that gives a positive length from one vector to another, but norms are defined in different ways. Two norms that are commonly used in the world of mathematics are the  $L_1$  and  $L_2$  norms. The discrete definition of the norms is:

#### $L_1$ norm

$$\|x\|_1 = \sum_{i=1}^n |x(i)|, \quad (5.2)$$

#### $L_2$ norm

$$\|x\|_2 = (\sum_{i=1}^n [x(i)]^2)^{1/2}, \quad (5.3)$$

where  $n$  is the number of elements in the vector [13].

From a physical perspective, the  $L_1$ -norm should be used when discussing preserved quantities, such as enthalpy. The  $L_1$ -norm tells us exactly how much the difference between the two vectors (that contains a conserved quantity) are. When discussing quantities that are not preserved, such as speed, it is more natural to use the  $L_2$ -norm. Since we are using enthalpy in our model, we use the  $L_1$ -norm.

### 5.3.2 Interpolation

To use norms to compare two solutions, the same number of grid points has to be used. There are two ways to do this, either to interpolate the solution with the fewest grid points such that both solutions have the same amount, or to just use some of the grid points in the solution with the most grid points. As some of the point of doing simulations with more grid points is to get a better solution, it seems better to interpolate the solution with the fewest grid points.

To do this we use linear interpolation of the solution in the old grid points to create the solution in the new grid points. A linear interpolation has its limits, therefore the solution with the fewest grid points will have some error when compared to the solution with the most, even if the solution is the same in the common grid points of the solutions.

### 5.3.3 Convergence

To know if the different solvers are stable and give approximately the same solutions, a convergence test has to be done. How many grid points, block layers, and which solver that is used should be irrelevant, all the solutions should converge.

### Late time asymptotic

When we run different solvers with the production well open for the whole simulation, the solution shall in theory not change after being run for a long time. The solution has reached equilibrium. This is in fact also what we observe when we run our solvers with different running time. As an example of this observation, we have plotted the enthalpy versus the time in one grid point, in figure 5.1. In the comparison section, we use norms to see that equilibrium is reached after some time.

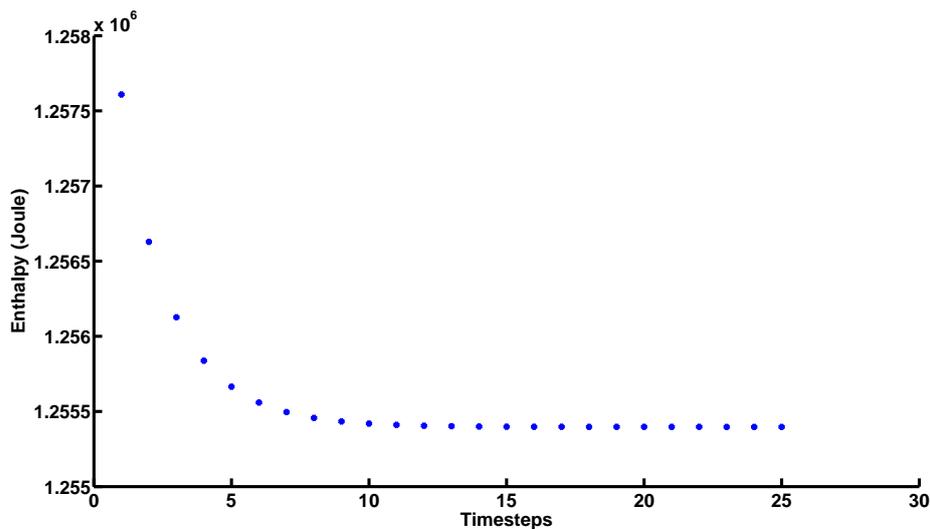


Figure 5.1: Here we plot the enthalpy in one grid point ( $x=0.6,y=0.5$ ) against time.

### Refinement

Running the solvers with different amount of grid points converge to one solution. The smaller grid cells we are using, the higher accuracy the solution has. In theory we should get the same solution in common grid points. In the figures 5.2, and 5.3 we see that the solution from the implicit solver converges when we use different sized grid cells, in the water phase and first block layer respectively. The plots are only examples that shows that the solvers converge.

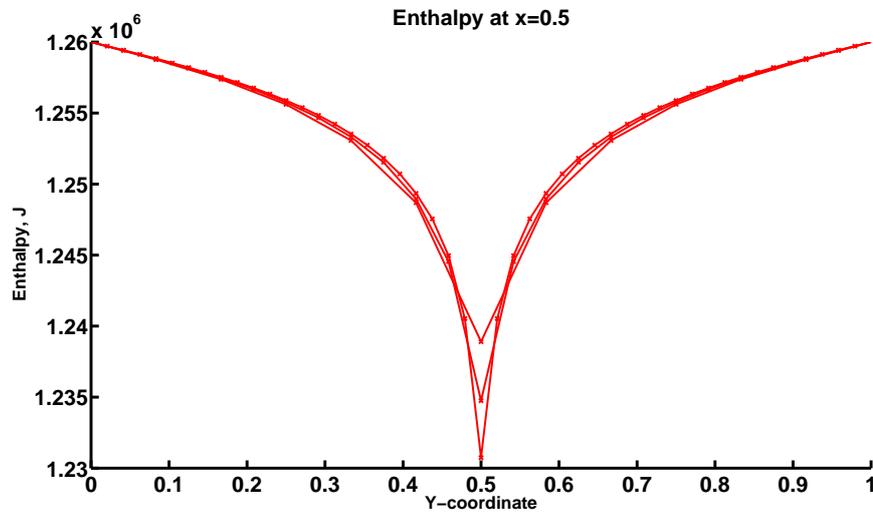


Figure 5.2: The solution in the water phase solved with different amount of grid points converges. In the plot the all implicit solver with  $N = 11, 23, 47$  is used.

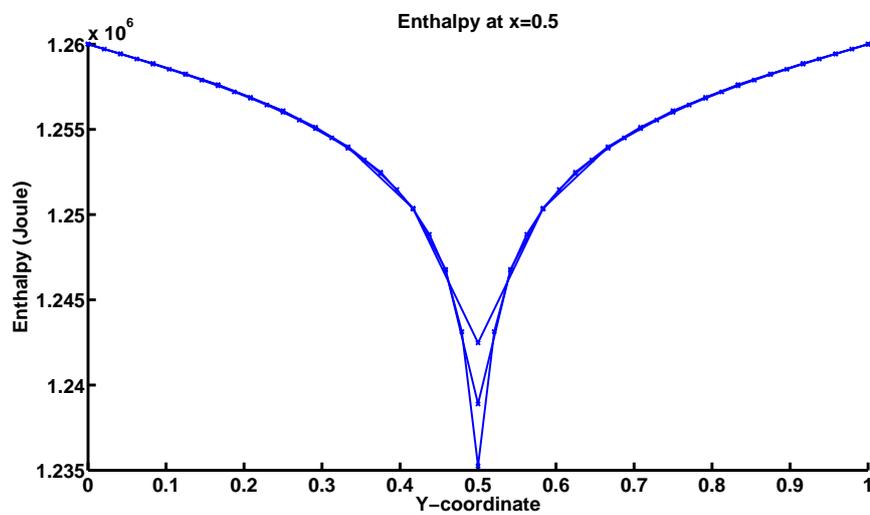


Figure 5.3: The solution in the first block layer solved with different amount of grid points converges. In the plot the all implicit solver with  $N = 11, 23, 47$  is used.

### Different solvers

If a solver discretizes a term explicitly or implicitly it should in theory give approximately the same result. In the plot 5.4, we observe that the different solvers obtains approximately the same solutions. When we later in this chapter compare the relative errors in the solvers, we will also observe that they produce approximately the same solutions.

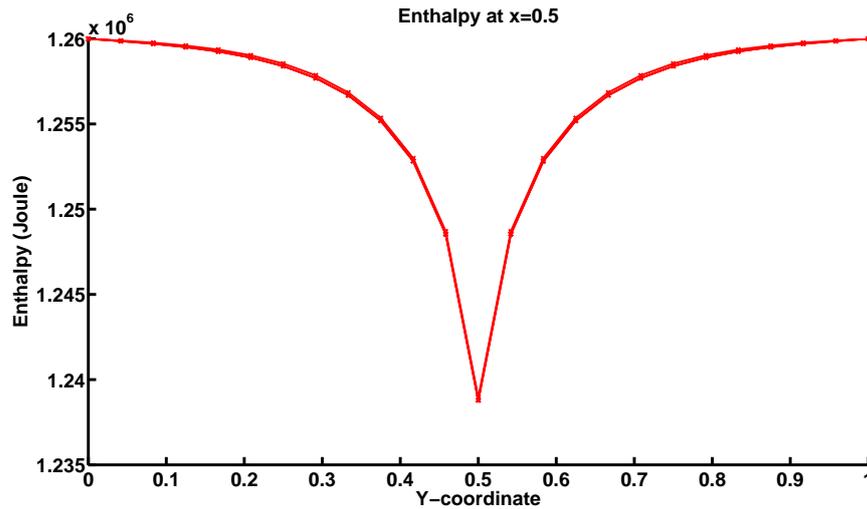
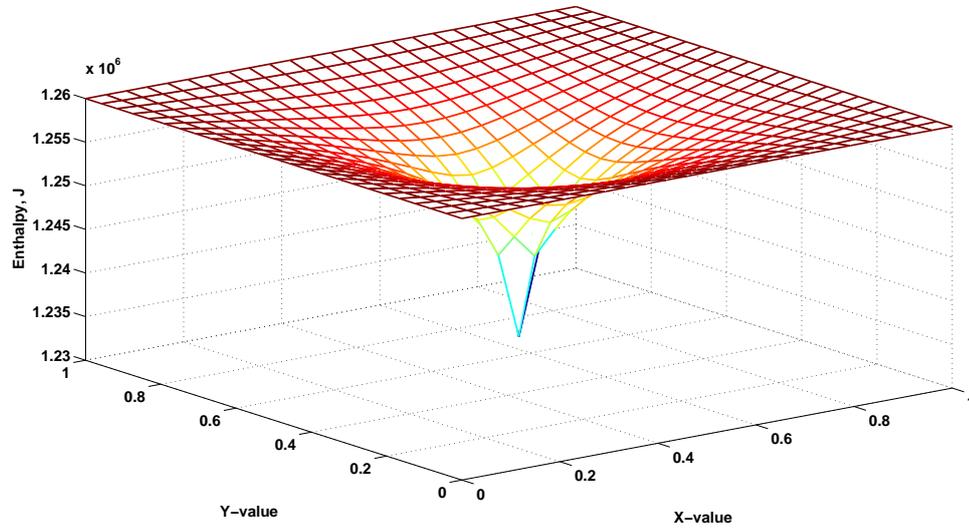
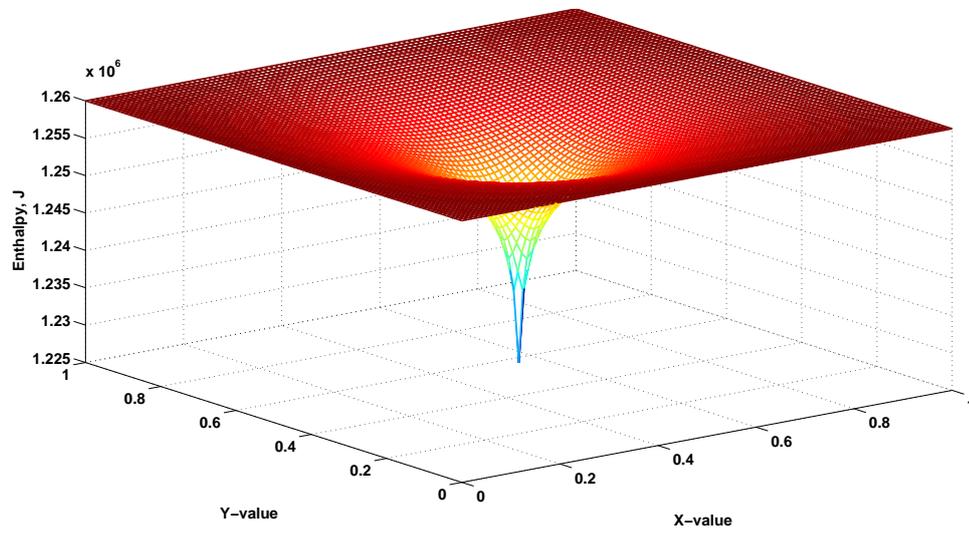


Figure 5.4: The solution in the water phase solved with the different solvers. In the plot  $N = 23$  is used.

### 5.3.4 Solution plots

In the plots 5.5 and 5.6 we see the shape of our solution. The solution has a singularity in the production well. We see that the enthalpy levels sink towards the well, as there is taken out enthalpy there.

Figure 5.5: Solution with  $N = 11$ .Figure 5.6: Solution with  $N = 95$ .

### 5.3.5 Comparison of the solvers

With the coefficients as in table 5.1, the CFL conditions for the diffusion term are stricter than for the heat exchange and the transport terms. If we follow our adaptive strategy it is clear that the heat exchange term and/or the transport term should be solved explicitly, as this will not give a strict CFL condition. In this comparison we will only compare the results from simulations where one of the heat explicit and transport terms is solved explicitly, and when all the terms are solved implicitly. We have to verify that the strategy is good, and to do this the first step is to check if the solution loses regularity when a term is solved explicitly.

To check the regularity of the solutions we compare the results from the simulations to the reference solution. The heat exchange explicit solver with  $N = 95$  has been used as the reference solution. The three solvers are run with  $N = 11, 23, 47$ , with different simulation time for all solvers, and different time steps in the implicit solver. The simulation time is the observation time, for which we observe the heat distribution in the reservoir.

The simulations we have done starts after the injection time, so the production well is open for the whole simulation. The solution from each simulation is interpolated, such that it gets as many grid points as the one with  $N = 95$ . To compare the solutions, the  $L_1$  norm is calculated. We also compare plots of the relative error along the line  $x = 0.5$ , for some of the simulations. The calculation times of the three solvers are also discussed.

#### $L_1$ norm calculations

In the tables 5.4, 5.5, 5.6, 5.7, and 5.8 the  $L_1$ -norms for simulations with different simulation times are listed. The CFL condition and the total simulation time decides the time step in the heat exchange explicit and in the transport solvers. The time step will be the largest possible time step that is allowed by the CFL condition that goes up in the simulation time, such that it is run for the same time as the implicit solver. The time step in the implicit solver will be different from simulation to simulation, to see how the results vary.

	<b>Solver</b>					
	All implicit		Heat exchange explicit		Transport explicit	
<b>N</b>	Water phase	Rock layer 1	Water phase	Rock layer 1	Water phase	Rock layer 1
11	$1.6062 \cdot 10^6$	$0.8174 \cdot 10^6$	$0.8086 \cdot 10^6$	$1.4187 \cdot 10^6$	$0.6329 \cdot 10^6$	$1.0438 \cdot 10^6$
23	$1.1285 \cdot 10^6$	$0.6117 \cdot 10^6$	$0.5545 \cdot 10^6$	$1.2099 \cdot 10^6$	$0.5323 \cdot 10^6$	$0.9521 \cdot 10^6$
47	$0.5373 \cdot 10^6$	$0.3050 \cdot 10^6$	$0.3350 \cdot 10^6$	$0.5811 \cdot 10^6$	$0.5383 \cdot 10^6$	$0.7955 \cdot 10^6$

Table 5.4: The  $L_1$ -norms for the results from the three solvers, when the production well is open for the whole simulation time, and the simulation is run for 10 time steps with time step 0.2 in the all implicit solver.

	<b>Solver</b>					
	All implicit		Heat exchange explicit		Transport explicit	
<b>N</b>	Water phase	Rock layer 1	Water phase	Rock layer 1	Water phase	Rock layer 1
11	$4.0275 \cdot 10^6$	$3.5750 \cdot 10^6$	$0.8086 \cdot 10^6$	$1.4187 \cdot 10^6$	$0.6329 \cdot 10^6$	$1.0438 \cdot 10^6$
23	$4.3888 \cdot 10^6$	$3.6872 \cdot 10^6$	$0.5545 \cdot 10^6$	$1.2099 \cdot 10^6$	$0.5323 \cdot 10^6$	$0.9521 \cdot 10^6$
47	$4.5640 \cdot 10^6$	$3.7884 \cdot 10^6$	$0.3350 \cdot 10^6$	$0.5811 \cdot 10^6$	$0.5383 \cdot 10^6$	$0.7955 \cdot 10^6$

Table 5.5: The  $L_1$ -norms for the results from the three solvers, when the production well is open for the whole simulation time, and the simulation is run for 1 time steps with time step 2 in the all implicit solver.

	<b>Solver</b>					
	All implicit		Heat exchange explicit		Transport explicit	
<b>N</b>	Water phase	Rock layer 1	Water phase	Rock layer 1	Water phase	Rock layer 1
11	$3.1195 \cdot 10^6$	$0.9798 \cdot 10^6$	$2.9367 \cdot 10^6$	$1.0014 \cdot 10^6$	$3.0727 \cdot 10^6$	$0.9818 \cdot 10^6$
23	$2.4088 \cdot 10^6$	$0.6127 \cdot 10^6$	$2.4062 \cdot 10^6$	$0.6184 \cdot 10^6$	$2.3712 \cdot 10^6$	$0.6186 \cdot 10^6$
47	$1.0957 \cdot 10^6$	$0.1708 \cdot 10^6$	$1.0967 \cdot 10^6$	$0.1714 \cdot 10^6$	$1.0804 \cdot 10^6$	$0.1778 \cdot 10^6$

Table 5.6: The  $L_1$ -norms for the results from the three solvers, when the production well is open for the whole simulation time, and the simulation is run for 25 time steps with time step 0.2 in the all implicit solver.

	<b>Solver</b>					
	All implicit		Heat exchange explicit		Transport explicit	
<b>N</b>	Water phase	Rock layer 1	Water phase	Rock layer 1	Water phase	Rock layer 1
11	$1.7487 \cdot 10^6$	$1.2402 \cdot 10^6$	$2.9378 \cdot 10^6$	$1.0014 \cdot 10^6$	$3.0727 \cdot 10^6$	$0.9818 \cdot 10^6$
23	$1.1314 \cdot 10^6$	$0.9880 \cdot 10^6$	$2.4076 \cdot 10^6$	$0.6182 \cdot 10^6$	$2.3712 \cdot 10^6$	$0.6186 \cdot 10^6$
47	$0.3043 \cdot 10^6$	$0.6680 \cdot 10^6$	$1.0981 \cdot 10^6$	$0.1710 \cdot 10^6$	$1.0804 \cdot 10^6$	$0.1778 \cdot 10^6$

Table 5.7: The  $L_1$ -norms for the results from the three solvers, when the production well is open for the whole simulation time, and the simulation is run for 2 time steps with time step 2.5 in the all implicit solver.

N	Solver					
	All implicit		Heat exchange explicit		Transport explicit	
	Water phase	Rock layer 1	Water phase	Rock layer 1	Water phase	Rock layer 1
11	$3.1785 \cdot 10^6$	$0.9780 \cdot 10^6$	$3.1785 \cdot 10^6$	$0.9780 \cdot 10^6$	$3.1785 \cdot 10^6$	$0.9780 \cdot 10^6$
23	$2.4488 \cdot 10^6$	$0.6071 \cdot 10^6$	$2.4488 \cdot 10^6$	$0.6071 \cdot 10^6$	$2.4488 \cdot 10^6$	$0.6071 \cdot 10^6$
47	$1.1049 \cdot 10^6$	$0.1669 \cdot 10^6$	$1.1049 \cdot 10^6$	$0.1669 \cdot 10^6$	$1.1049 \cdot 10^6$	$0.1669 \cdot 10^6$

Table 5.8: The  $L_1$ -norms for the results from the three solvers, when the production well is open for the whole simulation time, and the simulation is run for 100 time steps with time step 0.2 in the all implicit solver.

In the simulations that produce the norms that are given in the tables 5.4-5.8 the production well is open for the whole simulation time. The initial condition for the enthalpy was set to  $1.26 \cdot 10^6$  (Not joules, because we use an inaccurate proportionality constant between temperature and enthalpy, but another 'entity' that we do not give a name.), and that is the reason why the norms produce such high numbers. The interpolated solutions have 9801 grid points, and if the norms are divided with the number of grid points, then we see that if the norm is  $0.5 \cdot 10^6$  (it is around this level in many cases), then the error on average is only 51 per grid point. This will only be 0.15 percent of the difference between the maximum and minimum value in the reference solution.

Overall the results seem to be about the same with the three solvers. We see in the table 5.8 that when the simulation is run for a long time, the three solvers gives exactly the same results. This has to do with equilibrium being reached when the simulation time is long enough. In table 5.5 we observe that with a long time step in the implicit solver, the solution does not converge. The reason behind this is that the error due to the time discretization is dominant over the error due to the spatial discretization. We observe in the table 5.7 that when the simulation time is longer, the results will converge even with a long time step. The solution will be closer to equilibrium, and because of this and the fact that two time steps is used, the error due to the time discretization will not dominate.

### Relative error

We have used two methods to determine the relative error, and the first steps in them are:

- We choose the common grid points from the reference solution and the solution we compare with.
- We have interpolated the solution we compare with the reference solution, such that it has as many grid points.

The second step in both is to divide by the difference between maximum and minimum of the reference solution,

$$\text{rel. error} = \frac{\text{difference of solutions}}{\max(\text{ref. solution}) - \min(\text{ref. solution})} \cdot 100 \text{ percent} \quad . \quad (5.4)$$

All the plots of the relative error are of the enthalpy in the water phase along  $x = 0.5$ , and the simulation time is the same in all the plots. In the plots 5.7 and 5.8 we see plots of the relative error for simulations with the heat exchange explicit solver. In plot 5.7 we see the relative error when the common grid points are used. In plot 5.8, we see the relative error when the solutions are interpolated to have as many grid points as the reference solution. As we have a singularity in the well, there is large relative error near it. Therefore we have produced the plots 5.9 and 5.10 that plots the solution on the one side of the singularity. In both the plots the relative error is below 3 percent until we reach the singularity, with both the methods. We also observe that the relative error is smaller when we use the common grid points. This is natural, as there will be an interpolation error when we interpolate the solutions. Another observation is that the relative error is smaller when more grid points are used.

In the plots 5.11 and 5.12, the relative error from simulations with three different solvers are compared. We again see that the relative error are below 3 percent. The solutions from the different solvers are slightly different, but approximately the same.

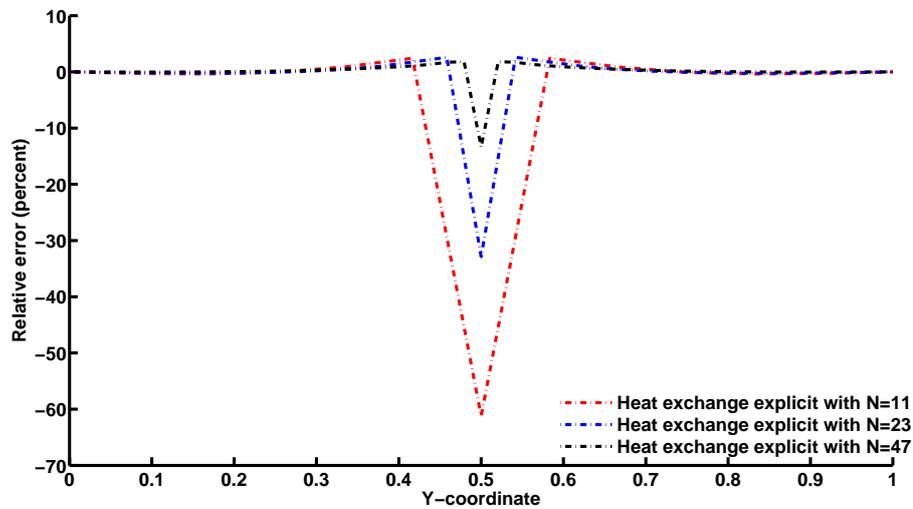


Figure 5.7: The relative error of the solution with the heat exchange explicit solver with different sized grid cells, when it is compared to the reference solution. The common grid points in the reference solution and the simulation solution have been plotted.

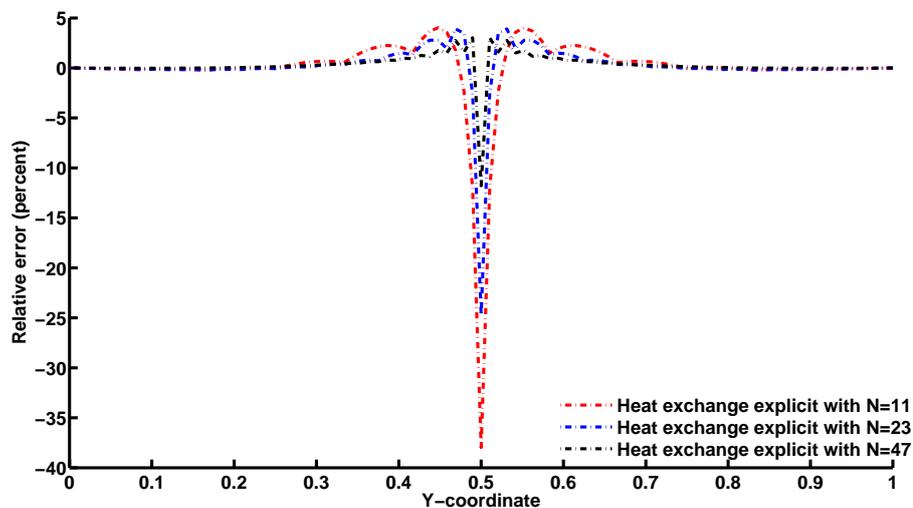


Figure 5.8: The relative error of the solution with the heat exchange explicit solver with different sized grid cells, when it is compared to the reference solution. The solutions have been interpolated linearly, such that they have the same grid points as the reference solution.

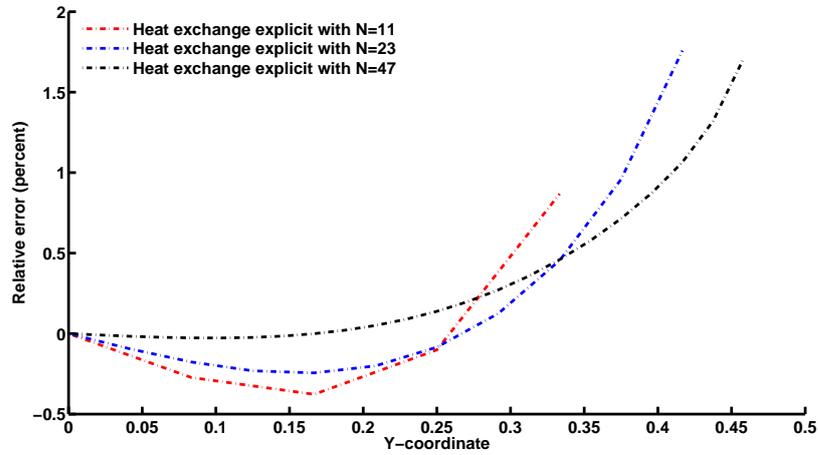


Figure 5.9: The relative error of the solution with the heat exchange explicit solver with different sized grid cells, when it is compared to the reference solution. The common grid points in the reference solution and the simulation solution have been plotted. Only the left side of the reservoir has been plotted, such that the well is excluded.

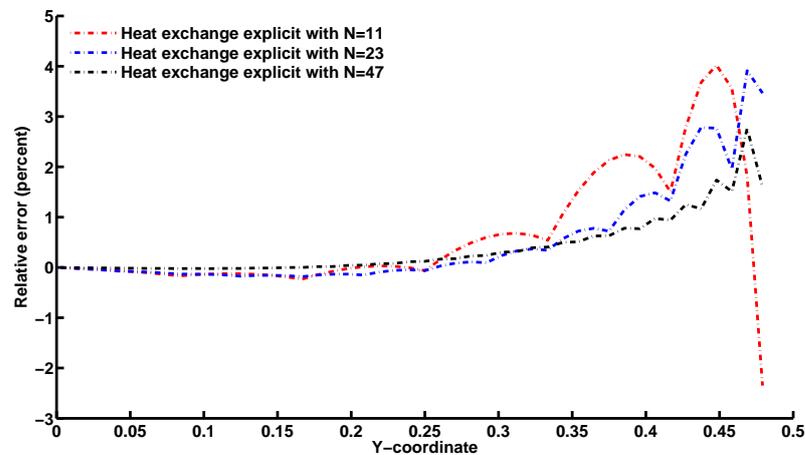


Figure 5.10: The relative error of the solution with the heat exchange explicit solver with different sized grid cells, when it is compared to the reference solution. The solutions have been interpolated linearly, such that they have the same grid points as the reference solution. Only the left side of the reservoir has been plotted, such that the well is excluded.

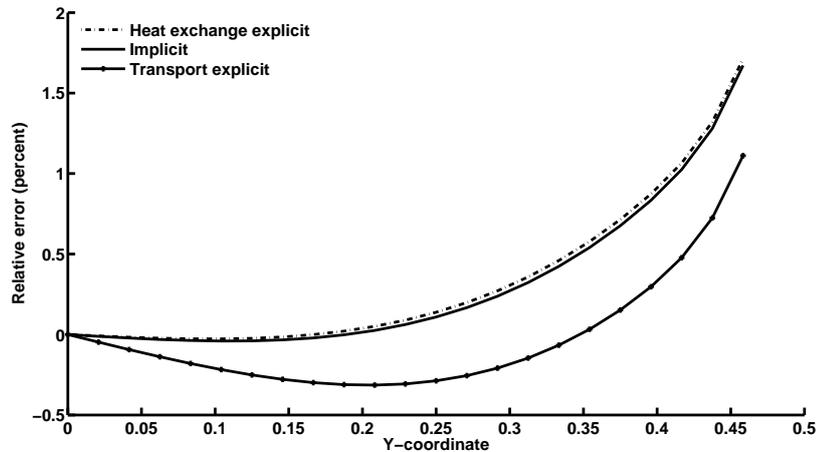


Figure 5.11: The relative error of the solution with the heat exchange explicit solver, the transport explicit solver and the implicit solver with  $N = 47$ , when it is compared to the reference solution. The common grid points in the reference solution and the simulation solution have been plotted. Only the left side of the reservoir has been plotted, such that the well is excluded.

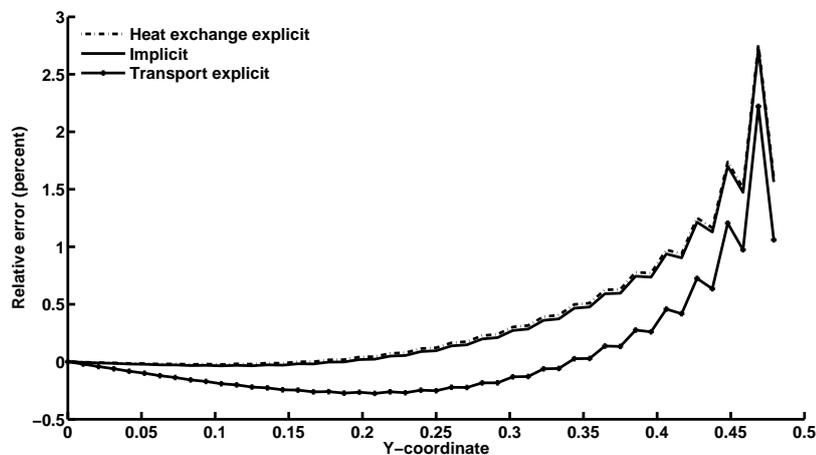


Figure 5.12: The relative error of the solution with the heat exchange explicit solver, the transport explicit solver and the implicit solver with  $N = 47$ , when it is compared to the reference solution. The solutions have been interpolated linearly, such that they have the same grid points as the reference solution. Only the left side of the reservoir has been plotted, such that the well is excluded.

### Calculation time

Between the heat exchange explicit solver and the all implicit solver there is substantial difference in calculation time unless the time steps in the implicit solver are very large. The transport explicit solver lies somewhere in between. In table 5.9 we see the calculation time compared between the solvers with set simulation time, time step, and different amount of grid points.

The solvers we have created and used for the simulations are not the most efficient. We have used Matlab's programming language, which is not the fastest programming language to solve large matrix systems in, and the numerics are not optimized. The calculation time of the simulations can therefore be a bit misleading, and it is difficult to conclude that the one solver is substantially faster than the other. However, the indications are that the heat exchange explicit solver is faster than the all implicit solver. It is reasonable to assume that some of the reason behind this is that we get a more sparse matrix when solving the equations.

N	Heat exchange	All implicit	Transport explicit
11	8.70	10.37	9.47
23	16.38	28.07	20.34
47	77.97	250.47	160.74

Table 5.9: Calculation time in seconds for the three solvers when the time step is 0.2, and the simulation is run for 10 time steps in the implicit solver.

We do not lose regularity when a term is solved with explicit time discretization. The terms with slight CFL conditions should have explicit time discretization, as the calculation time is reduced when we do so. Operator splitting has ensured that the matrix(es) that is(are) solved is(are) less complex, and the memory use has gone down. Thus the adaptive strategy is a good one, and there are no reasons not to follow it.

# Chapter 6

## Summary and conclusion

In this chapter, we summarize this master thesis. We come to a conclusion of our research, and point at possible improvements that can be done to our model.

### 6.1 Summary

In this master thesis we have used background knowledge from geothermal energy and reservoir mechanics, to both create a mathematical model and a numerical model for geothermal energy extraction. We model a reservoir filled with blocks of rock and fractures, that is filled with water, between them. The mathematical model is based on a local and a reservoir scale conservation law for enthalpy. The local conservation law models enthalpy conservation in one block and the fractures next to it.

To model the local conservation law, we introduced layers in the blocks. The local conservation law transfers enthalpy between these layers. The reservoir scale conservation law models conservation of enthalpy inside a REV. The flux term in the reservoir scale conservation law consists of three fluxes, a diffusive flux, a advective flux and a heat exchange flux. The diffusion term and the heat exchange term are both diffusive fluxes, but while the diffusion term transfer enthalpy inside each block layer, the heat exchange transfer enthalpy between the different block layers. The heat exchange flux is based on the local conservation law.

This mathematical model was then developed into numerical models, that modeled the reservoir scale conservation law. We made four numerical models. In all of them we used a FEM for spatial discretization of the diffusion term, and a FDM for spatial discretization of the transport term. The first numerical model had implicit time discretization in all the equation terms. The three other solvers had

one term each that was solved with explicit time discretization.

We created an adaptive strategy that decide which terms that should be computed explicitly, when the model equations are solved numerically. We then followed this strategy, in our simulations. To test the different numerical solvers against each other we made a simple test case, with one production well in the middle of the reservoir.

## 6.2 Conclusion

The comparison of the solutions with the different solvers shows that our adaptive strategy when only one term is solved explicitly works. In the simulations we have only tested the strategy using explicit time discretization for one term at a time, and this reduces the complexity of the problem and the memory use goes down. The strategy in that case was to solve the term with the slightest CFL condition explicitly.

If the transport term is solved with explicit time discretization, and it often is, then our strategy was to also solve the terms that do not have stricter CFL conditions than the transport term explicitly. If the transport term is solved implicitly or there are no terms that do not have stricter conditions, the terms with the slightest conditions should be done explicitly. It needs to be individually determined how many terms that should be done explicitly. A natural next research step would be to create solvers that do more terms with explicit time discretization.

There is no loss of regularity in the solutions when we do one term with explicit time discretization. In addition the calculation time is reduced, when a term with slight CFL condition is done explicitly. This implies that doing terms explicitly saves calculation time. If several terms are solved explicitly, the complexity of the problem will be further reduced. Therefore, our adaptive strategy is a good one.

With implicit time discretization, a very large time step has to be used to bring the calculation time down to the same level that the explicit solvers use. Using a large time step usually implies that the error due to the time discretization is the dominant error. There is no point in doing a grid refinement, when the time discretization error is dominant.

When all the terms are solved implicitly, the terms are combined into a large block matrix that is used to calculate the enthalpy. Solving a term with explicit discretization is done with operator splitting. The problem is split into smaller matrices, and the complexity and memory use is reduced. This also ensures that the different terms do not smoothen each other out, as is the case if all the terms are solved in one large matrix.

There is no reasonable argument that backs up the idea to always solve equations with implicit time discretization. Only in some special cases, where the CFL

conditions in all terms are very strict, an implicit time discretization in all terms can be defended, for this is the exception and not the rule.

### 6.3 Possible additions

One of the natural additions to our model is to build solvers that do several of the terms explicit at once, and that solve the diffusion and the transport term with explicit time discretization in some layers and implicit in others. The last part would require much implementation. To save further calculation time a Domain Decomposition method could be used, and the calculations could be done in parallel for the layers. It would also be a natural improvement to implement a pressure solver that is used to create the flow field in the transport term. To use a non-linear temperature distribution, where the temperature rises above or sinks below the boiling point for water could also be interesting.



# Bibliography

- [1] J. W. Lund, *Geothermal energy focus: Tapping the earth's natural heat*, Elsevier Ltd, 2006.
- [2] J. W. Lund and D. H. Freeston, *World-wide direct uses of geothermal energy 2000* Geothermics Vol. 30, Elsevier, 2001.
- [3] J. W. Lund et al., *Worldwide Direct-Uses of Geothermal Energy 2005*, Geothermics Vol. 34, Elsevier, 2005.
- [4] R. Bertani, *World Geothermal General 2001-2005 – State of the Art*, Geothermics Vol. 34, Elsevier, 2005.
- [5] A. Kagel et al., *A Guide to Geothermal Energy and the Environment*, Geothermal Energy Association, 2005.
- [6] D. K. Garman et al., *Enhanced Geothermal Systems*, Geothermal Technologies Program, U.S. Department of Energy, 2004.
- [7] J. W. Tester et al., *The Future of Geothermal Energy, MIT report*, Massachusetts Institute of Technology, 2006.
- [8] *What is an Enhanced Geothermal System*, Earth Sciences Division, Berkeley Lab, 2009.
- [9] Ø. Pettersen, *Grunnkurs i RESERVOARMEKANIKK*, Matematisk institutt, Universitetet i Bergen, 1990.
- [10] J. Bear, *dynamics of fluids in porous media*, American Elsevier, 1972.
- [11] Z. Chen et al., *Computational Methods for Multiphase Flows in Porous Media*, Society for Industrial and Applied Mathematics, Philadelphia, 2006.
- [12] D. Braess, *Finite elements*, Cambridge, 2007.
- [13] W. Cheney, *Analysis for Applied Mathematics*, Springer, 2001.

- [14] L. C. Evans, *Partial Differential Equations*, American Mathematical Society, 1998.
- [15] R. J. LeVeque, *Finite Difference Methods for Ordinary and Partial Differential Equations*, Siam, 2007.
- [16] K. Pruess et al., *TOUGH 2 User's guide, version 2.0*, Earth Sciences Division, Lawrence Berkeley National Laboratory University of California, 1999.
- [17] <http://wallpaper-s.org/>, *Death Valley with dry rock wallpaper*, 2009.
- [18] J. E. Warren and P. J. Root, *The Behavior of Naturally Fractured Reservoirs*, Society of Petroleum Engineers Journal, AIME, 1963.
- [19] K. Pruess and T. N. Narasimhan, *A Practical Method for Modeling Fluid and Heat Flow in Fractured Porous Media*, Society of Petroleum Engineers Journal, 1985.
- [20] K. Pruess and Y. Wu, *A Multiple-Porosity Method for Simulation of Naturally Fractured Petroleum Reservoirs*, Society of Petroleum Engineers Journal, 1988.
- [21] E. Lillestøl et al., *Generell fysikk for universiteter og høyskoler, Del 2: Varmelære, termodynamikk, elektrisitet, magnetisme, optikk og elektromagnetisme*, UiB/NTNU, 2001.