Modeling
Thermal Energy Storage Systems
with OpenFOAM

Candidato
Gian Maria Di Stefano
Matricola 725463

Relatore
Tommaso Lucchini

Anno Accademico 2013-2014
# Contents

## Introduction

1 Renewable energy from the Sun
   1.1 Overview .......................................................... 3
   1.2 Power plants using solar energy ................................. 4
   1.3 Energy storage ...................................................... 8
   1.4 Energy storage using sensible heat ............................. 10

2 Literature review ....................................................... 14
   2.1 Summary ............................................................ 26

3 TES modeling .............................................................. 28
   3.1 Free air zones ...................................................... 29
      3.1.1 Conservation laws: mass and momentum .................... 29
      3.1.2 Enthalpy conservation law .................................. 29
   3.2 The porous zone .................................................. 30
      3.2.1 Flow rate in porous zone .................................... 30
   3.3 Heat transfer between fluid and rocks .......................... 35
      3.3.1 Effective thermal conductivity of air ....................... 38
   3.4 Gravity effects ................................................... 39
   3.5 Turbulence modeling .............................................. 40

4 OpenFOAM .................................................................. 43
   4.1 The choice of OpenFOAM ......................................... 43
      4.1.1 Free software .................................................. 43
## List of Figures

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.1</td>
<td>Example of parabolic collectors</td>
<td>5</td>
</tr>
<tr>
<td>1.2</td>
<td>Example of disc collector</td>
<td>6</td>
</tr>
<tr>
<td>1.3</td>
<td>Example of a central receiver tower system</td>
<td>6</td>
</tr>
<tr>
<td>1.4</td>
<td>Example of linear Fresnel collectors</td>
<td>7</td>
</tr>
<tr>
<td>1.5</td>
<td>Storage methods</td>
<td>9</td>
</tr>
<tr>
<td>2.1</td>
<td>Porous media representation</td>
<td>14</td>
</tr>
<tr>
<td>2.2</td>
<td>Effective thermal conductivity calculated using different models using $\epsilon = 0.36$ (neglecting contact area) versus experimental results</td>
<td>25</td>
</tr>
<tr>
<td>2.3</td>
<td>Effective thermal conductivity calculated using different models (contact area included, radiation neglected) versus porosity variation</td>
<td>26</td>
</tr>
<tr>
<td>3.1</td>
<td>TES representation</td>
<td>28</td>
</tr>
<tr>
<td>3.2</td>
<td>Darcy, Darcy-Forchheimer, and Forchheimer flow</td>
<td>31</td>
</tr>
<tr>
<td>3.3</td>
<td>Laminar flow</td>
<td>32</td>
</tr>
<tr>
<td>3.4</td>
<td>Turbulent flow</td>
<td>32</td>
</tr>
<tr>
<td>3.5</td>
<td>Electrical analogy: series model</td>
<td>38</td>
</tr>
<tr>
<td>3.6</td>
<td>Representation of buoyancy due to gravity vector</td>
<td>40</td>
</tr>
<tr>
<td>4.1</td>
<td>PISO algorithm flowchart</td>
<td>47</td>
</tr>
<tr>
<td>4.2</td>
<td>SIMPLE algorithm flowchart</td>
<td>49</td>
</tr>
<tr>
<td>4.3</td>
<td>PIMPLE algorithm flowchart</td>
<td>50</td>
</tr>
<tr>
<td>5.1</td>
<td>rhoHeatPorousPimpleFoam flowchart</td>
<td>53</td>
</tr>
<tr>
<td>6.1</td>
<td>Solar Air Receiver TSA: storage representation</td>
<td>57</td>
</tr>
</tbody>
</table>
### LIST OF FIGURES

<table>
<thead>
<tr>
<th>Figure Reference</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.2</td>
<td>Solar Air Receiver TSA: results</td>
<td>58</td>
</tr>
<tr>
<td>6.3</td>
<td>Representation of TES: zones and boundary conditions</td>
<td>59</td>
</tr>
<tr>
<td>6.4</td>
<td>Computational mesh</td>
<td>60</td>
</tr>
<tr>
<td>6.5</td>
<td>Comparison between Meier and Fluent® results</td>
<td>63</td>
</tr>
<tr>
<td>7.1</td>
<td>Velocity magnitude inside TES at different times</td>
<td>67</td>
</tr>
<tr>
<td>7.2</td>
<td>Velocity contours at 3600s</td>
<td>68</td>
</tr>
<tr>
<td>7.3</td>
<td>Contours of velocity magnitude along axis - 3600 seconds</td>
<td>69</td>
</tr>
<tr>
<td>7.4</td>
<td>Contours of vertical velocity - 3600s</td>
<td>69</td>
</tr>
<tr>
<td>7.5</td>
<td>Contours of radial velocity - 3600s</td>
<td>70</td>
</tr>
<tr>
<td>7.6</td>
<td>Effective viscosity $\mu_{eff}$ at 3600s</td>
<td>71</td>
</tr>
<tr>
<td>7.7</td>
<td>Laminar thermal conductivity along axis 3600 seconds</td>
<td>72</td>
</tr>
<tr>
<td>7.8</td>
<td>Temperature inside TES at different time</td>
<td>73</td>
</tr>
<tr>
<td>7.9</td>
<td>Temperature inside TES at different time</td>
<td>74</td>
</tr>
<tr>
<td>7.10</td>
<td>Contours $T$ along axis - 1200s → 6000s - OpenFOAM</td>
<td>75</td>
</tr>
<tr>
<td>7.11</td>
<td>Contours of $T$ along axis - comparison between solvers</td>
<td>76</td>
</tr>
<tr>
<td>7.12</td>
<td>Contours of $T$ and $\rho$ - 3600s</td>
<td>76</td>
</tr>
</tbody>
</table>
List of Tables

1.1 Properties of some liquids suitable for energy storage [5] . . . . . . 11
1.2 Properties of some solids media suitable for energy storage [5] . . . 12

6.1 Solar Air Receiver TSA: geometrical characteristics . . . . . . . . . 56
6.2 Solar Air Receiver TSA: air and rocks characteristics . . . . . . . . 57
6.3 Boundary conditions . . . . . . . . . . . . . . . . . . . . . . . . . . 59
6.4 Mesh statistics . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 60
6.5 Fluent ® settings used to perform simulations of TES . . . . . . . . 62
6.6 Fluent ® settings used to perform simulations of TES . . . . . . . . 62

7.1 Open∇FOAM settings used to perform simulations of TES . . . . . 65
7.2 Open∇FOAM settings used to perform simulations of TES . . . . . 65
7.3 Open∇FOAM settings to perform TES simulations . . . . . . . . . . 66
### List of Symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$k_{\text{eff}}$</td>
<td>effective thermal conductivity</td>
<td>$\frac{W}{m \cdot K}$</td>
</tr>
<tr>
<td>$k_e^0$</td>
<td>effective thermal conductivity with motionless fluid</td>
<td>$\frac{W}{m \cdot K}$</td>
</tr>
<tr>
<td>$k_g$</td>
<td>thermal conductivity of fluid</td>
<td>$\frac{W}{m \cdot K}$</td>
</tr>
<tr>
<td>$\epsilon$</td>
<td>porosity</td>
<td>$\frac{1}{</td>
</tr>
<tr>
<td>$k_f$</td>
<td>thermal conductivity of fluid fraction</td>
<td>$\frac{W}{m \cdot K}$</td>
</tr>
<tr>
<td>$k_a$</td>
<td>thermal conductivity of solid fraction</td>
<td>$\frac{W}{m \cdot K}$</td>
</tr>
<tr>
<td>$D_p$</td>
<td>particle diameter</td>
<td>$m$</td>
</tr>
<tr>
<td>$k_e^g$</td>
<td>effective thermal conductivity</td>
<td>$\frac{W}{m \cdot K}$</td>
</tr>
<tr>
<td>$K$</td>
<td>ratio between $k_f$ and $k_s$</td>
<td>$\frac{</td>
</tr>
<tr>
<td>$\rho$</td>
<td>density</td>
<td>$\frac{kg}{m^3}$</td>
</tr>
<tr>
<td>$D_e$</td>
<td>equivalent spherical particle diameter</td>
<td>$m$</td>
</tr>
<tr>
<td>$V$</td>
<td>superficial mean fluid velocity through a rockbed, computed as air flow divided by the rockbed cross-sectional area</td>
<td>$\frac{m}{s}$</td>
</tr>
<tr>
<td>$h_v$</td>
<td>volumetric convective heat transfer coefficient</td>
<td>$\frac{W}{m^3 \cdot K}$</td>
</tr>
<tr>
<td>$G$</td>
<td>mass air flow rate per unit cross section</td>
<td>$\frac{m}{s}$</td>
</tr>
<tr>
<td>$d_p$</td>
<td>equivalent diameter of the rocks</td>
<td>$m$</td>
</tr>
<tr>
<td>$T$</td>
<td>temperature</td>
<td>$K$</td>
</tr>
<tr>
<td>$t$</td>
<td>time</td>
<td>$s$</td>
</tr>
<tr>
<td>$u_0$</td>
<td>superficial velocity</td>
<td>$\frac{m}{s}$</td>
</tr>
<tr>
<td>$u$</td>
<td>interstitial fluid velocity, calculated as $\frac{m}{\epsilon}$</td>
<td>$\frac{m}{s}$</td>
</tr>
<tr>
<td>$k_e^R$</td>
<td>radial effective heat transfer coefficient</td>
<td>$\frac{W}{m \cdot K}$</td>
</tr>
<tr>
<td>$k_e^x$</td>
<td>axial effective heat transfer coefficient</td>
<td>$\frac{W}{m \cdot K}$</td>
</tr>
<tr>
<td>$c$</td>
<td>heat capacity</td>
<td>$\frac{J}{kg \cdot K}$</td>
</tr>
<tr>
<td>$h$</td>
<td>interphase heat transfer coefficient</td>
<td>$\frac{W}{m^2 \cdot K}$</td>
</tr>
</tbody>
</table>
**LIST OF SYMBOLS**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>$a$</td>
<td>interphase surface area per unit volume: $a = \frac{6(1-\epsilon)}{D_s}$</td>
<td>$m^2$</td>
</tr>
<tr>
<td>$\chi$</td>
<td>solid fraction temperature</td>
<td>$K$</td>
</tr>
<tr>
<td>$\Phi$</td>
<td>wall temperature</td>
<td>$K$</td>
</tr>
<tr>
<td>$k_w$</td>
<td>wall thermal conductivity</td>
<td>$W/mK$</td>
</tr>
<tr>
<td>$h_w$</td>
<td>wall heat transfer coefficient</td>
<td>$W/m^2K$</td>
</tr>
<tr>
<td>$A_w^i$</td>
<td>inner wall surface area per unit length</td>
<td>$m^2$</td>
</tr>
<tr>
<td>$A_w^o$</td>
<td>outer surface area per unit length</td>
<td>$m^2$</td>
</tr>
<tr>
<td>$U_{inf}$</td>
<td>overall heat transfer coefficient from the wall</td>
<td>$W/mK$</td>
</tr>
<tr>
<td>$T_f$</td>
<td>temperature of the fluid fraction</td>
<td>$K$</td>
</tr>
<tr>
<td>$T_s$</td>
<td>temperature of the solid fraction</td>
<td>$K$</td>
</tr>
<tr>
<td>$\rho_s$</td>
<td>density of solid fraction</td>
<td>$kg/m^3$</td>
</tr>
<tr>
<td>$c_s$</td>
<td>heat capacity of solid fraction</td>
<td>$kJ/kgK$</td>
</tr>
<tr>
<td>$\rho_f$</td>
<td>density of fluid fraction</td>
<td>$kg/m^3$</td>
</tr>
<tr>
<td>$c_f$</td>
<td>heat capacity of fluid fraction</td>
<td>$kJ/kgK$</td>
</tr>
<tr>
<td>$\lambda_{inf}^e$</td>
<td>effective thermal conductivity</td>
<td>$kJ/msK$</td>
</tr>
<tr>
<td>$w$</td>
<td>artificial velocity</td>
<td>$m/s$</td>
</tr>
<tr>
<td>$\alpha_v$</td>
<td>heat transfer coefficient</td>
<td>$kJ/sm^2K$</td>
</tr>
<tr>
<td>$V_1$</td>
<td>first temperature wave velocity</td>
<td>$m/s$</td>
</tr>
<tr>
<td>$U_{su}$</td>
<td>superficial fluid velocity</td>
<td>$m/s$</td>
</tr>
<tr>
<td>$V_s$</td>
<td>first temperature wave velocity</td>
<td>$m/s$</td>
</tr>
<tr>
<td>$A_0$</td>
<td>total surface</td>
<td>$m^2$</td>
</tr>
<tr>
<td>$n$</td>
<td>number of spheres in control volume</td>
<td>$-$</td>
</tr>
<tr>
<td>$h$</td>
<td>convection coefficient</td>
<td>$W/m^2K$</td>
</tr>
<tr>
<td>$A_c$</td>
<td>cross-sectional area packed-bed</td>
<td>$m^2$</td>
</tr>
<tr>
<td>$k_{ge}$</td>
<td>effective thermal conductivity through the fluid and point contact</td>
<td>$W/mK$</td>
</tr>
<tr>
<td>$k_{ce}$</td>
<td>effective conductivity through contact area</td>
<td>$W/mK$</td>
</tr>
<tr>
<td>$k_{cr}$</td>
<td>effective conductivity due to radiation</td>
<td>$W/mK$</td>
</tr>
</tbody>
</table>
Sommaio

Ai giorni nostri, l’approvvigionamento dell’energia da fonti rinnovabili è visto come una valida risposta alla continua richiesta di energia, unita alla attenzione verso l’ambiente. Tra le fonti rinnovabili disponibili, l’energia solare è virtualmente infinita, ma di scarso utilizzo: attualmente i migliori pannelli fotovoltaici garantiscono un rendimento energetico massimo del circa 30%. Un modo più efficiente per utilizzare la radiazione solare è quella di trasferirla ad un fluido termovettore che ne permetta il trasporto e lo stoccaggio in dispositivi noti come TES: “Thermal Energy Storage”.

Il lavoro di tesi si focalizza su questo punto, proponendo uno strumento adeguato per il calcolo delle prestazioni del dispositivo di stoccaggio. Attualmente per simulare il funzionamento si fa uso di software commerciali. Lo scopo della tesi, quindi, è elaborare un solutore con software open-source Open\(\nabla\)FOAM che possa sostituire efficacemente i software commerciali ed essere utilizzato in seguito per eseguire altre prove su differenti impianti, geometrie e materiali.

Abstract

Nowadays, the supply of energy from renewable resources, can be a valid answer to the continuous demand for energy and environment protection. Among all the available renewable resources solar energy is virtually endless, and could be the best choice, even though not fully usable yet: currently, solar panels efficiency is less than 30%. A more efficient way to use solar radiation is to transfer it to a heat transfer fluid, that allows the energy storage in devices known as TES: “Thermal Energy Storage”.

Currently, commercial softwares are used to simulate TES functioning: thesis work deals about this issue: the goal of the project is to develop a solver with open-source software Open\(\nabla\)FOAM that can replicate commercial software outputs.
Introduzione

A causa del continuo incremento della richiesta di energia da parte del mondo industrializzato, da alcuni anni si è ormai imposto all’attenzione mondiale il problema di come ricavare energia da fonti alternative a quelle fossili da sempre in uso, essendo queste ultime destinate ad esaurirsi. L’attenzione si è così spostata su fonti di energia alternative e rinnovabili, quali l’acqua e la luce solare. soprattutto la radiazione solare, virtualmente infinita, sembra essere, se sfruttata a dovere, una delle soluzioni dalle migliori prospettive. D’altro canto, come ogni nuova tecnologia, essa è ad oggi ancora ben lontana dall’essere realmente competitiva rispetto ai combustibili fossili, e necessita di continui studi volti a migliorarne la resa. Il maggior problema che è necessario affrontare al fine di utilizzare l’energia fornita dal Sole, è dovuto all’intermittenza della radiazione solare, causata dal ciclo giorno/notte. Si pone dunque il problema di come immagazzinare questa forma di energia, in modo che possa essere utilizzabile anche nei momenti in cui manchi la luce solare diretta.

Un nuovo tipo di concentratore per l’utilizzo della radiazione solare sta recentemente concretizzando diverse idee innovative, atte a contenere i costi dell’impianto, pur garantendo un’ottima resa energetica. In questo caso la radiazione solare viene riflesse a raccolta, in modo da scalare un fluido termovettore, il quale ha la funzione di immagazzinare energia sotto forma di calore. Tale calore viene poi utilizzato per generare vapore dall’acqua, che, espanso in turbina, produce elettricità. Come accennato, condizioni metereologiche avverse possono rendere nulla la produzione di energia elettrica in taluni frangenti, o durante la notte.

Uno dei modi più semplici ed efficienti per immagazzinare l’energia del fluido termovettore consiste nell’utilizzare la capacità termica dei materiali solidi, stoccati in appositi serbatoi. Il fluido, caldo, viene pompato in tali serbatoi, ed
attraversando il materiale stoccante trasferisce l'energia interna accumulata sotto forma di calore. In questo modo il materiale di stocaggio aumenta la propria energia interna, potendola rilasciare, in caso di necessità, invertendo il ciclo. Ad esempio, durante la notte, pompando il fluido termovettore freddo, attraverso il serbatoio di calore caldo, si riesce ad ottenere energia che possa poi essere utilizzata per convertirla mediante apposite turbine in energia elettrica.

Il presente lavoro di tesi si concentra sulla simulazione numerica del ciclo di carico e scarico nel serbatoio, in modo da poter valutare, senza dover produrre appositi impianti e senza usare test reali, soluzioni preliminari per il corretto dimensionamento del serbatoio e per la scelta dei materiali in base ai tempi di carica e scarica desiderati. In un primo momento verrà modellato numericamente il comportamento del fluido all'interno del serbatoio, capendo quali sono i principi termodinamici sui quali si basa lo scambio di calore sopra citato. Tale problema fu affrontato per la prima volta nel 1929, a opera di Schumann ed in seguito tale problema fu indagato in modo profondo ed estensivo, arrivando a proporre diversi tipi di approcci e le relative soluzioni numeriche. Tra le formulazioni proposte, verrà scelta quella che maggiormente si presterà alla modellazione numerica necessaria per ottenere un solutore numerico efficace e che fornisca risultati affidabili. In particolare, tale solutore verrà scritto basandosi sul software Open\textregistered\textsc{FOAM}. Tale software è una suite per risolvere equazioni alle derivate parziali, in particolare modo utilizzato per risolvere campi di moto di fluidi. Nello specifico, poiché tale software non presenta una solutore per lo scambio di calore in mezzi porosi (quale è di fatto il serbatoio in questione), sarà necessario scrivere da zero un solutore appropriato traducendone la formulazione precedentemente ottenuta, in un linguaggio adatto al compilatore.

I risultati ottenuti verranno confrontati con un caso test disponibile in letteratura, e con i risultati ottenuti con un software di simulazione fluidodinamica commerciale, Fluent®, onde poter comparare, oltre ai risultati, pregi e difetti dei due diversi software e valutarne l'efficacia.
Introduction

In these years, a new awareness relative to energy supply was globally formed: due to the increasing energy demand by industries and domestic needs, and due to depletion of oil reserves, progressive replacement of fossil fuels with alternative energy sources is now becoming fundamental. Use the abundant and infinite energy of the Sun seems to be one of the very promising solutions. However, like every other new and developing technology, there are many challenges to overcome until an economically competitive solution is found: one of the most drawbacks to the use of solar energy is, in fact, its cost. Another important drawback of this form of energy is also its intermittent nature: in order to be able to continuously produce energy, this gap has to be closed by using adequate storing mechanisms able to store the energy produced during the day, under direct solar irradiation, to use it during the night hours.

One of the simplest, most efficient and economic way to store the energy provided by the Sun is to use it to increase the internal energy of a thermovector media: this media can be either used to operate in a Rankine cycle plant, or to store the collected energy, by capacity of rocks to store energy released from hot air, and save it in their mass. In order to make this operations, a tank filled by rocks is needed. This plant is called TES: thermal energy storage. In this application, during the day the phase called charging happens: hot air flows through top to bottom of the rock-filled tank, releasing heat to rocks. The stored energy is then extracted during the night by reversing the flow, in the phase called discharging: air at ambient temperature is pumped in the storage from the bottom, is heated up by the hot stones leaving the tank from the top to be used in a steam engine.

The idea of storing the energy in a packed bed of rocks is not new. Schumann, in 1929 [31], was one of the first studying analytically the heat transfer between
the fluid and the solid and, according to Singh and Furnas [13], was probably one of the first conducting experimental studies on the concept of packed bed storage in 1930.

Since then, this idea subject has been investigated extensively and different analytical approaches and experimental setups have been introduced. However, for various reasons, first the lack of incentive in investing in solar energy systems, due to cheap fuel fossil prices, the packed bed storage concept has not found yet its place in the industry.

With modern technological methods and computers, is now possible, with adequate analytical model, to solve complex simulation, involving complex geometries, different materials and relative phisical characieristics, different velocity of fluid. Using this new methods this technology can be investigated, in order to evaluate its profitability compared with other systems used to obtain energy.

Thesis goal is, in particular, the "creation" of a numerical solver to correctly simulate how a thermal energy storage system works. The solutoor was created using the open-source software called Open∇FOAM. This software is a numerical suite born to solve partial differential equation, with focus on fluid fields. Open∇FOAM [21], during the thesis work period, did not offer any solutoor to simulate the heat exchange between air and rocks inside a thermal energy storage, so a properly solver was created. In order to choose a proper numerical model to develop and compile the solver (in order to use it in a 3-D model), different numerical representations, found in literature and developed by authors during past years, were investigated and compared.

Computed results were compared to what is available in literature, in order to validate them, and then were also compared with a test obtained using a commercial numerical tools, Fluent © [4], using identical set up in both the tests.
Chapter 1

Renewable energy from the Sun

1.1 Overview

This chapter’s purpose is to provide an overview of the technology available nowadays. By searching for informations it can be noted that most of the available documentation goes back to the 80s: there is a lack of more recent informations. An explanation for this fact can be found considering the historical and political context before of years 80s, dominated by two serious energy crises, in 1973 and in 1979. In both situations the price of crude oil reached very high levels causing great inconvenience to supply all the Western countries.

These crises pushed many countries towards research and development of new renewable energy alternatives to oil: atomic energy, natural gas, solar energy, wind energy, and so on. In 80s considerable progresses were made, but there was a formerly stop because of the end of the second crises: the price of fossil fuels became lowered again, making inconvenient spending time and resources studying alternative forms of energy production.

Nowadays the price of fossil fuels is steadily increasing, and all the world has a new and renewed environmental awareness. Collective conscience is now adult: saving ambient is now a real problem, and it is necessary to act to make a more green world. Another question, probably the most important, that pushes to investigate a new way to produce energy, is that fossil resources are going to be depleted in the future, hence it is now necessary to find new forms of energy.
1.2. POWER PLANTS USING SOLAR ENERGY

After this brief historical reference, the attention will be focused on the description of one type of generation of renewable energy: the solar power. Will be explained how the solar power plants work, and then will be justified the inclusions, in this type of plants, of a system of storage.

1.2 Power plants using solar energy

Solar energy power plants can be splitted in two conceptual blocks:

**Concentrated photovoltaic - CPV** these plants are made from many panels consisting of photovoltaic cells which are capable of converting directly the solar energy absorbed into electrical energy.

**Concentrated solar power - CSP** these other plants, instead, are made by large surfaces covered with mirrors which, by exploiting the refraction of the solar light, convey it in a single point, called fire, where the boiler is located. This concentration of energy heats the water contained in the boiler, which reaches a temperature high enough to insert it in a thermodynamic cycle for the production of electricity.

Focusing on CSP type of power plants, the significant role of mirrors can be easily understood: they redirect solar rays into the fire, in order to give energy from sun to the liquid used. Mirrors are divided in four main categories, due to their arrangement with respect to the receiver and depending on their geometry.

**Parabolic through collectors** to capture solar energy, these plants use large distributions of linear parabolic collectors, able to rotate along their longitudinal axis to follow the solar rotation, maximizing the efficiency of these plants, called solar field. Their task is to reflect and concentrate solar energy on an insulated pipe, called receiver, placed at the top. Inside the receiver a heat transfer fluid flows, generally synthetic oil, which, along the entire length of the duct exposed to the radiation, greatly increases its internal energy reaching a temperature of about 400 °C. Once reached the desired temperature, the fluid flows through an heat exchanger, in which it releases the thermal energy stored in the water, transforming
1.2. POWER PLANTS USING SOLAR ENERGY

it into steam, which is then expanded in the turbine. The electrical energy is then produced from turbine using a thermodynamic cycle such as the Rankine cycle or the Brayton cycle \cite{36}. The image 1.1 shows a view of the solar field in a power plant: it can be seen the vast surface needed to meet the energy requirements.

![Figure 1.1: Example of parabolic collectors](image)

Parabolic collectors point or disc in this type of plant reflective panels have a parabolic shape and can rotate around two orthogonal axes to follow the solar trajectory, as visible in figure 1.2. Solar radiation is concentrated at the focal point located in the center of each reflector. Obtained heat is then transferred to a fluid and used immediately to produce electrical or mechanical energy, using the aid of a motor placed on the top of the receiver. Currently, in industrial applications motors with Stirling or Bryton cycle are used.

Systems with central receiver tower system the most important difference in this type of plants is that solar rays are reflected in a single common point, such as previous seen fire, using plane mirrors (called heliostats) able to rotate around two orthogonal axes, as can be seen in figure 1.3.
1.2. POWER PLANTS USING SOLAR ENERGY

![Figure 1.2: Example of disc collector](image1.png)

![Figure 1.3: Example of a central receiver tower system](image2.png)

**Linear Fresnel collectors** this type of collector is the newer and, so far, is the only one that can be adapted for the production of electricity in small companies. Similarly to *parabolic through collector*, this type of plant is made by several rows of mirrors reflecting sunlight to a receiver pipe. The main difference is due to the type of mirrors: Fresnel collectors use Fresnel lens [20], that are more efficient
than normal mirror, permitting a better concentration of solar irradiation and so, higher temperature level. The achievement of higher temperature permits to use directly water as transfer fluid, that become steam, ready to get into the turbine. An example can be seen in figure 1.4.

![Example of linear Fresnel collectors](image)

Figure 1.4: Example of linear Fresnel collectors

The main advantages of the collector are:

- Eco-friendly: dangerous or toxic materials and substances are reduced to a minimum, and the consumption of grey energy is also been minimized.

- Efficient: they provide a better ratio of electricity produced to solar power captured, competitive systems.

- Flexible: thanks to the long-term energy storage system, it is possible to supply electricity on demand (from production during peak times to continuous production 24 hours a day).

The major disadvantage of these plants is that a good amount of energy production occurs only during the day and with favourable weather conditions. The passage of a cloudy disturbance above the solar field can cause a sudden decrease in the production of electricity, but the night hours are the most critical, because of the lack of the primary source of energy: solar irradiation.

To overcome these drawbacks, generally an auxiliary boiler fueled by gas or fossil fuels is used in conjunction with the power plant: this solution permits to respond quickly to any cloud cover without affecting the normal operation of the turbines, but feed the power plant during this period with the auxiliary boiler entails enormous costs of fuel, that can make pointless the advantage of using
solar irradiation during the day. To solve this problem the idea is to incorporate an energy storage disposal, that can be able to supply the system during periods of absence of the Sun.

1.3 Energy storage

As seen, a big challenge to make solar plants competitive, is to ensure a continuous supply of hot air, in order to maintain plants active also during

- alternation between day and night;
- passing cloudy phenomena;
- seasonal variations in irradiance due to the annual cycle;
- unfavourable weather conditions;

Many of the existing and emerging technologies can potentially be adapted to convert solar energy into other forms, to store it. Main ones are:

- electrochemical storage batteries;
- chemical conversion;
- conversion to mechanical energy;
- storage of thermal energy by exploiting the physical properties of substances such as: the sensible heat, latent heat and the reversibility of thermochemical reactions.

Heat transfer from thermo-vector fluid to storage disposal takes place by means of forced convection: accumulator media or fluid circulates itself through the heat exchanger. This type of storage can be further divided into direct, if the fluid is used as the storage media, or indirect, in case there is another media that provides to the storage of thermal energy.
1.3. ENERGY STORAGE

Direct storage  this type of storage is obtained using two different schemes of construction: individual container (code-tank) or double container (two-tank).

In case of single container, energy storage takes the name of thermocline storage: it allows to obtain a good natural stratification of the temperature inside it, due to differences in temperature of the tank, which brings into a difference in density of the heat transfer fluid between the hot and cold part. Due to this stratification the liquid is at a higher temperature in the upper part. Filling the tank with a second storage medium such as rock, sand or steel helps to maintain a good stratification. This particular type of storage will be addressed in detail later.

Double container system required a reservoir for the hot heat transfer fluid (output from the solar field) located near the steam generator, and another reservoir for the cold fluid ready to be placed again in the receiver of the solar field. Between the two tanks a steam generator is placed: it is an heat exchanger which allows the thermal exchange between the solar energy carried by the fluid and the water used for the production of electricity.
1.4. ENERGY STORAGE USING SENSIBLE HEAT

**Passive storage**  in passive system the heat transfer fluid transports, through an heat exchanger, the stored energy to the means of storage in the tank during the loading phase. Note that the storage, solid or liquid, does not undergo any displacement, remaining fixed.

A disadvantage of this arrangement is that the fluid coming from the solar field does not come directly in contact with the medium, because storage media uses another fluid interposed between the two. This implies that the maximum temperature in the tank is less than that of the fluid due to the losses and efficiency of the exchanger.

### 1.4 Energy storage using sensible heat

Once introduced the concept of energy storage, it is necessary to focus on the physical phenomenon of heat storage: thermal energy can be stored using the sensitive heat (showed by a change of temperature) of a substance that changes its internal energy. The amount of sensible heat gained or transferred to a material can be calculated as:

\[
Q = m \cdot \int_{T_1}^{T_2} c_p \cdot dT
\]

(1.1)

where \(Q\) is the total amount of heat exchanged and \(m\) indicates the mass of material affected by the thermal variation and \(c_p\) indicates the average specific heat between the temperature \(T_1\) and the \(T_2\). Other properties such as density, conductivity and thermal diffusivity play an important role for the purposes of the dimensioning of this type of storage. Store the thermal energy using the sensible heat offers important advantages: proven reliability and ease of monitoring of charging and discharging phases. As mentioned above, storage media can be liquid or solid. The following paragraphs will discuss the characteristics of each one.

**Liquid media**

**Water** For temperatures ranging between 0°C a 100°C, water is certainly the main fluid. Water is economic, abundant, high available, non-toxic, non-flammable, has excellent thermal properties and can be easily pumped. How-
1.4. ENERGY STORAGE USING SENSIBLE HEAT

However, the main drawback is high corrosiveness: this property forces to use stainless materials, leading to an increase of plants costs.

**Organics fluids** Organics fluids are used in power plants based upon Rankine cycle. They allow a storage temperature, in standard pressure condition, much greater than water, but carry high costs for plants, because of their inflammability. These fluids are subject to natural degradation, so a cyclic change is needed, with requirement to dispose the fluid become unusable.

**Molten salts** They are the only ones that can allow very high temperatures, thanks to the great capacity for receiving and storing heat. In plants using salts temperatures close to 900°C can be reached, but so high temperature implies fairly complicated design features.

**Liquid metal** The liquid sodium can reach high temperatures of storage using special attention. It is very expensive and, because of its high thermal conductivity, does not allow a good temperature stratification in the tank.

<table>
<thead>
<tr>
<th>Storage media</th>
<th>Fluid type</th>
<th>Temperature range [°C]</th>
<th>Density [kg/m³]</th>
<th>Specific heat [J/kgK]</th>
<th>Thermal cond. [W/mK]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Water</td>
<td>-</td>
<td>0 to 100</td>
<td>1000</td>
<td>4190</td>
<td>0.63 at 38°C</td>
</tr>
<tr>
<td>Caloria HT43</td>
<td>oil</td>
<td>-10 to 315</td>
<td>-</td>
<td>2300</td>
<td>-</td>
</tr>
<tr>
<td>Dowtherms</td>
<td>oil</td>
<td>12 to 260</td>
<td>867</td>
<td>2200</td>
<td>0.112 at 260°C</td>
</tr>
<tr>
<td>Therminol 55</td>
<td>oil</td>
<td>-18 to 325</td>
<td>-</td>
<td>2400</td>
<td>-</td>
</tr>
<tr>
<td>Therminol 66</td>
<td>oil</td>
<td>-9 to 343</td>
<td>750</td>
<td>2100</td>
<td>0.106 at 343°C</td>
</tr>
<tr>
<td>Ethylene Glycol</td>
<td>-</td>
<td>-</td>
<td>1116</td>
<td>2382</td>
<td>0.249 at 20°C</td>
</tr>
<tr>
<td>Hitec</td>
<td>molten salt</td>
<td>141 to 540</td>
<td>1680</td>
<td>1560</td>
<td>0.61</td>
</tr>
<tr>
<td>Engine oil</td>
<td>oil</td>
<td>up to 160</td>
<td>888</td>
<td>1880</td>
<td>0.145</td>
</tr>
<tr>
<td>Draw salt</td>
<td>molten salt</td>
<td>220 to 540</td>
<td>1733</td>
<td>1550</td>
<td>0.57</td>
</tr>
<tr>
<td>Lithium</td>
<td>liquid salt</td>
<td>180 to 1300</td>
<td>510</td>
<td>4190</td>
<td>38.1</td>
</tr>
<tr>
<td>Sodium</td>
<td>liquid salt</td>
<td>100 to 760</td>
<td>960</td>
<td>1300</td>
<td>67.5</td>
</tr>
<tr>
<td>Ethanol</td>
<td>organic liquid</td>
<td>up to 78</td>
<td>790</td>
<td>2400</td>
<td>-</td>
</tr>
<tr>
<td>Propanol</td>
<td>do</td>
<td>up to 97</td>
<td>800</td>
<td>2500</td>
<td>-</td>
</tr>
<tr>
<td>Butanol</td>
<td>do</td>
<td>up to 118</td>
<td>809</td>
<td>2400</td>
<td>-</td>
</tr>
<tr>
<td>Isobuthanol</td>
<td>do</td>
<td>up to 100</td>
<td>808</td>
<td>3000</td>
<td>-</td>
</tr>
</tbody>
</table>

Table 1.1: Properties of some liquids suitable for energy storage [5]
1.4. ENERGY STORAGE USING SENSIBLE HEAT

Solid media

The advantages arising from the use of solid media instead of fluid media are: low cost, wide range of operating temperature and simpler storing structure.

<table>
<thead>
<tr>
<th>Storage media</th>
<th>Density $\frac{kg}{m^3}$</th>
<th>Specific heat $\frac{J}{kgK}$</th>
<th>Heat capacity $\frac{10^6 J}{m^3K}$</th>
<th>Thermal diff $\frac{m^2}{s}$</th>
<th>Thermal cond. $\frac{W}{mK}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Aluminium</td>
<td>2707</td>
<td>896</td>
<td>2.4255</td>
<td>84'100</td>
<td>204 at 20°C</td>
</tr>
<tr>
<td>Aluminium oxide</td>
<td>3900</td>
<td>840</td>
<td>3.276</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Aluminium sulphate</td>
<td>2710</td>
<td>750</td>
<td>2.0325</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Brick</td>
<td>1698</td>
<td>840</td>
<td>1.4263</td>
<td>0.484</td>
<td>0.69 at 29°C</td>
</tr>
<tr>
<td>Cement</td>
<td>2240</td>
<td>1130</td>
<td>2.531</td>
<td>0.356 - 0.514</td>
<td>0.9 - 1.3</td>
</tr>
<tr>
<td>Cast iron</td>
<td>7900</td>
<td>837</td>
<td>6.6123</td>
<td>4'431</td>
<td>29.3</td>
</tr>
<tr>
<td>Pure iron</td>
<td>7897</td>
<td>452</td>
<td>3.5694</td>
<td>20'450</td>
<td>73.0 at 20°C</td>
</tr>
<tr>
<td>Calcium chloride</td>
<td>2510</td>
<td>670</td>
<td>1.6817</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Copper</td>
<td>8954</td>
<td>383</td>
<td>3.4294</td>
<td>112'300</td>
<td>385 at 20°C</td>
</tr>
<tr>
<td>Wet soil</td>
<td>1700</td>
<td>2093</td>
<td>3.5581</td>
<td>0.750</td>
<td>02:51</td>
</tr>
<tr>
<td>Dry soil</td>
<td>1260</td>
<td>795</td>
<td>1.0017</td>
<td>0.250</td>
<td>00:25</td>
</tr>
<tr>
<td>Potassium chloride</td>
<td>1980</td>
<td>670</td>
<td>1.3266</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Potassium sulfate</td>
<td>2660</td>
<td>920</td>
<td>2.4472</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Sodium carbonate</td>
<td>2510</td>
<td>1090</td>
<td>2.7359</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Granite stone</td>
<td>2640</td>
<td>820</td>
<td>2.1648</td>
<td>0.799 - 1.840</td>
<td>1.73 - 3.98</td>
</tr>
<tr>
<td>Limestone</td>
<td>2500</td>
<td>900</td>
<td>2.25</td>
<td>0.560 - 0.591</td>
<td>1.26 - 1.33</td>
</tr>
<tr>
<td>Marble stone</td>
<td>2600</td>
<td>800</td>
<td>2.08</td>
<td>0.995 - 1.413</td>
<td>2.07 - 2.94</td>
</tr>
<tr>
<td>Sandstone</td>
<td>2200</td>
<td>710</td>
<td>1.562</td>
<td>1'172</td>
<td>1.830</td>
</tr>
</tbody>
</table>

Table 1.2: Properties of some solids media suitable for energy storage [5]

Between all the available media, mainly two are used, because of their good price-performance ratio:

Sand and natural rock For seasonal storage systems, these media are a great alternative to water. They are practically free and require not complicated containment structures. The use of earth and rock is a solution present in many thermal energy storage systems in both industrial and private installations.
Sand and treated rock, also known as packed-bed For storage systems that are in function every day, the use of a bed of treated stone as accumulator is the one offering the most interesting features.

Advantages of the use of a bed of rocks are:

- ease of retrieval of the material;
- rocks allow to obtain a good natural stratification of temperature.
- non-toxicity and non-flammability;
- high storage temperatures obtainable;
- no problems related to corrosion;

parallel to the advantages, are also present some disadvantages, which can not be overlooked

- bed of rocks requires a large storage volume;
- there are significant pressure drops;
- simultaneous loading and unloading are not allowed.

Nevertheless, advantages overcome disadvantages, and rock bed can be a promising solution to store energy. Some authors during years investigates packed bed system searching for correlation between rocks size and performances. Torab and Beasley [16] state that particle diameters should be larger than $12.7\text{mm}$ but less than one thirtieth of the bed diameter. Larger particle sizes result in a lower pressure drop, but also lower the volumetric heat transfer coefficient in the bed. Sanderson and Cunningham [32], according to Torab and Beasley, stated that the equivalent diameter of particles in a packed bed should be greater than $13\text{mm}$ to avoid excessive pressure losses and high pumping power requirements.

Small particles result in better stratification, which brings to a steeper temperature wave [32]. Smaller particles allow less axial thermal dispersion through the bed – which always occurs to some extent – than larger particles do.
Chapter 2

Literary review

The idea of accumulating heat in rocks, using hot air like thermo vector fluid, as seen, dates back to the 30s of the last century. During years many scientists confronted the problem of the numerical modeling of a TES, developing different numerical models to represent complex phenomena that happen inside the tank when an hot fluid flow through a packed bed.

First of all, it must be provided a univoque definition of *porous zone*. The porosity of a material, or void fraction, is defined as the ratio $\epsilon$ of the void volume $V_f$ to the total volume $V_o$ of a sample volume including both void and solid, like depicted in figure 2.1.

![Porous media representation](image)

Figure 2.1: Porous media representation
By definition, porosity value of a medium can be easily calculated. It is often represented by greek letter $\epsilon$ and it is bounded by value of 0 and 1: in case it is equal to zero, it implies solid material, vice versa, in case of 1 value, it implies empty space.

$$\epsilon = \frac{V_f}{V_o} = \frac{V_f}{V_f + V_s} = 1 + \frac{V_f}{V_s}$$ (2.1)

Usually porosity will vary throughout the medium: several authors proposed different methods for measuring the porosity of samples. The simplest is to fill a container with rock, putting water into the container until it fill the void space between the rock. Measuring total volume of water inside the tank, the fluid fraction of the total volume is known, and so on porosity value [15]. Other methods were proposed in by Dullien [12] and Kaviany [19]: authors listed some other methods that can be used to determine porosity.

Once porosity is defined, different articles made by authors in years will be evaluated. The core idea of the following formulations is to model heat exchange between fluid and solid in porous zone by an equivalent, also called effective, value of thermal conductivity. This term will be indicated as $k_{\text{eff}}$ and will be calculated taking into account conductivity of fluid $k_f$ and solid phase $k_s$, or the ratio between cited quantities $K = \frac{k_f}{k_s}$. Here are shown, in chronological order, some formulations proposed by authors concerning heat transfer mechanism inside packed bed.

**Schumman model**

Schumman was probably the first to study heat transfer between air and rocks inside a tank, in 1929 [31]. His numerical model considers fluid and solid phase acting two separated roles: fluid phase has the convection role, while solid phase is modeled as static. The model has an important limitation: in fact it is a one-dimensional (1-D) model. Equation that describes fluid temperature is time depending, with a convective term that describes the transport along the main direction of the velocity vector. Thermal capacity is multiplied by $\epsilon$ in order to consider the porosity and the effective volume of air. Heat lost by air is absorbed by rocks, using the $h_s$ factor, and by heat exchange with the storage walls, indicated by the term $U_w$. 

15
Fluid phase

\[
\epsilon \rho_f c_f \left( \frac{\partial T_f}{\partial t} + V_\infty \frac{\partial T_f}{\partial x} \right) = h_s A_s (T_f - T_f) - U_w a_w (T_f - T_0)
\]  

(2.2)

Vice versa, \(1 - \epsilon\) value, that determinates the solid fraction, multiplies the evolutive equation for the temperature inside the rock bed, built without considering heat transfer between each rocks (this is a 1-D model), and without convective term due to fixity of matrix.

Solid phase:

\[
(1 - \epsilon) \rho_s c_s \frac{\partial T_s}{\partial t} = h_{fs} A_{fs} (T_f - T_s)
\]  

(2.3)

Yagi and Kunii model

Yagi and Kunii, in 1957 [30], obtained theoretical formulas for effective thermal conductivities \(k_{\text{eff}}\) in packed beds, by searching for correlation between \(k_{\text{eff}}\) value and packing characteristics and temperature inside the tank. According to this paper, effective thermal conductivity can be separated into two terms: one independent from fluid flow, the other dependent on the lateral mixing of the fluid in the packed beds. Focusing on heat transfer phenomena independent of fluid flow, the authors identified these different contributions:

1. thermal conduction through solid;
2. thermal conduction through the contact surfaces of two packings;
3. radiant heat transfer between the surfaces of two packings (in case of gas);
4. radiant heat transfer between the neighboring voids.

Vice versa, considering heat transfer dependent on fluid flow there are:

5. thermal conduction through the fluid film near the contact surface of two packings;
6. heat transfer by convection, solid-fluid-solid;
7. heat transfer by lateral mixing of fluid.

When Reynolds number is small, the boundary layer around the solid packings is thick; therefore mechanisms numbered 1, 3, 4 and 5 are predominant. However, in the case of a large Reynolds number, process 7 controls the heat flux in any packed bed, and therefore the effect of mechanism 6 on the total rate of heat flow is slight. By these assumptions, authors assumed that:

- thermal conduction though the thin film of fluid near the contact surfaces is not affected by fluid flow;
- the convensional heat transfer mechanism solid-fluid-solid is less important than the other mechanisms and can be safely neglected.

Authors consider that radiant heat transfer fraction is negligible if the flow temperature is less then 400 °C. Considering this assumption, effective thermal conductivity of packed bed can be modeled as

\[
\frac{k^0_e}{k_g} = \beta \frac{1 - \epsilon}{\frac{k_a}{k_g} + \phi}
\]

Kunii and Smith

In 1960, Kunii and Smith \[8\] performed a similar study, but limited to the case of effective conductivity in a packed bed filled by stagnant flow. This article introduces an interesting relationship between packed bed and consolidated porous rocks. Authors, similarly to previous mentioned study, distinguished heat transfer mechanisms by two different ways:

1. heat transfer through the fluid in the void space by conduction;
2. heat transfer by radiation between adjacent voids.

Considering heat transfer through the solid phase, authors enumerate different mechanisms, like

3. heat transfer through the contact surface of the solid particles;
4. conduction through the stagnant fluid near the contact surface;
5. radiation between surfaces of solid;

6. conduction through the solid phase.

Inside TES, the two types of mechanisms are in parallel with each other, instead of mechanism 4 that is in series with the combined result of parallel mechanism 1, 2 and 3. According to these basis, authors proposed this formulation to calculate the value of the effective conductivity

\[
\frac{k_0^0}{k_g} = \epsilon + \frac{\beta(1 - \epsilon)}{\phi + \gamma \left(\frac{k_g}{k_s}\right)}
\]

Above formulation is valid if temperature is less than 485 °C. If temperature is greater than 485 °C, radiant contribution can not be neglected, so it must to be used a more detailed formulation:

\[
\frac{k_0^0}{k_g} = \epsilon \left[ 1 + \beta \frac{h_{rs} D_p}{k_g} \right] + \frac{\beta(1 - \epsilon)}{\frac{1}{\phi + \frac{\gamma}{D_p} (h_p + h_{rs})}} + \gamma \left(\frac{k_g}{k_s}\right)
\]

According to the authors, quantities \( \beta, \gamma \) and \( \phi \) must be calculated. For close packing of spheres the average value of \( \beta \) can be calculated as follows:

\[
\beta = \frac{1}{D_p} \frac{1}{3} \left[ \left(\frac{2}{3}\right)^{1/2} + 1 + \left(\frac{3}{2}\right)^{1/2} \right] D_p = 0.895
\]

For most of applications, the value of \( \beta \) calculated will range from 0.9 to 1; the value of \( \gamma \) depends upon \( l_s \): it will be assumed to be the length of a cylinder having the same volume as the spherical particle, that is

\[
\gamma = \frac{L_s}{D_p} \quad l_s = \frac{\pi}{6} \frac{D_p^2}{\frac{4}{3} D_p} = \frac{2}{3} D_p \quad \gamma = \frac{l_s}{D_p} = \frac{2}{3}
\]

The same authors, in 1961 [9], focused their attention on the calculation of the radiative contribution mentioned above, but considered negligible by other authors. Radiant contribution can be evaluated by calculating the Nusselt number and the surface area of every part of packed bed: this approach is evidently impracticable in
real case of big dimension tank. The obtained formulation of $k_e$ given by authors is therefore dependent by number and dimensions of particle: in fact this idea is not applicable because of excessive complication and therefore, computing costs.

$$\frac{k_e}{k_f} = \epsilon + \frac{\beta(1-\epsilon)}{\psi_t + \frac{2}{\kappa}}$$

where coefficient $\beta$ and $\gamma$ are evaluated empirically.

**Hashin and Shtrikman**

Hashin and Shtrikman, in 1962 [37] proposed two formulations for evaluate $k_{eff}$ value: first must be considered as the upper bound, and second as the lower bound.

**Upper bound** This equation is valid if $k_s/k_f > 1$, and represent geometry using the $\epsilon$ parameter only

$$\frac{k_{eff}}{k_f} = k_s \left[ 1 + \frac{3\epsilon(1-k_s/k_f)}{(1-\epsilon) + k_s/k_f(2+\epsilon)} \right]$$

(2.10)

**Lower bound** For lower bound are valid the same restriction used for upper bound.

$$\frac{k_{eff}}{k_f} = \left[ 1 + \frac{3(1-\epsilon)(k_s/k_f - 1)}{3 + \epsilon(k_s/k_f - 1)} \right]$$

(2.11)

**Zehner and Schlunder**

Zehner and Schlunder in 1970 [26] proposed their formulation for effective thermal conductivity:

$$\frac{k_{eff}}{k_f} = \left[ 1 - \sqrt{1-\epsilon} \right] + \frac{2\sqrt{1-\epsilon}}{1-k^{-1}B} \cdot \left[ \frac{(1-k^{-1})B}{(1-k^{-1}B)^2} \log(\frac{1}{k^{-1}B}) - \frac{B+1}{2} - \frac{B-1}{1-k^{-1}N} \right]$$

(2.12)

where the deformation parameter $B$ is related to the porosity by

$$B = 1.25 \left( \frac{1-\epsilon}{\epsilon} \right)^{10}$$

(2.13)
However, Hsu et al. in 1994 [6] found that

$$B = 1.364 \frac{1 - \epsilon}{\epsilon^{1.055}}$$

(2.14)

results in more accurate prediction.

**Chandra and Willis**

Chandra and Willis, in 1981 [24], focused their attention on pressure drop in packed-bed. They proposed the following equation to predict pressure drop of air through a rockbed:

$$\frac{\delta p\rho D_e^3}{\mu^2} = \epsilon^\alpha \left(185 \left(\frac{\rho V D_e}{\mu}\right) + 1.7 \left(\frac{\rho V D_e}{\mu}\right)^2\right)$$

(2.15)

with some variations, depending on $\epsilon$, that is *porosity fraction* value:

- in case of $0.33 < \epsilon < 0.38$ and $1 < \frac{\rho V D_e}{\mu} < 257$, the exponent $\alpha$ must be changed to 2.5;
- in case of $0.38 < \epsilon < 0.46$ and $100 < \frac{\rho V D_e}{\mu} < 1000$ the exponent $\alpha$ must be changed to 2.5;
- in case of $0.33 < \epsilon < 0.46$ and $1 < \frac{\rho V D_e}{\mu} < 1000$ the exponent $\alpha$ must be changed to 2.6.

More generally, the pressure drop can be evaluated as follows:

$$\frac{\delta p\rho D_e^3}{\mu^2} = f \left(\frac{\rho V D_e}{\mu}, \epsilon\right)$$

(2.16)

Note that this formulation does not contain any explicit dependence of the pressure drop on the air and rock temperatures, but it is clear that pressure drop depends on Reynolds number and porosity.
Beasley and Clark

Beasley and Clark, in 1984 [11], went forward with their studies: they considered a 2-D model instead of previous models that were 1-D models, but taking into account the influence of both radial thermal dispersion and radial variations in the inlet fluid temperature. Unfortunately, they developed an equation model made upon radial coordinate system. The following axisymmetric equations govern the temperature distribution in a 2-D packed bed where the void fraction, $\epsilon$, velocity, $u$, and transport coefficients vary spatially with radial location.

**fluid phase**

$$\frac{\partial T}{\partial t} + u \frac{\partial T}{\partial x} = \frac{k_x}{\rho c d e} \frac{\partial^2 T}{\partial x^2} + \frac{k_R}{\rho c d e} \left( \frac{1}{R} \frac{\partial T}{\partial R} + \frac{\partial^2 T}{\partial R^2} \right) + \frac{h a}{\rho c} \left( \chi - T \right)$$  \hspace{1cm} (2.17)

**solid phase**

$$\rho_s c_s (1 - \epsilon) \frac{\partial \chi}{\partial T} = h a (T - \chi)$$  \hspace{1cm} (2.18)

**wall**

$$\rho_w c_w \frac{\partial \Psi}{\partial t} = k_w \frac{\partial^2 \phi}{\partial x^2} + h_w A^i_w (T - \Psi) + U_{inf} A^0_w (T_{inf} + \Psi)$$  \hspace{1cm} (2.19)

These equations include the effects of radial and axial thermal dispersion, wall heat capacity, and axial conduction in the wall, as well as energy losses from the bed.

Deissler and Boegli

Deissler and Boegli in 1958 [29] proposed an alternative modelling of monodirectional flow through parallel layers of solid and fluid phases, obtaining this equation to calculate the effective thermal conductivity:

$$\frac{k_{eff}}{k_f} = \epsilon + (1 - \epsilon) k$$  \hspace{1cm} (2.20)

and the minimum by a series representation

$$\frac{k_{eff}}{k_f} = \frac{1}{\epsilon + (1 - \epsilon) k}$$  \hspace{1cm} (2.21)
where \( k = \frac{k_s}{k_f} \). These two correlations also act as the bounds for the other effective thermal conductivity correlations.

**Batchelor and O’Brien**

Batchelor and O’Brien in 1977 [14] derived a correlation with the focus on particle to particle contact. They, after fitting an empirical curve through the experimental data obtained this formulation to calculate effective thermal conductivity

\[
\frac{k_{eff}}{k_f} = 4.0 \log \left( \frac{k_s}{k_f} - 1 \right)
\] (2.22)

How can be seen, this equation is not an explicit function of porosity due to the assumed packing arrangement in the bulk region.

**Vortmeyer**

Vortmeyer, in 1989 [10], focused his attention on the solution of transient modelling of packed bed problem. Author investigated one-phase and two-phase models, searching for conditions that allow to represent packed bed with a single-phase model. Starting from Schumman two-equation model, he noticed that in case of low Reynolds number inside packed bed (Re < 100) the second derivatives of the solid and fluid temperature profiles can be considered the same

\[
\frac{\partial^2 T_f}{\partial x^2} = \frac{\partial^2 T_s}{\partial x^2}
\] (2.23)

becoming possible to replace the equations of the two-phase or two-phase model proposed by Schumman and other authors by an equivalent one-phase model equation, that is, after some mathematical manipulation, the following

\[
(1 - \epsilon) \rho_s c_s \frac{\partial \Theta}{\partial t} = -w \rho_f c_f \frac{\partial \Theta}{\partial x} + \left[ \lambda_0^c + \frac{(w \rho_f c_f)^2}{\alpha_S} \right] \frac{\partial^2 \Theta}{\partial x^2}
\] (2.24)

where the gas-phase heat capacity is neglected. Note that in this formulation temperature \( \Theta \) is equivalent to \( T_s \). Author also noticed that for high Reynolds numbers dispersion effects become increasingly negligible, but at very low Reynolds number
(Re < 10) packed beds are dominated by the effective quiescent conductivity.

Meier, Winkler and Wuillemin

Meier, Winkler and Wuillemin, in 1991 [3], investigated this problem using a different approach. Contrarily to other transient models which treat the solid phase as a continuum, their numerical model, called PACKBEDA, considers the solid bed as a discontinuous phase. The rock particles are approximated by spheres of equivalent diameter \(d_k\), and the rock pile is divided into sphere layers. In each layer \(n\), the air temperature \(T_{L,n}\) and the air velocity \(w_{L,n}\) are assumed to be constant and distributed homogeneously, which implies a uniform porosity \(\epsilon\). This model was validated with physical experiments, in a temperature range between 100° C and 800° C, and for different inlet temperature and different air flow rates. Authors found that the temperature stratification is improved by

- small porosity;
- low air velocity;
- small particle size;
- high density;
- large heat capacity of the storage material.

A higher thermal conductivity improves the temperature stratification only for large particle size and high air mass flow. Pressure drop, however, decreases mainly with a high porosity, a low air velocity and a big particle size.

Sanderson and Cunningham

In 1995 Sanderson and Cunningham [32] focused their attention on the phenomenon of radial thermal dispersion: in fact, through the packed bed an ideal temperature wave would move at constant velocity, with zero axial dispersion.
Authors proposed this formulation to calculate the velocity of the wave, formulated by Torab and Beasley:

\[ V_L = \frac{\rho_f c_f u_{su}}{\rho_f c_f \epsilon + \rho_s c_s (1 - \epsilon)} \]  

(2.25)

This formulation was obtained by assuming the working fluid has no axial dispersion, i.e. the temperature front moves as a step change through the packing. But these two velocities are idealizations: in practice the axial dispersion always occurs, and the wave does not move with constant velocity but slows down and spreads out as the wave progresses deeper into packing. Nonetheless, equation above may be used as first approximations in the absence of a more detailed numerical study.

**Mawire, McPhersons, van den Heetkamp and Mlatho**

Mawire, McPherson, van den Heetkamp and Mlatho in 2007 \[2\] implemented a numerical model of Schumman’s method. They tried four different solid phase, such as fused silica glass, alumina and stainless steel. Authors used Vortmeyer single-phase model, derived from Schumman’s model, and calculated the effective combined thermal conductivity from fluid-to-solid using this expression

\[ k_{eff} = k_s (1 - \epsilon) + k_f \epsilon \]  

(2.26)

obtaining very good results. Considering this, single-phase can be considered, upon this results, a very good model, and will be implemented inside Open\(\nabla\)FOAM solutor.

**Antwerpen, du Toit and Rousseau**

Antwerpen, du Toit and Rousseau recently, in 2010 \[33\], made a review of correlations between the packing structure and effective thermal conductivity in packed beds of mono-sized spherical particles. In this paper particular attention was given to the packing structure and heat transfer by solid conduction, gas conduction, contact area, surface roughness, as well thermal radiation. In this study the effective thermal conductivity \(k_{eff}\) is divided into three components:
• the effective conductivity $k^g_e$ through the fluid and point contact;
• the effective conductivity $k^c_e$ through contact area;
• the effective conductivity $k^r_e$ due to radiation.

The following summations gives total effective thermal conductivity $k_{eff}$:

$$k_{eff} = k^g_e + k^c_e + k^r_e \quad \text{or} \quad k_{eff} = k^{g,c}_e + k^r_e$$  \hspace{1cm} (2.27)

where $k^{g,c}_e$ is the combination of $k^g_e$ and $k^c_e$. Authors reviewed different formulations, some of the ones above mentioned, stating that the simple formulation proposed by Deissler and Boegli, that modeled effective heat transfer coefficient using an electrical similitude, act as bounds for the other correlations. This is demonstrated by authors plotting as example the calculated values of $\frac{k_{eff}}{k_f}$ respect to $K$, fixing porosity value $\epsilon$ equal to 0.36, as in figure 2.

![Figure 2.2: Effective thermal conductivity calculated using different models using $\epsilon = 0.36$ (neglecting contact area) versus experimental results.](image)

Another test, calculating effective thermal conductivity versus porosity, gave the same results, as showed in figure 2.
2.1 Summary

From the literature review it is clear that a univoque way to model heat transport phenomena inside porous media does not exist, but all focus on two aspects: the possibility to represent the porous media and the crossing fluid as a unique medium, and the concept of effective thermal conductivity in order to represent the thermal properties of fluid and solid together.

Inside the solver that was developed, the above explained one equation model was implemented: by this, the porous zone is the zone in which the crossing flow modify its thermal characteristics, that are dependant by fluid and solid thermal capacity and density. One equation model introduces the concept of thermal inertia that explain how temperature wave flows through porous zone. A solver based up one-equation model instead of two equation models, will be less computational expensive, and can be otained quick simulations due to the necessity to solve only one energy equation for each time step.

The second aspect which must be modeled to represent heat transfer in porous zone is the effective thermal conductivity that must take into account the fluid and
the solid contribution. While conduction heat transfer within a porous medium
is three-dimensional, the success of the two-dimensional spatially-periodic models
of Kunii and Smith and Zehner and Schlunder suggests that a high degree of
symmetry exists in porous media composed of solid particles. It is remarkable
how well these models predict the effective thermal conductivity for $1 \leq K \leq 10^3$, 
while very little empiricism is introduced. When $K > 10^3$, however, the accuracy
of these models diminishes. In case of $K > 10^3$, it is generally accepted that most
of the heat transfer through the porous medium occurs through the solid phase.
The solver was developed using the in series equivalence in order to obtain the
value of effective thermal conductivity inside porous zone. Although there is no
agreement between all the models presented, this model is a fixed point: the in
series equivalence, as stated by Deissler and Boegli, is the lower limit, as depicted
in figures 2 and 2. This model will be a conservative choice, and is the same upon
Fluent® calculates thermal properties in case of porous media crossed by fluid
flow. This choice allows to compare the results obtained by the simulation on the
same theoretical bases, in order to emphasize any possible discrepancies.
Chapter 3

TES modeling

In order to develop the solver, the numerical model of the TES must be deeply known. Mathematical model will cover the two zones TES is composed by, called in this work "free air zone", and "porous zone", as shown in figure 3.1.

Inside the "free air zone" the flow field must be correctly simulated to represent flow trajectory: in fact, it is necessary to know how, where, and with what velocity, fluid impacts with first rock layer. In "porous zone", elsewhere, heat transfer takes place and flow field must be represented taking into account heat transfer contribution, in order to obtain the exact values of velocity and pressure losses.
3.1 Free air zones

Thermo-vector fluid is not directly injected into the rock layer: on the top there is a zone of free air. The role of this zone is to make fluid stationary, and to spread it over the area above the first rock layer, in order to make the penetration inside the porous zone as uniform as possible.

3.1.1 Conservation laws: mass and momentum

Air flow is modeled using the Navier-Stokes equations, solving them using the so-called RANS form (Reynolds-average Navier-Stokes) that were developed to be used in numerical simulations \[28\]:

\[
\begin{align*}
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U}) &= 0 \\
\frac{\partial \rho \mathbf{U}}{\partial t} + \nabla \cdot (\rho \mathbf{U} \mathbf{U}) &= -\nabla (P + \frac{2}{3} \mu \nabla \cdot \mathbf{U}) + \nabla \cdot [\mu (\nabla \mathbf{U} + (\nabla \mathbf{U})^T)]
\end{align*}
\]

Note that in case of turbulent flows viscosity \( \mu \) value must take into account turbulence effect:

\[
\mu = \mu_{\text{lam}} + \mu_t
\]  \hspace{1cm} (3.1)

In this formulation \( \mu_{\text{lam}} \) is the laminar viscosity, depending on fluid, while \( \mu_t \) represents the turbulent viscosity and must be obtained using a suitable turbulence model: once noted turbulent kinetic energy \( k \) and turbulent dissipation ratio \( \epsilon \), setting \( C_\mu = 0.09 \) \[28\] :

\[
\mu_t = C_\mu \frac{k^2}{\epsilon}
\]  \hspace{1cm} (3.2)

These equations, when solved, give the velocity field inside the numerical domain.

3.1.2 Enthalpy conservation law

In order to calculate how temperature will spread, the well-known equation describing scalar transport is used \[35\]. In this case, instead of using temperature, it is more correct to calculate the value of enthalpy for unit of volume: once known, it is possible, through the relation \( h = c_p T \), to obtain the fluid temperature.
3.2. THE POROUS ZONE

Enthalpy distribution follows this equation

$$\frac{\partial \rho h}{\partial t} + \nabla \cdot (\rho h \mathbf{U}) = \nabla \cdot (k_{\text{eff}} T) + S_h$$  \hspace{1cm} (3.3)

The first part of the equation shows the time varying term, where the second part shows the convective term: it describes the dragging effect due to external motion, according to the velocity vector $\mathbf{U}$. These terms are matched by the diffusivity effect, and by a source term that, in case exists, must be specified. In TES case the diffusive term is dependent by the turbulent model used, while the source term is not present.

3.2 The porous zone

Packed bed, composed by rocks of similar form and diameter, is not a continuous media. Nevertheless, to make suitable the numerical simulation, it must be represented as a continuous media, in order to use the Navier-Stokes equations also in this zone of TES. The passage from free air zone to porous zone will bring some modification to conservation laws above explained, because of heat transfer mechanisms that are present in packed bed, and pressure losses caused by passage through rocks must be modelled. The above explained mass conservation law will remain the same, because there are not sinks in this zone.

3.2.1 Flow rate in porous zone

Once the flow had become stationary in the free air zone above the packed bed, it flows inside the porous zone, pushed by the pumping power. Inside the porous zone velocity decreases because of the resistance made by the rocks: Reynolds number changes. Reynolds number associated to porous zone is calculated taking into account the fluid velocity and the rock diameter, $d$. So, $Re$ of free air zone and porous zone are different.

$$Re_{\text{air}} = \frac{\rho U_a d_s}{\mu} \hspace{1cm} Re_{\text{rock}} = \frac{\rho U_r d_r}{\mu}$$
Different Reynolds numbers correspond to different types of flows. According to Dybbs and Edwards, which made this subdivision using laser anemometry and other visualization techniques \cite{1}, four type of flows are related to increasing $Re$:

- $Re < 1$: flow is laminar, is called of Darcy and is dominated by viscous forces. Pressure gradient varies linearly with velocity;

- $\sim 1 < Re < \sim 150$: increasing Reynolds number there is a transition that brings fluid to be dominated by inertial effect instead of viscous forces. Flow remains laminar. In this case flow is called of Darcy-Forchheimer;

- $\sim 150 < Re < \sim 300$: in this range compare oscillations and instabilities that generate vortices. Flow tends to become a turbulent fully developed flow;

- $Re > 300$: flow is dominated by turbulence, and is fully developed. Pressure losses are high.

As seen in schematization, transition between Darcy and Darcy-Forchheimer flow happens when $Re$ is equal to $10^2$. In the figure 3.2 is noticeable that friction factor changes formulation, passing from one linear to velocity (through $Re$) to another non-linear.

![Figure 3.2: Darcy, Darcy-Forchheimer, and Forchheimer flow](image)

Figure 3.2: Darcy, Darcy-Forchheimer, and Forchheimer flow
3.2. THE POROUS ZONE

Transition from a flow regime to another is due to generation of firsts eddies in flow, caused, for example, by a rotational fluid due to separation after an obstacle or a descending step, as in figure 3.3 and figure 3.4.

![Flow past a sphere (Re ≈ 0.1)](image1)

![Flow past a cube (Re ≈ 0.1)](image2)

Figure 3.3: Laminar flow

![Flow past a sphere (Re > 100)](image3)

![Flow past a cube (Re > 100)](image4)

Figure 3.4: Turbulent flow

Transition Reynolds number must be calculated taking into account the effect of porous zone. Porous zone is defined by geometrical properties: particle diameter $d_p$ and porosity $\epsilon$.

Combining these two parameters the definition of permeability $K$ can be achieved. Permeability is a value that quantifies the ability of the fluid to flow across the porous zone. Considering that, in a first approximation, porosity is the same in every direction, $K$ can be calculated as follows [34]:

$$K = \frac{d_p^2 \epsilon^3}{180(1 - \epsilon)^2}$$

(3.4)

or

$$K = \frac{1.39d_p^2}{e^{10.1(1-\epsilon)} - 1}$$

(3.5)
3.2. THE POROUS ZONE

In both cases, it results a quantity with measurement unit of \( [m^2] \). Inside the tank velocity averaged with volume, \( u_{\epsilon} \), has the same velocity scale of velocity that touches every element of porous zone: so it is possible to define a \( Re \) related to internal velocity, and based upon this classification, define laminar, transitional or turbulent flow inside the porous zone. It is noticed that first vortex shedding phenomena happens for \( Re_{dp} \sim 150 \), where \( Re_{dp} \) is calculated as follows:

\[
Re_{dp} = \frac{uD}{\nu}
\]

Because \( d_p \) is present in the equation for \( K \) and in equation for \( Re_{dp} \), a form for \( Re_{dp} \) independent from \( d_p \) can be obtained, replacing it inside the equation for \( Re_{dp} \):

\[
d_p = K^{1/2} \left[ \frac{180(1 - \epsilon)^2}{\epsilon^2} \right]^{1/2} \tag{3.6}
\]

or

\[
d_p = K^{1/2} \left[ \frac{\epsilon^{10.1(1-\epsilon)} - 1}{1.39} \right]^{1/2} \tag{3.7}
\]

Ignoring numerical part under the root, this equation is obtained:

\[
d_p \sim K^{1/2} \tag{3.8}
\]

and replacing it into Reynolds formulation, this form for Reynolds inside porous zone is obtained:

\[
Re_K = \frac{uK^{1/2}}{\nu} \tag{3.9}
\]

\( Re_K \) must be calculated in case of porous zone, before executing the simulation. Because eddies will appear for \( Re_K \) values between 100 and 200, in case of values less than 100 the flow inside the porous zone will be laminar, and so turbulence effect must be deleted and the numerical model updated.

Pressure losses

As previous seen, hot air motion can be modeled using Navier-Stokes equations, with some modification passing from free air zones to porous zones. In the first zones Navier-Stokes equations solve exactly the velocity field, but flow through
porous media is modeled by attenuating the time derivative and by adding a sink term to the Navier-Stokes equations.

\[
\frac{\partial}{\partial t} (\epsilon \rho u_i) + u_j \frac{\partial}{\partial x_j} (\rho u_i) = - \frac{\partial p}{\partial x_i} + \mu \frac{\partial \tau_{ij}}{\partial x_j} + S_i
\]

Sink term previous cited can be modeled using different formulations:

**Darcy-Fochheimer law** Source term, indicate as \( S_{v/i} \), is composed by two parts: the first is relative to pressure losses due to viscous effect, and is linearly proportional to velocity of fluid inside the packed bed, while the second term, due to inertial effects, is proportional to square of velocity:

\[
S_{v/i} = -\left( \mu D + \frac{1}{2} \rho |u_{jj}| F \right) u_i
\]

Note that the solver need as input the value of the two dimensional vectors \( D \) end \( F \) (which default value is zero). They are then added to diagonal term of tensors \( D_{ij} \) and \( F_{ij} \) in 3.11. \( D \) and \( F \) value are related to the properties of the fluid and the porous zone. In particular, respect to rock geometry (with the diameter \( d_r \)) and porosity \( \epsilon \) (fraction of rocks respect to air).

- **Viscous resistance** \( D \) is the inverse of the cited permeability:

\[
D = \frac{150 \left( 1 - \epsilon \right)^2}{d_p^2 \epsilon^3}
\]

- **Viscous inertia** is modeled by term \( F \), calculated as follows:

\[
F = \frac{3.5(1 - \epsilon)}{d_p \epsilon^3}
\]

Once calculated, \( D \) and \( F \) represent the effective properties of porous zone: the developed solver obtains itself these values by porosity and particle diameter, instead of Fluent \(^\text{®} \) solver that ask to the user directly \( D \) and \( F \). Obtained equation
3.3 Heat transfer between fluid and rocks

is known as the Darcy-Forchheimer equation.

$$\frac{\partial}{\partial t} (\epsilon \rho u_i) + u_j \frac{\partial}{\partial x_j} (\rho u_i) = - \frac{\partial p}{\partial x_i} + \mu \frac{\partial \tau_{ij}}{\partial x_j} - \left( \mu D_{ij} + \frac{1}{2} \rho |u_{kk}| F_{ij} \right) u_i$$  \hspace{1cm} (3.14)

In the case of simple homogeneous porous media it becomes

$$\frac{\partial}{\partial t} (\epsilon \rho u_i) + u_j \frac{\partial}{\partial x_j} (\rho u_i) = - \frac{\partial p}{\partial x_i} + \mu \frac{\partial \tau_{ij}}{\partial x_j} - \left( \mu D_{ij} + \frac{1}{2} \rho |u_{jj}| F_{ij} \right) u_i$$  \hspace{1cm} (3.15)

where $D_{ij}$ and $F_{ij}$ are represented as the scalars $D$ and $F$.

**Power-law model**  The source term, that models pressure losses, can be modeled also using a power law, different from formulation shown above. This solution is almost never used, but it is shown to complete this overview.

$$S = -\rho C_0 |u_i|^{(C_1-1)/2}$$  \hspace{1cm} (3.16)

3.3 Heat transfer between fluid and rocks

Traditional scalar transport law for enthalpy is not suitable inside the porous zone, because rocks absorb heat from fluid, growing their internal energy.

Two equations models are most used working with finite differences, because they give good results relatively to modest calculation power they need. The core of this working way is to write an equation for each phase, by inserting a matching condition to link one equation to the other. It is important to note that, under certain conditions, tho-phase model brings to the one-equation version.

**Applicability condition for 1-equation model**

In order to use one-equation instead of two-equation model for thermal exchange simulation inside the porous zone, condition that rocks change their temperature instantly, coming to fluid temperature, must be satisfied. This condition can be evaluated considering the concept of thermal equilibrium. In analytical approach, conditions of instantly change in temperature can be obtained and demonstrated.
3.3. HEAT TRANSFER BETWEEN FLUID AND ROCKS

using the definition of \( NTU \): number of transfer unit, as proposed by Hughes in 1975 [25]. NTU is defined as

\[
NTU = \frac{h_v AL}{\dot{m}c_p}
\]

(3.17)

and indicates how quickly temperature of rocks changes in contact area between fluid and rocks:

\[
\frac{\partial T_s}{\partial \tau} = NTU \left(T_f - T_s\right)
\]

(3.18)

where \( \tau \) is adimensionalized time:

\[
\tau = \frac{(\dot{m}c_p) ft}{(\rho c_p)_s (1 - \epsilon) AL}
\]

(3.19)

NTU depends on thermal properties of air and on his velocity. In fact, heat volumetric exchange coefficient, multiplied for rock volume, appears in numerator, and at denominator there is the mass flow, that is related to velocity of fluid through rocks. Bigger is heat transfer coefficient, and lower is convective velocity, bigger will NTU result.

Equation 3.16 can be interpreted in two ways, bringing the same result. If \( NTU \rightarrow \infty \) derivative of temperature tends to \( \infty \), indicating that the temperature variation is instantaneous, as requested to have thermal equilibrium. Focusing on term \((T_f - T_s)\), the NTU coefficient goes to denominator of the derivative, making sure that the term on the left of the equal tends to 0. Also in this case, thermal equilibrium is satisfied, obtaining \( T_f = T_r \). As mentioned, theory requires a NTU value equal, or tending, to \( \infty \) to satisfy thermal equilibrium, but experimental tests have shown the a NTU pair or bigger than 25 give same results of NTU pair to \( \infty \). Once this requirement is satisfied, there is the "local thermal equilibrium".

\[
\Delta T_d < \Delta T_l < \Delta T_L
\]

(3.20)

where pedicles has these meanings: \( d = \) distance solid – air, \( l = \) distance in representing volume e \( L = \) system dimension. Temperature diffusion in air or in rocks has therefore the same behaviour:

\[
\frac{\partial^2 T_f}{\partial x^2} = \frac{\partial^2 T_s}{\partial x^2}
\]

(3.21)
3.3. HEAT TRANSFER BETWEEN FLUID AND ROCKS

It must be noticed that these values are multiplied for conduction coefficient of air and rocks, so before to sum them, an effective coefficient that can takes into account the porosity of the medium must be calculated. If the condition above mentioned are satisfied, is possible to use the so-called 1 equation model. From previous equations the term representing the thermal exchange on the surface can be isolated establishing it equal to the obtained results.

- From fluid phase it is obtained:

\[ h_{fs}A_{fs}(T_s - T_f) = \epsilon \rho_f c_f \left( \frac{\partial T_f}{\partial t} + V_\infty \frac{\partial T_f}{\partial x} \right) \]  

\[ (3.22) \]

- from solid phase, proceeding in the same way:

\[ h_{fs}A_{fs}(T_s - T_f) = -(1 - \epsilon) \rho_s c_s \frac{\partial T_s}{\partial t} + k_r \frac{\partial^2 T_s}{\partial x^2} \]  

\[ (3.23) \]

Forcing \( T_{\text{fluid}} = T_{\text{solid}} = \theta \), previously verified, end setting equal the equations, it can be obtained this formulation:

\[ \left( \epsilon \rho_f c_f + (1 - \epsilon) \rho_s c_s \right) \frac{\partial \theta}{\partial t} + \rho_f c_f V_\infty \frac{\partial \theta}{\partial x} = k_{\text{eff}} \frac{\partial^2 \theta}{\partial x^2} \]  

\[ (3.24) \]

Obtained equation shows the temperature distribution in time and in space, inside a porous media crossed by a thermovector fluid. As previously mentioned, also \( k_{\text{eff}} \) must be calculated to complete this equation, in order to consider the different convection factor of air and rocks. To make a more general formulation, spatial derivatives can be shown in form of divergence; \( V_\infty \) term indicates convection velocity of the fluid inside the porous zone, and must be calculated taking into account pressure losses inside the zone, using the Darcy model above introduced.

In the obtained equation can be recognised, in the first term, the thermal inertia, in the second term the theoretical temperature wave velocity (the real value must be divided by the thermal inertia) and in the third term the effect of the rocks thermal conductivity.

\[ \left( \epsilon \rho_f c_f + (1 - \epsilon) \rho_s c_s \right) \frac{\partial \theta}{\partial t} + \rho_f c_f V_\infty \nabla \theta = \nabla \cdot (k_{\text{eff}} \nabla \theta) \]  

\[ (3.25) \]
3.3.3 Heat Transfer Between Fluid and Rocks

3.3.1 Effective thermal conductivity of air

Effective thermal conductivity in free air zones In the enthalpy conservation law the heat diffusivity is modeled by the term called effective thermal conductivity: \( k_{\text{eff}} \). In case of turbulent flows, as done in momentum equation with air viscosity, it must be updated, because of turbulence effect. Effective thermal conductivity of air will become the sum of air property and turbulence effect, modeled by \( \nu_t \)

\[
k_{\text{eff}} = k_{\text{lam}} + \frac{\rho \nu_t}{\sigma_t}
\] (3.26)

where \( \nu_t \) is known from previous calculation and \( \sigma_t \) is the turbulent Prandtl number. Note that, by definition, in RANS model \( \sigma_t = 1 \). [35].

Effective thermal conductivity inside porous zone Calculation of thermal conductivity inside the porous zone is not easy because of the non-continuous form of this zone: adequate simplification must be used. \( k_{\text{eff}} \) value can be obtained taking into account air and rock properties using electrical similitude: as just seen in literary review, Bejan proposed an electrical analogy, using a "series" model to calculate how conductivity of air and rocks interact. Inside the created solver will be used this model, the same used by the commercial code Fluent \(^\circledR\).

Role of porosity is represented by the different weighting factors that multiply the thermal conductivity: air thermal conductivity is multiplied directly for porosity, when rock conductivity is multiplied for the remaining fraction of the entire. Formulation obtained is the following:

\[
k_{\text{eff}} = \epsilon k_f + (1 - \epsilon) k_s
\] (3.27)
3.4 Gravity effects

Presence of gravity field cannot be excluded in this fluid dynamic solver. Because of the presence of hot and cold air, buoyancy effect is not negligible, and must be represented with its consequences above velocity field \cite{34}, \cite{7}, \cite{17}. Gravity acts on momentum equation: hot air tends to stay above cold air, so the velocity of penetration will be less than in a case without gravity field. This effect is modeled modifying momentum equation in order to add buoyancy effect, according to Stevino’s law \cite{36} that quantifies pressure value due to gravity vector and to distance (parallel to gravity vector) used as reference, \( h \) inside the formulation:

\[
p = \frac{F}{S} = \frac{mg}{S} = \frac{(\rho V)g}{S} = \frac{\rho Sh g}{S} = \rho gh \quad (3.28)
\]

Momentum equation must be modified adding gravity effect, that generates a thrust in direction opposite to flow, proportional to gravity vector value.

\[
\frac{\partial \rho U}{\partial t} + \nabla \cdot (\rho U U) + \nabla \cdot (\rho R) = -\nabla p + \rho g_{\text{gravity}} \quad (3.29)
\]

The difference between this and standard formulation is all in the \( \rho g \) term, that introduces gravity effect in the model. Focusing on the storage problem, because of the high velocity in use, gravity effect during first seconds of charge is not visible: the inertial forces are too large and overtake the buoyancy effect due to gravity.

Buoyancy effect had also a big role in stratification: inside the porous zone the velocity becomes low, so the gravity force overtakes inertial effects, taking advantages for the correct behavior of the TES. Thanks to buoyancy due to cold, denser air at the bottom of the tanks, hot air tends to redistribute in horizontal sheets, making all the porous zone homogeneous. Homogeneization of the porous zone plays an important role in the discharging phase: hot air that will be ejected will not present cold spot that could give problems in the energy generator.
3.5 TURBULENCE MODELING

In order to calculate turbulence quantities inside the tank, the realizable $k-\epsilon$ model of RANS model was chosen. Turbulent kinetic energy $k$ and turbulence dissipation ratio $\epsilon$ are obtained solving a conservation equation for each one:

- Equation for $k$

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \mu_t \frac{\partial}{\partial x_j} \right) \frac{\partial k}{\partial x_j} \right] + P_k + P_b + \rho \epsilon - Y_m + S_k$$

- Equation for $\epsilon$

$$\frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial x_i}(\rho \epsilon u_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \mu_t \frac{\partial}{\partial x_j} \right) \frac{\partial \epsilon}{\partial x_j} \right] + \rho C_1 S \epsilon - \rho C_2 \frac{\epsilon^2}{k + \sqrt{\nu \epsilon}} + C_1 C_3 P_b + S_\epsilon$$
3.5. TURBULENCE MODELING

In these equations

- $P_k$ represents the generation of turbulence kinetic energy due to the mean velocity gradients;

- $P_b$ is the generation of turbulence kinetic energy due to buoyancy;

- $Y_M$ represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate;

- $C_{1\epsilon}, C_{2\epsilon}, C_{3\epsilon}$ are constants;

- $S_k$ and $S_\epsilon$ are user-defined source terms, if presents.

The turbulent (or eddy) viscosity acts on momentum conservation equation: it multiplies dissusive term. Turbulent viscosity $\mu_t$ is computed by combining $k$ and $\epsilon$ as follows:

$$\mu_t = \rho C_\mu \frac{k^2}{\epsilon}$$

where $C_\mu$ is obtained by solving $C_{\mu a} = \frac{1}{A_0 + A_2 \frac{\mu_t}{\rho \nu}}$.

In case of turbulent flow, the effective viscosity that will be the summation of laminar (property of the fluid) and turbulent contribution, so $\mu_{\text{eff}} = \mu_{\text{laminar}} + \mu_t$. It must be noted that, representing the porous zone as a fluid zone with different thermal and inertial properties, the conservation equations for turbulence quantities are solved without any modification inside the porous zone: the solid medium has no effect on the turbulence generation or dissipation.

This assumption may be reasonable if the medium’s permeability is quite large and the geometric scale of the medium does not interact with the scale of the turbulent eddies. In case of very low velocity inside porous zone, Reynolds number can become as low to make flow laminar. In case of laminar flow, must be suppressed the effect of turbulence in the medium.

In case of RANS model this result can be achieved setting equal to zero the turbulent contribution to viscosity $\mu_t$. By this way, the effective viscosity of the fluid, that act on momentum equation, will be the only laminar part, while outside the porous zone the effective turbulent viscosity is obtained with standard form.
3.5. TURBULENCE MODELING

- Effective viscosity inside the *free air zone*

\[ \mu_{\text{eff}} = \mu_{\text{lam}} + \mu_t \quad (3.30) \]

- Effective viscosity inside the *porous zone*

\[ \mu_{\text{eff}} = \mu_{\text{lam}} \quad (3.31) \]

The definition of turbulence of laminar flow inside porous zone must be obtained from previous cited \( R\ell_K \) that must be evaluated by user.
Chapter 4

Open∇FOAM

4.1 The choice of Open∇FOAM

Numerical solver object of this thesis project was based upon the free software Open∇FOAM, that was created by Hrvoje Jasak as PhD thesis work [18]. It presents two main advantages that make it preferable to other differential equation solvers.

**It is a free software** it is allowed to access to source files, in order to directly modify them if required, or adding modules if a request can not be solved by original software.

**It uses an intuitive form to represent equations** Open∇FOAM is an optimal linkage between theoretical study and numerical implementation of fluid dynamic concepts. Equation writing is very easy: it is similar to mathematical language. This approach allows a simple passage from numerical modelation to software compilation, also for beginners with little knowledge of coding.

It follows a discussion about the two points above mentioned.

4.1.1 Free software

A *Free software* is a software released with a licence that permits the use to anyone, encouraging studing, modifications and redistribution. Free software idea was
born in 1980, when R. Stallman created Free Software Foundation (FSF [23]) with objective to create an operative system completely free: this brought to creation of GNU/Linux in 1991, a UNIX clone freely distributable and modifiable. According to FSF, a software that can be defined free, must guarantee this fundamentals freedoms:

**Freedom 0** freedom to use software for every scope;

**freedom 1** freedom to study how software works and eventually adapt it to own necessity: access to code is obviously a pre-requisite.

**freedom 2** freedom to re-distribute copies of program to help others: in particular, in addition to source code must be provided also binary packages;

**freedom 3** freedom to improve the code and to distribute the improvements publicly, so that it can benefit the community.

However, the word free does not imply the ability to use the software in an indiscriminate manner. It is subject to a license designed to facilitate the sharing of knowledge without expropriating the authors of intellectual property: such as the original authors or copyright holders must always be mentioned in all the modified and subsequently distributed programs.

According to the supporters of free software, it has many advantages over proprietary software: thanks to the availability of the source code, user can adapt the program to his particular needs, and because the source code is subject to the continuous revision of a very large user base, it can speed considerably the process of identifying and correction of errors, optimizing calculation efficiency. However, it is undeniable that there are also limitations and disadvantages compared to proprietary software such as the documentation: if available, is often incomplete or out of date.

### 4.1.2 Code in Open\(\nabla\)FOAM

Central theme of design of Open\(\nabla\)FOAM is the code representation of equations. They are written using already available classes, built in a way that equation written in C++ language are as similar as possible to mathematical formulation,
including partial derivative \[22\]. For example, equation that describes momentum of a fluid:

\[
\frac{\partial \rho U}{\partial t} + \nabla \cdot \phi U - \nabla \cdot \mu \nabla U = -\nabla p
\]

is written in this form:

\[
\text{solve} \\
(fvm::ddt(rho, U) + fvm::div(phi, U) - fvm::laplacian(mu, U) == -fvc::grad(p));
\]

In this case are obvious the meanings of \(\text{ddt}\), \(\text{div}\), \(\text{laplacian}\) and \(\text{grad}\), while \(\text{fvm}\) and \(\text{fvc}\) indicate \textit{finite volume method} and \textit{finite volume calculus} respectively. \(\text{fvm}\) implies an implicit resolution, and \(\text{fvm}\) indicates an explicit resolution. Equations are represented by placing implicit terms on the left of equal sign, and explicit terms on the right. As seen, the passage from numerical formulation to code representation is not difficult, as long as terms of the equations are correctly discretized.

### 4.2 Pressure-Velocity Coupling

The core of fluid dynamic simulation is the solve of Navier-Stokes equations, that show how velocity and pressure are linked inside the fluid. This equation is just solved in various OpenFOAM solvers, and will be used without modifications.

To make it more comprehensible, a brief description of its resolution is reported.

Consider the discretised form of the Navier-Stokes system

\[
\begin{cases}
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho U) = 0 \\
\frac{\partial \rho U}{\partial t} + \nabla \cdot (\rho UU) = \rho g - \nabla \left( P + \frac{2}{3} \mu \nabla \cdot U \right) + \nabla \cdot [\mu(UU + (\nabla U)^T)]
\end{cases}
\]

The form of the equations shows linear dependence of velocity on pressure and vice
versa. This inter-equation coupling requires a special treatment. Simultaneous algorithms operate by solving the complete system of equations in the same time over the whole domain. Such a procedure might be considered when the number of computational points is small and the number of simultaneous equations is not too large. The resulting matrix includes the inter-equation coupling and is several times larger than the number of computational points. The cost of a simultaneous solution is great, both in the number of operations and memory requirements.

In the segregated approach the equations are solved in sequence. A special treatment is required in order to establish the necessary inter-equation coupling. PISO, SIMPLE and their derivatives are the most popular methods of dealing with inter-equation coupling in the pressure-velocity system.

4.2.1 The PISO Algorithm for Transient Flows

This pressure-velocity treatment for transient flow calculations, known as PISO algorithm, can be described as follows:

- **momentum predictor** the momentum equation is solved first. The exact pressure gradient source term is not known at this stage – the pressure field from the previous time-step is used instead. The solution of the momentum equation gives an approximation of the new velocity field;

- **pressure solution** using the predicted velocities, the pressure equation can be formulated. The solution of the pressure equation gives the first estimate of the new pressure field.

- **explicit velocity correction** solving previous step, a set of conservative fluxes, consistent with the new pressure field, is available. The velocity field should also be corrected as a consequence of the new pressure distribution. Velocity correction is done in an explicit manner.
In other words, the PISO loop consists of an implicit momentum predictor followed by a series of pressure solutions and explicit velocity corrections. The loop is repeated until a pre-determined tolerance is reached. After each pressure solution, a new set of conservative fluxes is available.
4.2.2 The SIMPLE Algorithm

If a steady-state problem is being solved iteratively, it is not necessary to fully resolve the linear pressure-velocity coupling, as the changes between consecutive solutions are no longer small. Non-linearity of the system becomes more important, since the effective time-step is much larger. The SIMPLE algorithm is formulated to take advantage of these facts:

- an approximation of the velocity field is obtained by solving the momentum equation. The pressure gradient term is calculated using the pressure distribution from the previous iteration or an initial guess. The equation is under-relaxed in an implicit manner, with the velocity under-relaxation factor $\alpha_U$;

- the pressure equation is formulated and solved in order to obtain the new pressure distribution;

- a new set of conservative fluxes is calculated. As it has been noticed before, the new pressure field includes both the pressure error and convection-diffusion error. In order to obtain a better approximation of the “correct” pressure field, it would be necessary to solve the pressure equation again. On the other hand, the non-linear effects are more important than in the case of transient calculations. It is enough to obtain an approximation of the pressure field. The pressure solution is therefore under-relaxed in order to take into account the velocity part of the error

$$p^{new} = p^{old} + \alpha_p (p^p - p^{old})$$

If the velocities are needed before the next momentum solution, the explicit velocity correction is performed.

The recommended values of under-relaxation factors are $\alpha_p = 0.2$ for the pressure and $\alpha_u = 0.8$ for momentum [27].
4.2. PRESSURE-VELOCITY COUPLING

Figure 4.2: SIMPLE algorithm flowchart

### 4.2.3 PIMPLE

PISO simply does not iterate over the equations. It is an explicit method, so, it need, in order to not diverge, that *CFL condition* is satisfied. This condition states that *CFL number*, calculated as $C = \frac{u \cdot dt}{\delta x}$ must be less than 1: in case of
great advection velocity $u$ and small mesh cells $\delta x$, time-step results very small, and can give to very time-expensive simulations. In these cases can be used a new iterative scheme, called PIMPLE. It is a merge between PISO and SIMPLE: it is a PISO scheme, with outer correctors that correct pressure value before of the next time-step.

**PIMPLE ALGORITHM**

![PIMPLE algorithm flowchart](image)

Figure 4.3: PIMPLE algorithm flowchart
PIMPLE algorithm offers more control by providing $n\text{Outer Iterations}$ parameter. Note that in case of $n\text{Outer Iterations} = 1$, PIMPLE is just simply a PISO algorithm. This algorithm allows the use of larger time step, greater than time step obtained satisfying CFL condition, and is used in case of large time-step transient solver for incompressible flow, like in TES case. So, definitely some addition costs on computation, caused by outer iterations, could improve the results: simulations will be more CPU-expensive, but larger time-steps are allowed.

4.3 Solution Procedure for Navier-Stokes system

It is now possible to describe the solution sequence for the Navier-Stokes system with additional coupled transport equations (e.g. a turbulence model, combustion equations, energy equation or some other equations that influence the system). In transient calculations, all inter-equation couplings apart from the pressure-velocity system are lagged. If it is necessary to ensure a closer coupling between some of the equations (e.g. energy and pressure in combustion), they are included in the PISO loop. A transient solution procedure for incompressible turbulent flows can be summarised as follows:

1. set up the initial conditions for all field values;
2. start the calculation of the new time-step values;
3. assemble and solve the momentum predictor equation with the available face fluxes;
4. go through the PISO loop until the tolerance for pressure-velocity system is reached. At this stage, pressure and velocity fields for the current time-step are obtained, as well as the new set of conservative fluxes;
5. using the conservative fluxes, solve all other equations in the system. If the flow is turbulent, calculate the effective viscosity from the turbulence variables;
6. if the final time is not reached, return to step 2.
Chapter 5

rhoPorousHeatPimpleFoam

5.1 Open\nFOAM solver

Developed solver is named \texttt{rhoPorousHeatPimpleFoam}, according to terminology used to designate other Open\nFOAM solvers:

\textbf{rho} denotes that in this case fluid is compressible, so \textit{rho} can change;

\textbf{Porous} means presence of porous zone;

\textbf{Heat} suggests presence of heat exchange effects;

\textbf{Pimple} indicates type of solver used (eg: SIMPLE, or PISO);

\textbf{Foam} by default

A description of the developed solver will follow, including subroutines which were added by author because not present in original code.

In figure 5.1 a flux diagram representing how solver works is depicted: as suggested by name, PIMPLE algorithm solves time-derivative equations. PIMPLE algorithm was chosen in order to increase time-step dimension to obtain good results in short time. Next chapters will explain, in order to appearance, commands that the solver calls.
5.1. **OPENFOAM SOLVER**

![Flowchart](5.1. rhoHeatPorousPimpleFoam flowchart)

**Figure 5.1: rhoHeatPorousPimpleFoam flowchart**

### 5.1.1 **rhoEqn.H**

First of all, once stored the previous time-step values of pressure and enthalpy, solver calls the "\( \rho \) equation", that updates air density inside every cell of domain.
in order to satisfy the conservation of mass equation:

$$\frac{\partial \rho}{\partial t} + \nabla (\rho U) = 0$$  \hspace{1cm} (5.1)

5.1.2 UEqn.H

Function describing momentum conservation was obtained in previous chapter, and is shown in \[5.2\]. It is the well-known momentum conservation equation for fluids, with an attenuating factor (called \textit{a.f.}) that multiplies time derivative, activated by a subroutine, and a sink term, on the right of equal, that acts in the porous zone:

$$\sum_{a.f.} \epsilon \frac{\partial}{\partial t} (\rho u_i) + u_j \frac{\partial}{\partial x_j} (\rho u_i) = - \frac{\partial p}{\partial x_i} + \mu \frac{\partial \tau_{ij}}{\partial x_j} - \left( \mu D_{ij} + \frac{1}{2} \rho |u_{jj}| F_{ij} \right) u_i$$  \hspace{1cm} (5.2)

5.1.3 pEqn.H

After the first cycle of calculation a new velocity field is obtained, using \textit{U} equation. Was also calculated the new thermal field inside the domain, so pressure field must be corrected to close the PISO loop to have the correct match between velocity and pressure

5.1.4 hEqn.H

Equation for enthalpy conservation was developed in chapter \[3\]. It is different for free air zones and porous zones, but can be fused inside one only, changing form depending on domain zone in which it is solved

- Free air zones

$$\frac{\partial h}{\partial t} + V_{\infty} \nabla h = \nabla \cdot \left( \frac{k_{\text{air}}}{\rho_{\text{air}} c_{\text{air}}} \nabla h \right)$$  \hspace{1cm} (5.3)

- Porous zones

$$\left( \epsilon + (1 - \epsilon) \frac{\rho_s c_s}{\rho_f c_f} \right) \frac{\partial h}{\partial t} + V_{\infty} \nabla h = \nabla \cdot \left( \frac{k_{\text{eff}}}{\rho_{\text{eff}} c_{\text{eff}}} \nabla h \right)$$  \hspace{1cm} (5.4)
5.2. HEATPOROUSZONE LIBRARY

Note that the term that multiplies derivation-in-time of enthalpy, acts only in porous zone, like in \( U \) equation. Term relative to heat transfer is instead present in both formulation, so \( k_{eff} \) variable is used in both cases, re-computing his value depending on presence of rocks and related heat transfer properties.

5.2 heatPorousZone library

Solver must be linked to libraries that execute low-level pre and post-processing operations. In this case, in addition to standard libraries a new dedicated library was created, focused on the porous zone, called heatPorousZone. Inside this library are included all functions that act only inside porous zone: thanks to this modification, equations solving inside porous zone is made easier and more clear, because they can be modified outside the solver. This library includes different sub-routines, relative to velocity field and enthalpy:

- sub-routine relative to velocity field:
  
  \text{modifyDdt} \quad \text{pre-multiplies time derivative term of the velocity equation by porosity value, attenuing it;}

  \text{addPowerLawResistance} \quad \text{introduces drag term due to porous zone, modeling drag using power-law form;}

  \text{addViscousInertialResistance} \quad \text{introduces drag term due to porous zone, modeling drag using visco-inertial form: inputs are value of } F e E \text{ due to packed bed properties;}

- sub-routine relative to enthalpy:
  
  \text{modifyDdtE} \quad \text{modifies time-derivative term, pre-multiply time derivative part of the enthalpy equation for the correspondent thermal inertia.}

  \text{alphaGlobal} \quad \text{calculates effective thermal diffusivity coefficient inside porous zone due to air and rock thermal conductivity.}
Chapter 6

Test case: *Phoebus Technology Program Solar Air Receiver TSA*

Developed solver was validated comparing experimental data from a test treated in the 1991 paper by A. Meier and C. Winkler "Performance characteristics of high temperature sensible thermal storage for solar air circuits" [3] that presents a comparison between the numerical model developed by authors and experimental results. These results, related to a TES device called *Phoebus Technology Program Solar Air Receiver TSA*, will be compared with results obtained by Open\∇FOAM solver object of this thesis work, acting on a 3d model that will represent the real cited TES device. In table 6.1 dimensions of the studied TES are shown: it is an axisymmetric tank, filled by rocks in the central part.

<table>
<thead>
<tr>
<th>Dimension</th>
<th>value [m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Overall height</td>
<td>4.00</td>
</tr>
<tr>
<td>Height of upper free air zone</td>
<td>0.42</td>
</tr>
<tr>
<td>Porous zone height</td>
<td>2.10</td>
</tr>
<tr>
<td>Height of lower free air zone</td>
<td>0.42</td>
</tr>
<tr>
<td>Input/output diameter</td>
<td>1.00</td>
</tr>
<tr>
<td>Porous zone diameter</td>
<td>3.30</td>
</tr>
</tbody>
</table>

Table 6.1: Solar Air Receiver TSA: geometrical characteristics

In figure 6.1 is given a representation of the storage disposal.
Data relatives to air in input and porous zone are shown in table 6.2.

### Air data

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Symbol</th>
<th>Unit</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Air temperature in inlet</td>
<td>$T_i$</td>
<td>$^\circ C$</td>
<td>700</td>
</tr>
<tr>
<td>Air temperature inside tank</td>
<td>$T_u$</td>
<td>$^\circ C$</td>
<td>200</td>
</tr>
<tr>
<td>Air mass flow</td>
<td>$m_f$</td>
<td>kg/s</td>
<td>3.41</td>
</tr>
<tr>
<td>Pressure drop</td>
<td>$\Delta p$</td>
<td>Pa</td>
<td>$\leq$ 2000</td>
</tr>
</tbody>
</table>

### Rocks data

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Symbol</th>
<th>Unit</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density</td>
<td>$\rho_r$</td>
<td>kg/m$^3$</td>
<td>2400</td>
</tr>
<tr>
<td>Average diameter</td>
<td>$d$</td>
<td>m</td>
<td>0.010</td>
</tr>
<tr>
<td>Specific heat</td>
<td>$c_{Pr}$</td>
<td>J/kgK</td>
<td>1075</td>
</tr>
<tr>
<td>Heat transfer coefficient</td>
<td>$hTc_r$</td>
<td>W/mK</td>
<td>10</td>
</tr>
<tr>
<td>Porosity</td>
<td>$\epsilon$</td>
<td>[-]</td>
<td>0.390</td>
</tr>
</tbody>
</table>

Table 6.2: Solar Air Receiver TSA: air and rocks characteristics
6.1. GEOMETRICAL DOMAIN AND BOUNDARY CONDITION

The experiment was carried out in order to monitor how the thermocline moves during charge cycle. Results was obtained measuring air temperature along tank vertical axis, and collected at different \( \Delta t \). These data, available inside the paper, was appropriately digitized in order to compare them with data obtained using Fluent \(^\circledR\) and Open\(\nabla\)FOAM.

The original graph is shown in figure 6.2: it represents the temperature distribution for charging cycle, focusing on porous zone. Temperature is displayed in Celsius degrees.

![High Temperature Thermal Storage - Temperature Distribution for Charging Cycle](image)

Figure 6.2: Solar Air Receiver TSA: results

6.1 Geometrical domain and boundary condition

Due to the axisymmetry of the geometrical domain, it can be split into four parts, solving the flow field on only one quarter, representing the others with adequate symmetrical boundary conditions. Table 6.3 shows boundary conditions imposed: at inlet mass flow and temperature is fixed and know, wall is set adiabatic.
and solid. Outlet condition is stated in order to not have any pressure drop across outlet so, measuring pressure on inlet surface, the total pressure drop can be obtained directly.

<table>
<thead>
<tr>
<th>Zone</th>
<th>boundary condition</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet</td>
<td>Mass flow, $T_{inlet}$</td>
</tr>
<tr>
<td>Outlet</td>
<td>Pressure</td>
</tr>
<tr>
<td>Symmetry</td>
<td>Symmetry for mass, momentum, heat transfer</td>
</tr>
<tr>
<td>Wall</td>
<td>Solid wall, no trespassing, adiabatic</td>
</tr>
</tbody>
</table>

Table 6.3: Boundary conditions

Figure 6.3 depicts geometrical domain with focus on different domain zone and boundary conditions.

Figure 6.3: Representation of TES: zones and boundary conditions

Note that external walls of the tank are considered adiabatic: there is not heat exchange within external domain. This boundary condition does not allow to extend computational domain over the tank, reducing computational time.
6.2 Computational mesh

Computational domain is split into cells: inside every one of them, equations above explained must be solved. In order to obtain good quality results cells must be more regular as possible: in this test cells are all hexahedriq, with maximum skewness factor, an indicator of the mesh quality, bounded between 0 and 1, equal to 0.69.

Mesh specifications are reported inside table 6.4.

<table>
<thead>
<tr>
<th>Mesh stats</th>
<th>Value</th>
<th>Mesh stats</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Points</td>
<td>163'493</td>
<td>Max face area</td>
<td>0.002</td>
</tr>
<tr>
<td>Faces</td>
<td>471'790</td>
<td>Min face area</td>
<td>6.260e-05 (m^2)</td>
</tr>
<tr>
<td>Internal faces</td>
<td>453'698</td>
<td>Max cells volume</td>
<td>8.901e-05 (m^3)</td>
</tr>
<tr>
<td>Cells</td>
<td>152'248</td>
<td>Min cells volume</td>
<td>1.878-06 (m^3)</td>
</tr>
<tr>
<td>Max aspect ratio</td>
<td>5.866</td>
<td>Max non-orthogonality</td>
<td>23.585</td>
</tr>
<tr>
<td>Max skewness</td>
<td>0.691</td>
<td>Ave. non-orthogonality</td>
<td>4.652</td>
</tr>
</tbody>
</table>

Table 6.4: Mesh statistics

In figure 6.4 can be seen a representation of computational domain.

![Computational mesh](image-url)
First comparison with experimental data was performed with commercial software Fluent®. In order to make transient simulation, type of simulation must be set on transient, and to activate energy exchange, Energy model must be turned on.

The porous zone is modeled by two values related to porosity and particle diameter, previous cited in chapter 4: $K^{-1}$ and $F$, representing the viscous resistance coefficient and the inertial resistant coefficient respectively.

- $D$ defines laminar part of porous drag. It is the inverse of permeability and its value is obtained by definition

$$K = \frac{d_p^2 \epsilon^3}{150(1-\epsilon)^2} \rightarrow D = \frac{150(1-\epsilon)^2}{d_p^2 \epsilon^3} = 9'944'963m^{-2}$$

- $F$ coefficient models turbulent part of pressure drop, and is calculated from

$$F = \frac{3.5(1-\epsilon)}{d_p \epsilon^3} = 3'600m^{-1}$$

Turbulence model used for this simulation was realizable $k$-$\epsilon$. Between all models available (such as $k$-$\omega$, $k$-$\epsilon$-model, Spalart-Allmaras etc) the "realizable" version of two-equation model $k$-$\epsilon$ is the most indicated for this specified case: this model is often used in case of geometrical domain with edge, or sudden variation of area of the geometry section [28]. In order to know if flow inside porous zone is laminar or turbulent, modified Reynolds number $Re_K$ must be calculated

$$Re_K = \frac{uK^{-1/2}}{v_{air,T_{average}}} = 9.81 \Rightarrow Darcy \ regime \ condition$$

How stated, if $Re_K$ is less than 100, flow inside porous zone is laminar: inside the rockbed turbulence is de-activate, setting to zero turbulent viscosity and turbulence generation.

Coupling between pressure and velocity was obtained with PISO algorithm, while interpolation scheme chosen for solving cell-face pressures was PRESTO! algorithm. Between model available in Fluent®, it is the best choice in case
of flows involving steep pressure gradients like in case of porous media. PRESTO discretization for pressure actually calculates pressure on the face, using staggered grids where velocity and pressure variables are not "co-located". All equations were solved by second order discretization and methods. Time-step size is equal to 30 seconds; were set 250 iteration as limit for each time-step.

**Fluent® settings**

<table>
<thead>
<tr>
<th>Fluent® settings</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Time</strong></td>
</tr>
<tr>
<td><strong>Transient Models</strong></td>
</tr>
<tr>
<td>Energy ON</td>
</tr>
<tr>
<td>Gravity ON</td>
</tr>
<tr>
<td><strong>Turbulence model</strong></td>
</tr>
<tr>
<td>Realizable $k - \epsilon$</td>
</tr>
<tr>
<td>Laminar flow in packed bed</td>
</tr>
<tr>
<td><strong>Pressure-velocity coupling</strong></td>
</tr>
<tr>
<td>PISO</td>
</tr>
<tr>
<td><strong>Pressure</strong></td>
</tr>
<tr>
<td>PRESTO!</td>
</tr>
<tr>
<td><strong>Model equation</strong></td>
</tr>
<tr>
<td>2nd order upwind</td>
</tr>
<tr>
<td><strong>Time step size</strong></td>
</tr>
<tr>
<td>30 seconds</td>
</tr>
<tr>
<td><strong>Number of time step</strong></td>
</tr>
<tr>
<td>219</td>
</tr>
<tr>
<td><strong>Max iteration / time steps</strong></td>
</tr>
<tr>
<td>250/step</td>
</tr>
</tbody>
</table>

**Table 6.5**: Fluent® settings used to perform simulations of TES

Under-relaxation factors and residuals are shown in the following table [6.5].

**Fluent® settings**

<table>
<thead>
<tr>
<th>Under relaxation factors</th>
<th>Residuals</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure</td>
<td>$10^{-4}$</td>
</tr>
<tr>
<td>Momentum</td>
<td>$10^{-4}$</td>
</tr>
<tr>
<td>Energy</td>
<td>$10^{-7}$</td>
</tr>
<tr>
<td>$k$ and $\epsilon$</td>
<td>$10^{-4}$</td>
</tr>
<tr>
<td>Density</td>
<td>$10^{-4}$</td>
</tr>
<tr>
<td>Body forces</td>
<td>$10^{-4}$</td>
</tr>
</tbody>
</table>

**Table 6.6**: Fluent® settings used to perform simulations of TES
6.4 Fluent® results

CFD simulation performed using Fluent® give an excellent correspondence as shown in figure 6.5.

Figure 6.5: Comparison between Meier and Fluent® results
Chapter 7

Open\n\n\nFOAM results and comparison

Once validated Fluent ® results, Open\n\n\nFOAM simulation was performed by developed solver. To compare results, the physical properties of the fluid, geometrical domain, computation mesh and boundary conditions were the same imposed in previous simulation. Operative conditions was also the same: a time-dependent simulation was performed, considering energy transfer and gravity effect, and imposing laminar flow inside packed bed.

Pressure-velocity coupling method used in Open\n\n\nFOAM simulation is the above described PIMPLE, insted of the described PRESTO! scheme used by Fluent ®. Another difference between the two simulations is relative to the time-step management: despite of Fluent ® model, where time-step was set equal to 30\textit{seconds}, after some tests, maximum time-step suitable for Open\n\n\n\nFOAM solver was found equal to 2.5\textit{seconds}.

Following table 7.1 shows settings used in order to perform the Open\n\n\nFOAM simulation.

Under relaxation factors and residual are shown in table 7.2. Note that in this case, despite Fluent ® case, must be solved the $p, gh$ term which is pressure term corrected by gravity.
### Open\n\n\n#### Open\n\n\n### Open\n\n\n#### General

<table>
<thead>
<tr>
<th>Time</th>
<th>Transient</th>
<th>Models</th>
<th>Energy ON</th>
<th>Gravity ON</th>
</tr>
</thead>
<tbody>
<tr>
<td>Turbulence model</td>
<td>Realizable $k - \epsilon$</td>
<td>Laminar flow in packed bed</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

#### Solution methods

<table>
<thead>
<tr>
<th>Pressure-velocity coupling</th>
<th>PIMPLE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Momentum predictor</td>
<td>yes</td>
</tr>
<tr>
<td>Correctors</td>
<td>5</td>
</tr>
<tr>
<td>Outer correctors</td>
<td>1</td>
</tr>
<tr>
<td>Time model scheme</td>
<td>Euler</td>
</tr>
</tbody>
</table>

#### Iterative processes

| First step size | $10^{-5}$ seconds |
| Time step size  | adjustable       |
| Maximum Courant number | 20               |

Table 7.1: Open\n\n\n#### Open\n\n\n### Open\n\n\n#### Under relaxation factors

<table>
<thead>
<tr>
<th>Residuals</th>
</tr>
</thead>
<tbody>
<tr>
<td>Velocity</td>
</tr>
<tr>
<td>Hentalpy</td>
</tr>
<tr>
<td>$k$</td>
</tr>
<tr>
<td>$\epsilon$</td>
</tr>
<tr>
<td>$p_g h$</td>
</tr>
</tbody>
</table>

Table 7.2: Open\n\n\n#### Open\n\n\n#### Numerical schemes chosen for the simulation were of the second order accurate, in order to improve quality of simulation, and solved using gaussian quadrature. In

65
particular, time derivative part of equations was solved using a backward, second order, implicit scheme, that permits to not depend on CFL condition. In case of divergence schemes the "limitedLinear" option was used, equal to TVD equations, set to 1, in order to obtain the best convergence, while in case of Laplacian schemes was set "corrected" option to calculate the surface normal. This option corresponds to explicit non-orthogonal correction. This settings are collected in table 7.3.

<table>
<thead>
<tr>
<th>Scheme type</th>
<th>Numerical schemes</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Time derivative</td>
<td>backward</td>
<td>second order, implicit</td>
</tr>
<tr>
<td>Gradient schemes</td>
<td>Gauss linear</td>
<td>second order</td>
</tr>
<tr>
<td>Divergence schemes</td>
<td>Gauss limitedLinear 1</td>
<td>second order</td>
</tr>
<tr>
<td>Laplacian schemes</td>
<td>Gauss linear corrected</td>
<td>second order, conservative</td>
</tr>
<tr>
<td>Interpolation schemes</td>
<td>Gauss linear</td>
<td>second order</td>
</tr>
</tbody>
</table>

Table 7.3: Open∇FOAM settings to perform TES simulations

Results obtained by Open∇FOAM will be presented in next paragraphs. First will be represented velocity contours inside tank, in order to evaluated porosity effect. Turbulence quantities, such as turbulent kinetic energy $k$ and dissipation rate $\epsilon$ are depicted after velocity contours, showing how they are related to advection term, and how they naturally goes down inside the porous zone, justifying the assumption that inside the porous zone flow is laminar: mathematical modeling of laminarization will be exposed.

Thermal diffusion is a consequence of velocity vector and thermal diffusivity, so temperature contours will be shown in last part of this chapter whitin discussion about the relation between temperatures and densities.
7.1 Velocity contours

Velocity magnitude contours are analyzed, measured at different time of the complete simulation. Figure 7.1 depicts velocity contours at 10, 20 and 30 seconds, until flow will become stationary. It can be noted how fluid flow impacts on threshold of rock bed. Hot air at 973 K comes inside the tank from inlet, at 12 m/s, while static air inside the tank is 400 degree colder.

The contact with the packed bed causes to air flow to decrease velocity because of the high pressure losses inside rocks, making more regular the flow inside the packed bed: inside the rock bed velocity vector will have only vertical component different from zero.

![Velocity contours](image)

**Figure 7.1:** Velocity magnitude inside TES at different times

After the first seconds the velocity contours, as visible in figure 7.1, reach a stationary form, like expected: the enlargement of the cross section causes a velocity decrement, while at the edges proximity can be seen recirculating zones. When the fluid flows come into the porous zone, velocity decrease drastically, and the heat exchange happens. Below the porous zone, the fluid flows at a higher velocity than in porous zone because of absence of obstacles, within a section shrinking that forces fluid to flow quicker, in order to satisfy mass conservation law. Velocity contours obtained by Fluent® and OpenFOAM simulations are depicted in figure 7.2. Contours are similar, showing the most relevant differences in the free air...
7.1. VELOCITY CONTOURS

Because of different turbulent wall modeling inside the two solvers: in case of Fluent® the geometrical edge does not cause detaching of the flow. Other differences, can be noticed focusing on the lower surface of the tank, where contours are similar and, like in the upper part, Fluent® does not introduce detaching effects. However, these differences cause negligible effects inside the porous zone. As stated, inside the packbed the flow is laminar so turbulent has no effect on flow motus and heat exchange.

Figures show how fluid velocity modify due to the thermocline crossing: this zone can be easily recognized because of change in value of velocity magnitude, due to different temperature. Where air is still cold, it is denser respect to inlet air, so velocity will be lower than upper thermocline.

Figure 7.2: Velocity contours at 3600s

Figure 7.3 depicts the magnitude of the velocity along the vertical axis. It can be noticed that the different turbulent treatment caused different velocity profile. Due to the detaching effect seen in previous figures, velocity in case of Open√FOAM are higher inside the free air zone, but in the porous zone difference are negligible,
and the thermo cline has got same profile in the case; the pressure drop inside the porous zone is correctly modeled. It can be predicted that wave of hot air will be the same, because its velocity is due to velocity of air inside packed bed.

Vertical velocity contours, depicted in the figure 7.4, are similar to magnitude contours, confirming that most of the velocity magnitude is due to axial component.

Figure 7.3: Contours of velocity magnitude along axis - 3600 seconds

Figure 7.4: Contours of vertical velocity - 3600s
7.1. VELOCITY CONTOURS

Radial velocity contours obtained by the two solvers are depicted in figure 7.5. In these graphs can be noticed that flow direction on the above and the bottom side are in constrast: where the section gets larger, flow goes toward the walls, and where the section becomes smaller, next to the exit, fluid flow become more compact. Differences, as discussed, are due to turbulence modeling.

Inside the porous zone the flow is regular and has got only axial direction.

![Figure 7.5: Contours of radial velocity - 3600s](image)

Recirculating effects, enlightened in the images, are due to porous zone: this zone, as seen in magnitude contours, causes a large decrement in velocity, acting like a wall. So part of the total flow coming must divert its path, spreading out. After porous zone, in bottom free air zone flow goes toward exit creating turbulence on the edge, because of detaching effects due to the edge.
7.2 Turbulence and laminarization

How explained, inside the porous zone the flow is laminar, either at inlet it is turbulent. In order to represent the packed bed a *soft-approach* was used: the "net" effect of laminarization on turbulent viscosity was modeled, instead of put equal to 0 the value of $k$ and $\epsilon$ inside the porous zone. Viscosity values depend on kinetic energy and dissipation ratio values, so results will be different from OpenFOAM respect to Fluent ® because of previously cited detaching effects: in case of Fluent ® model the flow is more regular.

Although in the free air zone viscosity present different contours, inside the porous zone there are no difference between the two solvers, because flow is laminar. In order to represent *laminarization* inside the porous zone, $\mu_t$ was set equal to 0, forcing $\mu_{eff}$ to be equal to the only part due to air properties. Fluent ® solver uses a similar approach, as can be noticed from figure 7.6.

![Effective viscosity $\mu_{eff}$ at 3600s](image)

Figure 7.6: Effective viscosity $\mu_{eff}$ at 3600s
7.3 Thermal conductivity inside the porous zone

Heat transfer is modeled by heat transfer conservation law that depends on turbulence quantities, rocks heat transfer coefficient that modifies thermal diffusivity, and rock thermal capacity that modifies time derivative term. The effective thermal diffusivity inside porous is obtained taking into account the effect of packed bed thermal diffusivity due to the contact area between rocks. The value obtained governs heat transfer inside rock bed, and it is called $k_{eff}$. It is calculated as

$$k_{eff} = \epsilon k_{fluid} + (1 - \epsilon) k_{solid}.$$ 

The figure [7.7](figure) depicts difference between thermal diffusion with inert rocks, and taking into account also packed bed contribution to thermal conductivity. Fluent® and OpenFOAM solver give the same results because of the same numerical model used to evaluate thermal conductivity inside the porous zone. In the figure can be easily recognised the porous zone, that is at the center of the tank, between 0.95m and 3.05m. Also in this case presence of the thermocline make effect on quantities depicted: in fact, can be noted a step in $k_{eff}$ graph, due to different $k_{air}$ at different temperatures. Changing the temperature, also $k_{eff}$ will change because it is dependant on $k_{air}$.

![Figure 7.7: Laminar thermal conductivity along axis 3600 seconds](figure)

Figure 7.7: Laminar thermal conductivity along axis 3600 seconds
7.4 Temperature profiles

Once velocity contours, thermal diffusion terms and attenuating factor value inside the porous zone are noted, temperature profile during simulation can be reasonably predicted.

During the first seconds of simulation the hot air is carried by convective term due to velocity. Diffusion inside free air zone is due only to laminar and turbulent diffusivity of the fluid.

When hot air impacts with surface of rock bed, thermal exchange happens: hot air releases heat to rocks, but heat transfer is slower than in free air zones. As can be seen in figure 7.8, the surface of porous zone acts like a "wall" in which heat cannot penetrate. This permits to saturate upper free air zone before hot air starts heating the porous zone. Note that, nevertheless, fluid flows through rockbed, as shown in previous graphs.

Once upper free air zone is fully saturated, hot temperature starts flowing inside packed bed. As can be predicted, effective wave velocity inside packed bed is proportional to fluid velocity and attenuation term.

Once the porous zone is fully saturated, hot temperature, as predictable, flows toward outlet following velocity vector.
7.4. TEMPERATURE PROFILES

As predicted by Torab and Beasley’s equation in chapter 2, the velocity of the temperature wave inside the porous zone is related to the magnitude of velocity and to the thermal inertia due to the rock heat capacity, and it is constant during all the time of the simulation. Wave’s shape is due to the rock thermal diffusivity that acts on diffusive term of enthalpy equation. The wave-step is directly the thickness of the thermocline. Wave has not a step-shape but presents smooth edges at top and down, due to diffusion.
It must be noted that wave thickness becomes more large at every time step, as visible superimposing two wave at different times. This effect is caused by thermal diffusivity inside the porous zone, that spread out heat during time. As can be seen from graphs, thermocline become more large during time steps.

It is now possible to make a comparison between data available from literature, and what obtained by simulations. Results are shown in graph 7.11 as visible, Open\textcopyright FOAM match test data, in particular in case of temperature wave velocity. As depicted, the center sections of thermocline are matching between experimental results and the two simulations.

Focusing on the wave shape, it can be noted that the Open\textcopyright FOAM solver developed match well the shape, with differences locate on the upper and lower side of the wave. Same effects appears in case of Fluent \textcopyright solver, how depicted more clearly in chapter 6. Wave shape is due to diffusive term, so differences are due to the model used. As explained, was used the in-series model to calculate thermal conductivity in order to make a comparison with Fluent \textcopyright solver, that, in upper part of wave shape, gives results identical to Open\textcopyright FOAM solver.
7.4. TEMPERATURE PROFILES

Figure 7.11: Contours of $T$ along axis - comparison between solvers

Temperature and density contours

The fluid model adopted for this simulation is perfect fluid: density is prop to temperature, how is possible to check in figure 7.12

Figure 7.12: Contours of $T$ and $\rho$ - 3600s
Chapter 8

Conclusions and outlooks

The results obtained by the developed solver rhoPorousHeatPimpleFoam were validated comparing them with data found in literature and with results given by the commercial code Fluent®. Data obtained are almost identical to what calculated by Fluent® simulations, in case of velocity vectors, temperature, and density value, showing differences in turbulent treatment inside and outside the porous zone. Turbulence models in OpenFOAM and Fluent® are not directly comparable, because Fluent® models are more complex, taking into account, for example, gravity effect and near wall treatment. Nevertheless, in case of TES simulations, errors in evaluation of turbulence inside the porous zone can be neglected because it is destroyed by porosity, while outside the porous zone they are of secondary importance because the modeled heat exchange happens inside packed bed.

On these bases, the created soluter could be safely used in future to perform new simulations about TES, considering different tank shapes and dimensions, due to 3-d form, different rockbed properties, and also making tests with a fluid different from air, modeled inside OpenFOAM if not available inside the standard database.

Changing initial conditions and inlet conditions, can be also simulate discharging phase, and complete cycles of charge and discharge, in order to evaluate lack of performances of the tank after different cycles.

It must be notice that all this advantages are related to a simplified version of
TES: in fact, in order to put only the bases to develop a really complete solver, were explored only principal phenomena inside TES, neglecting some aspects that must be developed and correctly modeled in order to use the solver in industry and real cases, that are more complex than what here represented and tested. In particular, the software can be also improved, focusing on the porous zone, modeling heat transfer taking into account radiative heat exchange, a more complex and detailed modeling of turbulence, and modeling heat exchange through the external wall of the tank.

In order to compare results of OpenFOAM to Fluent® results, the effective thermal conductivity was calculated from in serie electrical analogy. As shown in Chapter 2, different formulation were proposed over the years by authors. In this case, OpenFOAM reveals more flexible than Fluent®, because, modifying the porous zone library, can be inserted different forms of thermal conductivity. A possible future investigation can be a systematic and extended test campaign in order to evaluate which form of effective thermal conductivity seems to best represent the real physical phenomena.

The solver that was developed does not take into account radiative term inside the porous zone. As shown in literature review, good results can be obtained also neglecting radiative exchange. It must be noted that also Fluent® numerical models neglect radiative exchange. A deeper and more detailed representation of thermal exchange inside the porous zone will take into account, especially in case of very high temperature of thermovector fluid, radiative contribution that must be correctly modeled. As seen, actually there is no convergence between the formulations proposed: different formulations must be investigated and tested, in order to find the better representation of radiative exchange inside rockbed.

Concerning the exact representation of the turbulence inside the porous zone, in the thesis work was developed the so-called easy way excluding the possibility of a hard way, modifying turbulence values inside dedicated sub-routine, because of good quality of results obtained with the first way. Neither one of these models correctly represent the laminarization process: setting turbulent kinetic energy \( k \) equal to zero inside the porous zone will not represent real phenomena, because rocks and interstices act like turbulent generators. Meanwhile, the roughness of
the rocks destroys the turbulence, so the dissipation rate $\epsilon$ must be very large. If $\epsilon$ is large enough, $\mu_t$ can tend to zero: \[ \lim_{\epsilon \to \infty} \mu_t = C \mu \frac{k^2}{\epsilon} \to 0 \] forcing effective fluid viscosity to be equal to only laminar part. A way to represent laminarization inside the porous zone was considered but not developed: acting on turbulence subroutines, is possible to fix $\epsilon$ value, that is variable and depends on a conservative law, equal to $10^6$. In this case, a new model of turbulent equations would be used, a mixed-form between one equation model, and two equation model: turbulent kinetic energy would be calculated solving conservative equation for $k$, although $\epsilon$ is fixed. The use of this artifice will bring to a turbulent viscosity equal to zero, making velocity conservation law inside the porous zone as laminar version.

Heat exchange through the wall of tank in this work was not considered, because validation test used adiabatic walls, like Fluent® simulations. Real TES don’t have adiabatic walls, but usually are put into the ground, in order to make them more isolated as possible: adiabatic boundary conditions is a simplification not too far from reality. In case of not-perfectly isolated TES, it is possible to simulate the heat exchange between TES and surrounding acting directly on boundary walls, or representing them in computational domain in order to correctly model wall insulating in case of complex geometries. OpenFOAM, through not standard dedicated subroutine, can offer the opportunity to impose complex boundary condition on domain boundary: it is possible to impose a heat through the walls, thermal insulating properties and external reference temperature.

All this suggestions, once will be investigated, correctly modeled and implemented, can bring to a complete open source TES simulation software, which will be able to compare with commercial code in terms of results reliability, computational costs, flexibility and completeness.
Bibliography


BIBLIOGRAPHY


