NUMERICAL ANALYSIS OF HEAT TRANSFER AND FLUID FLOW IN HEAT EXCHANGERS WITH EMPHASIS ON PIN FIN TECHNOLOGY

Hamid Nabati

2012
NUMERICAL ANALYSIS OF HEAT TRANSFER AND FLUID FLOW IN HEAT EXCHANGERS WITH EMPHASIS ON PIN FIN TECHNOLOGY

Hamid Nabati

Akademisk avhandling

som för avläggande av teknologidoktorsexamen i energi- och miljöteknik vid Akademin för hållbar samhälls- och teknikutveckling kommer att officiellt försvaras fredagen den 13 april 2012, 10.00 i Delta, Västerås.

Fakultetsopponent: professorn Dan Loyd, Linköping University
Abstract

One of the most important industrial processes is heat transfer, carried out by heat exchangers in single and multiphase flow applications. Despite the existence of well-developed theoretical models for different heat transfer mechanisms, the expanding need for industrial applications requiring the design and optimization of heat exchangers, has created a solid demand for experimental work and effort. This thesis concerns the use of numerical approaches to analyze and optimize heat transfer and fluid flow in power generation industry, with emphasis on pin fin technology.

This research begins with a review on heat transfer characteristics in surfaces with pin fins. Different pin fins shapes with various flow boundaries were studied, and thermal and hydraulic performances were investigated. The impact of parameters such as inlet boundary conditions, pin fin shapes, and duct cross-section characteristics on both flow and heat transfer were examined. Two important applications in power generation industry were considered for this study: power transformer cooling, and condenser for CO2 capturing application in oxy-fuel power plants. Available experimental data and correlations in the literature have been used for models validation. For each case, a model based on current configuration was built and verified, and was then used for optimization and new design suggestions.

All numerical modeling was performed using commercial CFD software. A basic condenser design was suggested and examined, supplemented by the use of pin fin technology to influence the condensation rate of water vapour from a CO2/H2O flue gas flow. Moreover an extensive review of numerical modeling approaches concerning this condensation issue was conducted and presented.

The analysis results show that the drop-shaped pin fin configuration has heat transfer rates approximating those of the circular pin configuration, and the drop-shaped pressure losses are less than one third those of the circular. Results for the power transformer cooling system show those geometrical defects in the existing system are easily found using modeling. Also, it was found that the installation of pin fins in an internal cooling passage can have the same effect as doubling the radiator’s height, which means a more compact cooling system could be designed.

Results show that a condensation model based on boundary layer theory gives a close value to experimental correlations. Considering a constant wall temperature, any increase in CO2 concentration results in lower heat transfer coefficients. This is a subsequence of increased diffusivity resistance between combustion gas and condensing boundary layer. Also it was shown that sensitivity of heat transfer rate to inlet temperatures and velocity values decreased when these parameters increased.

The application of numerical methods concerning the condensation process for CO2 capturing required significant effort and running time as the complexity of multiphase flow was involved. Also data validation for the CO2/H2O condenser was challenging since this is quite a new application and less experimental data (and theoretical correlations) exist. However, it is shown that models based on numerical approaches are capable of predicting trends in the condensation process as well as the effect of the non-condensable CO2 presence in the flue gas.

The resulting data, conclusions, applied methodology can be applied to the design and optimization of similar industrial heat exchangers, such as oil coolers which are currently working at low efficiency levels. It can also be used in the design of electronic components, cooling of turbine blades, or in other design applications requiring high heat flux dissipation. Finally, the finding on water vapour condensation from a binary mixture gas can be referenced for further research and development in this field.
"Education is a progressive discovery of our own ignorance."

- Will Durant
Abstract

One of the most important industrial processes is heat transfer, carried out by heat exchangers in single and multiphase flow applications. Despite the existence of well-developed theoretical models for different heat transfer mechanisms, the expanding need for industrial applications requiring the design and optimization of heat exchangers, has created a solid demand for experimental work and effort. This thesis concerns the use of numerical approaches to analyze and optimize heat transfer and fluid flow in power generation industry, with emphasis on pin fin technology.

This research begins with a review on heat transfer characteristics in surfaces with pin fins. Different pin fins shapes with various flow boundaries were studied, and thermal and hydraulic performances were investigated. The impact of parameters such as inlet boundary conditions, pin fin shapes, and duct cross-section characteristics on both flow and heat transfer were examined. Two important applications in power generation industry were considered for this study: power transformer cooling, and condenser for CO₂ capturing application in oxy-fuel power plants. Available experimental data and correlations in the literature have been used for models validation. For each case, a model based on current configuration was built and verified, and was then used for optimization and new design suggestions. All numerical modeling was performed using commercial CFD software. A basic condenser design was suggested and examined, supplemented by the use of pin fin technology to influence the condensation rate of water vapour from a CO₂/H₂O flue gas flow. Moreover an extensive review of numerical modeling approaches concerning this condensation issue was conducted and presented.

The analysis results show that the drop-shaped pin fin configuration has heat transfer rates approximating those of the circular pin configuration, and the drop-shaped pressure losses are less than one third those of the circular. Results for the power transformer cooling system show those geometrical defects in the existing system are easily found using modeling. Also, it was found that the installation of pin fins in an internal cooling passage can have the same effect as doubling the radiator’s height, which means a more compact cooling system could be designed.

Results show that a condensation model based on boundary layer theory gives a close value to experimental correlations. Considering a constant wall temperature, any increase in CO₂ concentration results in lower heat transfer coefficients. This is a subsequence of increased diffusivity resistance between combustion gas and condensing boundary layer. Also it was shown that sensitivity of heat transfer rate to inlet temperatures and velocity values decreased when these parameters increased.

The application of numerical methods concerning the condensation process for CO₂ capturing required significant effort and running time as the complexity of multiphase flow was involved. Also data validation for the CO₂/H₂O condenser was challenging since this is quite a new application and less experimental data (and theoretical correlations) exist. However, it is shown that models based on numerical approaches are capable of predicting trends in the condensation process as well as the effect of the non-condensable CO₂ presence in the flue gas.

The resulting data, conclusions, applied methodology can be applied to the design and optimization of similar industrial heat exchangers, such as oil coolers which are currently working at low efficiency levels. It can also be used in the design of electronic components, cooling of turbine blades, or in other design applications requiring high heat flux dissipation. Finally, the finding on water vapour condensation from a binary mixture gas can be referenced for further research and development in this field.

Keywords: CFD, Heat exchanger, Pin fin technology, Cooling system, Power transformers, CO₂ capturing, Two phase flows.
Svensk sammanfattning

En av de viktigaste industriella processerna är värmeöverföring utförd av värmeväxlare i applikationer med en- och flerfasströmning. Trots välutvecklade teoretiska modeller för olika värmeöverföringsmekanismer finns det fortfarande många industriella tillämpningar där design och optimering av tillgängliga eller nya värmeväxlare kräver mycket experimentellt arbete. Denna avhandling fokuserar på användningen av numeriska metoder för att analysera och optimera vätskeflödet och värmeöverföring inom energiproduktionen med tonvikt på pin fin teknik.

Forskningsarbetet började med undersökningen av flöde och värmeöverföring i pin fin ytor. För att ge rekommendationer om optimerad geometri för värmeväxlare studerades olika pin fins former med varierande flödesgränser och termiska och hydrauliska prestanda undersöktes.


Tillämpning av numeriska metoder i kondenseringsprocesser för koldioxiduppfångning kräver mycket mer arbete och körtid jämfört med mindre experimentell data och färre teoretiska samband tillgängliga. Det visade sig emellertid att modeller baserade på numeriska metoder kan förutsäga trenden i kondenseringsprocessen och effekten av koldioxiens närvaro i rökgasen som en icke-kondenserbar gas.

Resultatet och slutsatserna från den aktuella studien och den använda metoden kan tillämpas inom design och optimering av liknande värmeväxlare inom industrin, till exempel oljekylare som för närvarande arbetar med låg kylningseffektivitet. Det kan även användas vid konstruktion av elektroniska komponenter, turbindeladskylnings och andra tillämpningar som kräver snabb värmespridning. Slutligen, resultatet på vattenånga kondens från en binär blandning gas skulle kunna användas för vidare forskning och utveckling inom detta område.
Acknowledgements

This thesis has been carried out at the School of Sustainable Development of Society and Technology, Mälardalen University, Västerås, Sweden as doctoral thesis.

I would like to express my most sincere gratitude to my academic supervisor, Professor Jafar Mahmoudi who encouraged me to start this work and patiently supported me in many different aspects throughout this work. Also my special appreciation goes to Professor Erik Dahlquist for his useful lectures and many hours of discussions on optimization, reviewing and ideas in articles and research methodologies. He always supported me like a close friend and gave me useful advices. I would like also to thank my co-supervisor Professor Jinyue Yan for his encouraging and stimulating guidance during study period and all supports in general. He opened new views to research field for me.

In addition to the above academic contribution, there are other persons in our department that I would like to thank for their supports and collaboration: Benny Ekman, Bengt Arnryd, Adel Karim, Robert Öman and Jan Sandberg. Your efforts in conducting me through different courses and also academic life are highly appreciated.

Special thanks to my friends and colleagues at MDU; Christer Karlsson, Anders Avelin, Weilong Wang, Keramatollah Akbari, Anna Paz, Lilia Daianova, Johan Lindmark and Adrian Rodriguez for all the friendship that we had together.

I also wish to thank my wife Narges for her patience and support and also my parents for continuously encouraging me. I always appreciate your supports. My lovely daughter, Armita has been the light of my way.

Without help of you all, I would never have been able to finish my degree. Thanks again.
List of appended publications

Publications included in the thesis:
This thesis is based on the following papers:


Publications not included in the thesis:

# Nomenclature

## Latin Letters

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>surface area [m²]</td>
</tr>
<tr>
<td>( C_p )</td>
<td>specific heat capacity [J/kg.K]</td>
</tr>
<tr>
<td>D</td>
<td>circular and drop-shaped fin diameter, Rectangular fin width [m]</td>
</tr>
<tr>
<td>( D_h )</td>
<td>hydraulic diameter [m]</td>
</tr>
<tr>
<td>( D_P )</td>
<td>pressure drop [Pa]</td>
</tr>
<tr>
<td>f</td>
<td>friction coefficient (Total pressure loss coefficient)</td>
</tr>
<tr>
<td>g</td>
<td>gravity acceleration [m²/s²]</td>
</tr>
<tr>
<td>Gr</td>
<td>Grashof number ( = \frac{g \beta (T_s - T_\infty) L_c^3}{\nu^2} )</td>
</tr>
<tr>
<td>h</td>
<td>convective heat transfer coefficient [W/m².K]</td>
</tr>
<tr>
<td>H</td>
<td>pin fin height [m]</td>
</tr>
<tr>
<td>i</td>
<td>internal energy [J]</td>
</tr>
<tr>
<td>k</td>
<td>turbulent kinetic energy [m²/s²]</td>
</tr>
<tr>
<td>( L_c )</td>
<td>characteristic length of the geometry [m]</td>
</tr>
<tr>
<td>( \dot{m} )</td>
<td>mass flow rate [kg/s]</td>
</tr>
<tr>
<td>Nu_D</td>
<td>Nusselt number based on ( D_h )</td>
</tr>
<tr>
<td>P</td>
<td>pressure [Pa]</td>
</tr>
<tr>
<td>Pr</td>
<td>Prandtl number</td>
</tr>
<tr>
<td>( Q, q )</td>
<td>rate of heat transfer [W]</td>
</tr>
<tr>
<td>R</td>
<td>thermal resistance [K/W]</td>
</tr>
<tr>
<td>Re_D</td>
<td>Reynolds number based on D</td>
</tr>
<tr>
<td>S,S</td>
<td>Surface</td>
</tr>
<tr>
<td>T</td>
<td>temperature [K]</td>
</tr>
<tr>
<td>( T_H )</td>
<td>heat source temperature [K]</td>
</tr>
<tr>
<td>u,U, ( \vec{U} )</td>
<td>velocity [m/s]</td>
</tr>
</tbody>
</table>

## Greek Letters

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \alpha )</td>
<td>thermal diffusivity [m²/s]</td>
</tr>
<tr>
<td>( \beta )</td>
<td>coefficient of volume expansion [1/K]</td>
</tr>
<tr>
<td>( \varepsilon )</td>
<td>turbulent dissipation rate [m²/s³]</td>
</tr>
<tr>
<td>( \varepsilon_f )</td>
<td>fin effectiveness</td>
</tr>
<tr>
<td>( \eta_f )</td>
<td>fin efficiency</td>
</tr>
<tr>
<td>( \eta_o )</td>
<td>overall efficiency</td>
</tr>
<tr>
<td>( \lambda )</td>
<td>thermal conductivity [W/m.K]</td>
</tr>
<tr>
<td>( \mu )</td>
<td>molecular viscosity [Pa.s]</td>
</tr>
<tr>
<td>( \mu_t )</td>
<td>eddy viscosity [Pa.s]</td>
</tr>
<tr>
<td>( \infty )</td>
<td>fluid bulk condition</td>
</tr>
<tr>
<td>( \nu )</td>
<td>kinematics viscosity [m²/s]</td>
</tr>
<tr>
<td>( \Theta )</td>
<td>non-dimensional temperature</td>
</tr>
<tr>
<td>( \theta )</td>
<td>mean temperature [K]</td>
</tr>
<tr>
<td>( \rho )</td>
<td>density [kg/m³]</td>
</tr>
</tbody>
</table>
List of figures

Figure 1-1: Typical power transformer equipped with auxiliary fans (Iran Transfo Corporation Documents, 2005).......................................................... 4
Figure 1-2: Some of the many varieties of finned tubes (Lienhard IV, et al., 2005)......................... 4
Figure 1-3: Direct contribution of greenhouse gases to climate change (IEA Greenhouse Gas R&D website, 2008)................................................................. 6
Figure 1-4: The main steps in CO₂ capturing processes................................................................. 7
Figure 1-5: The main technologies for CO₂ capturing. .................................................................. 7
Figure 1-6: Schematic diagrams of different heat sinks for tests (Christopher, et al., Feb. 2000) .... 9
Figure 1-7: Heat sink with in-line pin fins array (Moshfegh, et al., 2004). .................................... 10
Figure 2-1: Analysis of one-dimensional fin (Lienhard IV, et al., 2005) ........................................ 14
Figure 2-2: Heat exchanger domain. .......................................................................................... 17
Figure 2-3: Studied fins: Drop-Shaped, Rectangular, and Cylindrical........................................ 17
Figure 3-1: Comparison of Nusselt number in CFD modeling and theoretical model............... 22
Figure 3-2: Film condensation on a vertical plate. ..................................................................... 24
Figure 3-3: Water vapour condensation on a smooth vertical plane in presence of non-condensable gas......................................................................................... 25
Figure 4-1: Computational domain including the generic fin forms............................................ 31
Figure 4-2: Nusselt number versus Reynolds number................................................................. 32
Figure 4-3: Friction factor.......................................................................................................... 33
Figure 4-4: Velocity contour (ReD = 10000, Horizontal Surface, Y=75 mm) .............................. 33
Figure 4-5: Temperature distribution, Horizontal surface (ReD =1000, Y = Cte) .................... 35
Figure 4-6: Temperature distribution, Vertical surface (ReD =1000, X= Cte) ............................ 35
Figure 4-7: Temperature distribution, First pin (ReD =1000, Z = Cte)....................................... 35
Figure 4-8: Temperature distribution, last pin (ReD = 1000, Z = Cte). ...................................... 35
Figure 4-9: Temperature distribution, Horizontal surface (ReD = 50000, Y = Cte) ................ 36
Figure 4-10: Temperature distribution, Vertical surface (ReD = 50000, Y = Cte) ................ 36
Figure 4-11: Temperature distribution, First pin (ReD5 = 50000, Z = Cte)............................ 36
Figure 4-12: Temperature distribution, last pin (ReD = 50000, Z = Cte)................................. 36
Figure 4-13: Total pressure distribution, Outlet (ReD0 = 50000, Z = Cte)............................... 37
Figure 4-14: Simulated radiator block....................................................................................... 38
Figure 4-15: Temperature contour on the outer surface of radiator.......................................... 38
Figure 4-16: Temperature contour of oil (°C) in the vertical middle surface of radiators block... 39
Figure 4-17: Contour of velocity vector in the vertical middle surface of radiators block. ......... 39
Figure 4-18: Pressure distribution in the inlet header............................................................... 40
Figure 4-19: Pressure distribution in the outlet header............................................................. 40
Figure 4-20: Radiator elements & radiator model. ................................................................. 41
Figure 4-21: Verification of model validity.................................................................................. 41
Figure 4-22: Oil flow rate at different inlet temperatures............................................................ 42
Figure 4-23: Average oil velocity at different inlet temperatures ................................................. 42
Figure 4-24: Effect of pin fin on oil mass flow rate................................................................. 42
Figure 4-25: Oil outlet temperature with pin fin radiator........................................................... 42
Figure 4-26: Comparison of heat transfer for different cases...................................................... 43
Figure 4-27: Schematic of oxy-fuel process (Vattenfall, 2009) and considered model for condenser. 43
Figure 4-28: Comparing model results with experimental data (ΔTtw-g = 18°C, P = 1 bar) .... 44
Figure 4-29: Average heat transfer coefficient versus inlet velocity and CO₂ mass fraction...... 44
Figure 4-30: Condensation rate as a function of inlet velocity and inlet temperature............. 44
Figure 4-31: Verification of model validity (air/water vapour mixture) ...................................... 45
Figure 4-32: Comparison of condensation rate for two condensation models.......................... 45
Figure 4-33: Effect of inlet velocity on total heat transfer coefficient. ............................................... 46
Figure 4-34: Comparison of total heat transfer coefficient between simple case and pin-finned surface. ............................................................................................................................ 46
List of tables

Table 1-1: CO₂ emission share of large industrials activities (Metz, et al., 2006) ......................... 6
Table 3-1: Comparison of different turbulence modeling (Moshfeh, et al., 2004) .................... 21
Table 3-2: Test configuration ........................................................................................................... 23
Table 3-3: Required data for mass diffusivity calculation (Lienhard IV, et al., 2005) ................. 26
Contents

1. Introduction ......................................................................................................................................... 3
   1.1 Flow and heat transfer modeling solution for industry .............................................................. 3
   1.2 Background .................................................................................................................................... 3
   1.3 Motivation ....................................................................................................................................... 8
   1.4 Literature review ............................................................................................................................ 8
   1.5 Objectives ...................................................................................................................................... 12

2. Pin Fin Technology and Modeling .................................................................................................... 14
   2.1 Basic fin theory and definitions ................................................................................................... 14
   2.2 Pin fin study ................................................................................................................................. 17

3. Governing equations and numerical solution procedure ............................................................... 18
   3.1 Introduction ................................................................................................................................... 18
   3.2 Governing equations of fluid flow and heat transfer .................................................................... 18
   3.3 The instantaneous equations ........................................................................................................ 19
   3.4 Simulation of pin fin model ........................................................................................................... 21
      3.4.1 Numerical mesh ....................................................................................................................... 21
      3.4.2 Solution technique .................................................................................................................. 22
      3.4.3 Model validation ...................................................................................................................... 22
      3.4.4 Test approach ........................................................................................................................ 23
      3.4.4.1 Test matrix .......................................................................................................................... 23
      3.4.4.2 Data analysis ....................................................................................................................... 23
   3.5 Two-phase flow theory and modeling ........................................................................................... 23
      3.5.1 Surface condensation theory ................................................................................................. 23
      3.5.2 Physical model for binary system condensation .................................................................... 25
      3.5.3 Mass transfer considerations ................................................................................................. 25
      3.5.4 Influence of non-condensable gases on condensation ............................................................ 27
      3.5.5 Numerical modeling of two-phase flow ................................................................................... 27
         3.5.5.1 Comparison of different approaches for multiphase modeling ........................................... 27
      3.5.6 Governing equations .............................................................................................................. 28
      3.5.7 Continuity equation ............................................................................................................... 28
      3.5.8 Momentum equation .............................................................................................................. 29
      3.5.9 Multiphase species transport equation .................................................................................... 29
      3.5.10 Energy equation .................................................................................................................... 29

4. Results and Discussion ...................................................................................................................... 31
   4.1 Introduction ................................................................................................................................... 31
   4.2 Pin fin morphologies .................................................................................................................... 31
      4.2.1 Nusselt number ....................................................................................................................... 31
      4.2.2 Friction factor ........................................................................................................................ 32
      4.2.3 Heat transfer .......................................................................................................................... 34
   4.3 Industrial application in power transformers cooling ................................................................. 37
      4.3.1 Power transformer cooling system ......................................................................................... 37
         4.3.1.1 Radiators block .................................................................................................................. 37
1. Introduction

1.1 Flow and heat transfer modeling solution for industry

Continuous and rapid technological advances in industrial processing require that design and operation problems be resolved as quickly as possible in order to keep companies competitive, particularly in terms of energy efficiency and low costs. For many years, experiments and empirical analysis have been the preferred solution tools for industrial analysis. Despite the robust and reliable nature of experimental methodology, certain factors limit its applicability scope. For example, flows in process installations are usually very complex; the use of experimental methodology in related analysis may demand significant simplification or a vast number of experiments to achieve an acceptable solution, indicating both cost and time constraints. Thus it has become a necessity to use advanced modeling and simulation tools in industry, and the number of industries benefitting from these products continues to expand.

Computational fluid dynamics (CFD) is a computer simulation technique used for fluid flow and heat transfer modeling. Based on increasingly powerful computer resources, CFD can be applied to solve industrial flow and complex phenomenon problems. However, there still exists a lack of data for CFD applications in different industrial areas enabling the development of general guidelines in specific numerical heat and flow studies. This thesis attempts to provide CFD implementation methods and data appropriate for those varied industrial applications.

1.2 Background

To describe the behavior of flow dynamics, governing mathematical equations are solved numerically. This is what is known as computational simulation. Results of this simulation include the approximated velocity field and distribution of temperature and pressure in the entire flow domain. Related physical properties (e.g., temperature profiles and density) can be extracted easily from the modeling. The intent of much current researches is to improve the heat and mass transfer models and to extend the range of chemical and physical models in CFD codes for application in industrial problems.

Fluid flows and heat transfer play critical roles in industrial processing. For example, air or water flows are regularly used for cooling purposes. Improving the cooling performance requires insight into the cooling flow profile. Typically, flow information is acquired by measuring in experimental test facilities or conducting flow visualization studies. Both methods have limitations, and it is not always an easy job to get all flow parameters. Numerical methods accompanied with new modern and high speed computers have recently developed techniques to remove these limitations.

There are myriad cases in industry that can be studied and optimized using numerical simulations. Many processes contain fluid flows having multiple phases or component mixtures, all of which must be included in the simulation. For example, motion of bubbles or droplets in a fluid, mixing vessels, heat exchangers, furnaces, or HVAC equipment are typical cases that can be studied more easily by computational simulation. In turbines, fans or any other applications that contain moving parts, the numerical simulation can be implemented successfully, if correct conditions like time-variation of flow geometries are considered. In such cases, usually what is important to be calculated is transient flow field. In this thesis, numerical methods are applied in different industrial applications including: Power transformers cooling system and CO\textsubscript{2}/H\textsubscript{2}O condenser in an oxy-fuel process.
Transformers are widely used in industry, especially in power distribution networks where they are valuable assets and it is obvious that they require special attention to achieve reliable stability in power systems. Iran Transfo Corporation (ITC) is a leading Iranian manufacturer of single and three-phase oil-immersed distribution transformers that produces distribution transformer with a capacity of 12000 MVA and a power rating between 25 and 5000 kVA and a voltage ranging up to 36 kV. The energy losses in these power transformers are proportional to their loads. These losses that are typically heat losses cause temperature increment to high levels and subsequently performance of transformer decreases. Eventually its operational load capacity and its useful life are reduced. In small transformers heat removal through the unit’s own surface by natural convection is typically enough. However, this kind of heat removal is not enough for transformers of average and high power, which require more elaborate methods of cooling and generated heat removal (Carrothers, et al., 1961). In the ITC transformers, cooling is provided by the circulation of oil between ducts in the active parts and heat exchangers outside the transformer tank; this oil circulation results from free convection, or combined free and forced convection. Based on transformer design, it is in the interest of ITC to minimize the size of heat exchangers and internal cooling ducts.

In the literature, few studies exist on heat transfer and fluid flow outside the transformer; most have focused on processes inside the transformer. For example, Mufuta and Van Den Bulck (Mufuta, et al., 2000) studied a winding disc-type transformer. Results showed that there is a strong relation between Re.Gr⁻¹/₂ parameter and flow behavior. They presented some correlations for the heat transfer calculation inside the transformer. Heat transfer inside channels and the thermal behavior of flow in a pipe with different boundary conditions have been investigated widely in heat transfer texts, but no research with focus on the radiator has been found in the literature.

A typical transformer is shown in Figure 1-1. To achieve better performance, conventionally fins are added to plain tubes resulting in a larger external surface area, a higher heat transfer performance, and smaller dimensions. Figure 1-2 shows some various fins commonly used in heat exchangers.

Figure 1-1: Typical power transformer equipped with auxiliary fans (Iran Transfo Corporation Documents, 2005).

Figure 1-2: Some of the many varieties of finned tubes (Lienhard IV, et al., 2005).
One of the most interesting developments in enhanced heat exchanger design is in pin fin technology. Pin fins are proper solution in heat transfer enhancement in various applications including gases or liquids. They are manufactured from various materials including brass, copper, carbon steel, aluminum/brass, and other metal alloy. In recent years, this technology has become a major design component for electronics cooling. Its excellent capacity for heat transfer logically extends its use from electronics to industrial heat exchangers. This technology could remove or at least decrease most of the problems incorporated with heat exchangers operation: large pressure loss and low total heat transfer efficiency. Pin fin structure significantly increases the heat transfer surface area at both side of tube, with a reduced thickness of the boundary layers. Some of its features are presented below (Gree Thermo-Tech Co., Ltd. of Zhuhai, 2005):

- Convection heat exchanging coefficient for single-phase fluid is 2.5 to 6 times higher than that of the plane tube.
- Boiling heat transfer coefficient is 2 to 5 times higher than that of the plane tube.
- Condensing heat transfer coefficient is 3 to 6 times higher than that of the plane tube.
- Unique self-cleaning feature greatly reduces maintenance cost of heat exchangers."

Few companies produce pin fin tubes for use in heat exchangers' applications. Gree Thermo-Tech Co., Ltd. in China is a major producer and recommends them for use in different types of heat exchangers. However, no technical data are currently available on that company's web page (perhaps because the technology is still cutting edge and, therefore, highly guarded). Another company which uses this technology in its products is Raypak Company in the USA. The pin fin tubes are widely used in the company's heating products. It seems possible that the study of pin fin's applicability in heat exchangers could generate attractive solutions to challenges in heat exchangers' field and produce more efficient and compact heat exchangers.

The pulp and paper industry may present a strong potential for using this technology. The industry consumes a sizeable portion of primary energy consumed in the industrialized countries' manufacturing. The heart of the pulp and paper mill is its recovery boiler, which consists of several heat transfer components including a super heater, a reheater, an economizer, an evaporator and air heaters. Super heaters (especially the radiant types) are made from smooth tubes, which are less expensive and simpler to manufacture than finned ones; they are also less prone to fouling and can be cleaned easily. The drawback to their use is that in a moderate gas flow velocity, the heat absorption is limited. Importantly, heat transfer through tubular surfaces is restricted by the heat passing through the tube's external surface and therefore can be increased by extending (expanding) that surface. In light of the new cooling technologies used to improve energy efficiency, surface enhancement using pin fins could be a promising method to surmount the described problem as well as improve efficiency and reduce maintenance. Moreover, there are other heat exchangers used in the pulp and paper mills (for example in the drying section) which can operate with higher efficiency using enhanced heat transfer surfaces.

Another emerging issue that may be resolved through technological advances concerns greenhouse gases effect and global warming. When sunlight passes through the atmosphere and reaches to the earth’s surface, it is converted to heat. Some of this heat reflects directly to space in the form of infrared radiation and some is absorbed by greenhouse gases. This absorption is a natural phenomenon which helps keep the temperature of the earth at a level that makes it possible for life to exist. It appears that human activities have caused an increase in the level of greenhouse gases and, consequently, the global temperature has risen. It is so important to limit this change, so natural and human systems can adapt themselves to this change. One of the most important ways to control the rate of global temperature change is to reduce greenhouse gases in the atmosphere and thereby decrease their heat absorption.

The main greenhouse gases that contribute to global temperature rising are: carbon dioxide (CO₂), methane (CH₄), nitrous oxide (N₂O), perfluorocarbons (PFC’s), halo carbons (HFC’s), and sulphur hexafluoride (SF6). Figure 1-3 shows the direct contribution each. Among these, the main anthropogenic gas is CO₂. Approximately one-third of all CO₂ emissions come directly from electricity production.
sections that use fossil fuel combustion to generate electricity. Other industrial processes (including oil refineries, cement factories and steel producers) generate 40% of these emissions and the reminder comes from mobility and other sources. The CO₂ emissions from electricity production and other industrial processes could be substantially reduced by capturing and storing the CO₂, without major changes to main process. Table 1-1 shows CO₂ emissions from different major industrial resources. From this data, it is fairly obvious that carbon capturing and sequestration in stationary sources such as fossil fuel power plants is one of the important solution in reducing the CO₂ presence in the atmosphere.

![Figure 1-3: Direct contribution of greenhouse gases to climate change (IEA Greenhouse Gas R&D website, 2008).](image)

Various methods exist for reducing greenhouse gas emission such as reducing energy demand, improving energy efficiency, and switching to fuels with low or no carbon contents. These methods are good but insufficient. Since the world is at present heavily dependent on the exploitation and use of fossil fuels, it is important to focus on technologies for the capture and storage of CO₂ produced by the combustion of fossil fuels. Figure 1-4 shows the main steps in the CO₂ capturing process in a fossil-fuel power plant. First the fossil fuel is combusted to produce power; and then CO₂ is separated from combustion products. Finally depending on available facilities, the CO₂ is either used or stored.

Available technologies in CO₂-capturing are categorized into three main groups (Davison, et al., 2001):

- **Post-combustion**: CO₂ is captured from combustion products
- **Pre-combustion**: CO₂ is removed from the fossil fuel before combustion process
- **Oxy-fuel**: pure oxygen is used for fossil fuel combustion

**Table 1-1: CO₂ emission share of large industrials activities (Metz, et al., 2006).**

<table>
<thead>
<tr>
<th>Process</th>
<th>Emissions (MtCO₂ per year)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fossil fuels</td>
<td></td>
</tr>
<tr>
<td>Power</td>
<td>10.539</td>
</tr>
<tr>
<td>Cement production</td>
<td>932</td>
</tr>
<tr>
<td>Refineries</td>
<td>798</td>
</tr>
<tr>
<td>Iron and steel industry</td>
<td>646</td>
</tr>
<tr>
<td>Petrochemical industry</td>
<td>379</td>
</tr>
<tr>
<td>Oil and gas processing</td>
<td>50</td>
</tr>
<tr>
<td>Other sources</td>
<td>33</td>
</tr>
<tr>
<td>Biomass</td>
<td></td>
</tr>
<tr>
<td>Bioethanol and bioenergy</td>
<td>91</td>
</tr>
</tbody>
</table>
Figure 1-4: The main steps in CO₂ capturing processes.

Figure 1-5 shows the main technologies which are used for CO₂ capturing in power plants. The technical and economic comparison of the three technologies for large-scale industrial applications (like power plants) is still under study and a preferable technology may highly depend on further development and commercialization of the technologies.

This work considers the third technique, involving oxy-fuel combustion. Oxy-fuel systems use pure oxygen for fuel combustion, and the resulting flue gas contains mainly water vapour and CO₂. The CO₂ concentration of flue gas is high (greater than 80% by volume) and the water vapour content must be removed by condensation process in a CO₂/H₂O condenser. This technology is not fully mature. The operation and management as well as capital costs would be comparable to post-combustion capture; in specific, the oxygen separation unit consumes about 23%–37% of the total plant output and costs about the same as the absorber. To gain the high efficiency in H₂O removal from flue gas and to prepare a pure CO₂ stream, a precise condenser design is required. Despite wide usage of condensers in different industries, complete design data and codes are not yet available, especially for condensers that involve multi-component mediums. Thus new experimental and numerical studies are necessary to achieve the desired efficiency in CO₂ capturing and steam separation. Significant technical challenges regarding the condenser design must first be addressed before this method can be implemented. This option is most appropriate for new plant projects rather than for retrofitting existing plants. The main effort in the current project deals with the technical challenges of condenser design for CO₂ capturing using the oxy-fuel method.

Figure 1-5: The main technologies for CO₂ capturing.
1.3 Motivation

Compact heat exchangers (CHE) have recently become subject to extensive research because of their importance to a wide variety of engineering applications. Fins play a vital role in enhancing their performance.

Radiators used in power transformers are typical heat exchangers. However, their performance is not sufficient to resolve the overload conditions that take place during the hot seasons; therefore, it may be advantageous to use pin fin technology either inside or outside these exchangers to enhance their efficiency. Studies seeking to improve the heat transfer characteristics of radiators could involve changing the oil flowing passage profile, changing the radiator dimensions, or using width-wise pin fins between radiators, all of which require a solid understanding of pin fins behaviour and their heat transfer characteristics.

Large numbers of different heat exchangers are used in vital industries like power plants, power transformers, and pulp and paper mills, especially in their steam boilers—the heart of the industrial plant. The efficiency of the gas-side heat transfer is a primary consideration when determining the best heat exchanger for a particular application. Surface enhancement using pin fins could be a promising method to overcome the described problem as well as to improve maintenance requirements. As yet, no detailed study on the performance of such finned tubes has been carried out; the present research investigates the heat transfer characteristics of different shaped pin fins from numerical point of view.

The increase in greenhouse gases has caused the global temperature to rise and will eventually cause the global climate to change. CO$_2$ is the main anthropogenic gas, derived from fossil fuels for use in industry, and carbon capture and sequestration to reduce its release into the atmosphere is of critical importance. The power plant’s flue gas contains high CO$_2$ concentrations (greater than 80% by volume). Therefore, there is a significant need for an efficiently designed CO$_2$/H$_2$O condenser to capture this amount of CO$_2$. Despite of wide referring to this kind of condenser in different proposed Oxy fuel cycles; the appropriate design data are not yet available. New experimental and numerical studies are necessary to achieve the desired efficiency for CO$_2$ capturing and steam separation. Basic studies on water vapour condensation in presence of CO$_2$ as non-condensable gas can provide useful technical data for designers.

There are still many emerging concepts such as CO$_2$ capturing technologies that are being developed and assessed throughout the world and they need efficient tools to be evaluated carefully. CFD can be used as a robust tool for simulating the conceptual design in such projects before manufacturing and operating experiments, reducing the total effort required in experimental design and data acquisition. This means that the experiments can be performed once the design is more mature, resulting in the saving of both time and money. Technical efforts and practical bottlenecks in mathematical modelling may remain challenges; this thesis will focus on reducing some of them.

1.4 Literature review

This section presents a brief look at the existing research. It also includes a discussion of current state-of-the-art issues and optimization techniques involved with thermal management in heat exchangers and e-cooling. In addition, a comprehensive literature survey was conducted to identify previous research on two-phase flow modeling; this is applicable to the thesis section related to CO$_2$ capturing and condensation.

One of the most referenced relevant works is that of Christopher L. Chapman and Seri Lee (Christopher, et al., Feb. 2000). They carried out comparative thermal tests on aluminum heat sinks with both rectangular and elliptical-shaped pin fins for low ranges of air flows. In their design, the heat transfer surfaces were kept as large as possible and vortex flows were minimized by implemented airfoils.
The basic assumption in their research was that both heat sink (with elliptical fins and conventional fins) have same volume. They used thermal resistance and flow bypass quantities to measure the effects of different parameters like pressure drop, flow characteristics, and thermal conductivity on cooling performance of heat sink. Different heat sinks that used in their experimental test set-up are schematically shown in figure 1-6.

Chapman and Lee found that with same thermal resistance, heat sink with rectangular fin has about 40% more air flow, while the elliptical fin design shows better heat transfer characteristics. They found that for a wide range of air flow, the extruded straight design has meaningfully better performance compare to the other designs.

Boulares (Jihed Boulares, 2003) studied the pressure drop and heat transfer characteristic of a compact heat exchanger which equipped with pin fins. He used ANSYS to conduct numerical modeling to achieve optimum pin fin form and heat exchanger configuration. Then based on numerical results, he performed an experimental study that validated the numerical model.

More recently, Moshfegh and Nyierdy (Moshfegh, et al., 2004) discussed different approaches for numerical modeling of turbulent flow with application in electronic cooling. Their guidelines are very useful for similar works. They simulated a circular pin fin heat sink with a 3-D model. (Figure 1-7). Channel Reynolds numbers were adjusted to a range between 5000 to 14500 by applying proper inlet velocity. A wide range of results were investigated including pressure drop, temperature distribution, and flow bypass effects. The Gambit grid generation package was used to generate an unstructured 3D grid. Considering the symmetric shape of channel, only half of it was modeled. For better computational results, two grids with different mesh density were used and in the areas close to the pins the finer mesh were implemented. Their results showed that both the pressure drop and the heat transfer coefficient are dominated by proper turbulence model and correct near-wall treatments.

Because of the various industry applications for finned tubes, there are many different fields of research. Research papers referred to frequently in the literature concern correlations (Kröger, 1986), convection (Hahne, et al., 1994)and conduction (Ranganayakulu et al., 1999).
Performance of different types of finned-tubes has been assessed by Kroger (Kröger, 1986). He derived a method to evaluate the pressure drop and heat transfer characteristics of such a finned tube. Another correlation for finned-tubes was empirically derived by Idem et al. (Idem, et al., 1987). This correlation is useful in prediction of hydrothermal performance of other heat exchangers with similar configuration.

The purpose of an extended surface heat exchanger is to increase heat transfer. In this section a couple of heat exchangers have been review from this point of view. The literature study was limited to roughened surfaces, finned tubes, offset strip fins, pin fins, and micro fins as follows:

- **Roughened Surfaces**
  Heat transfer enhancement takes place through roughened surfaces by promoting turbulence (Bergles, 1988). In an article by Firth and Meyer (Firth, et al., 1983) the ribs of four different surfaces were evenly spaced and the heat transfer coefficients and friction factors were compared. Zhang et al. (Zhang, et al., 1994) used a parallel plate channel with staggered ribs to study the effects of compound turbulators on both heat transfer coefficient and pressure drop. The results of these articles showed that any advancement in heat transfer was counter-balanced by the pressure drop. Several different configurations were studied to find the design with the least pressure drop. Finding the best rib angle was the objective of a paper by Han et al. (Han, et al., 1978). It was found that ribs at a 45° angle of attack had better heat transfer characteristics than those with a 90° angle of attack for a given pressure drop. The effects of conduction were included in a study by Webb and Ramadhyani (Webb, et al., 1985). While investigating the heat transfer and friction properties for a parallel plate channel with staggered ribs, they found that conduction in the channel walls contributed to an enhanced heat transfer.

- **Internally Finned Tubes**
  An internal finned tube is round or rectangular with continuous fins inside it which enhance heat transfer. An advantage of these tubes is that they provide an easy and efficient way to enhance internal convection (Zhang et al., 1996). In Soliman et al.’s paper (Soliman, et al., 1980), the distribution of fluid temperature, fin temperature, and local heat flux with a uniform outside wall temperature was numerically derived for both finned and unfinned configurations. The results showed that the distribution of heat flux was dependent on the quantity of fins and the fin height, and that a finned tube had better heat transfer characteristics surface than a smooth tube. Renzoni and Prakash (Renzoni et al., 1987) derived heat transfer flow characteristics for developing flow in the entrance region of an internally finned heat exchanger. Their theoretical results showed the relation between Nusselt number and pressure drop for the entrance regions as well as estimates for the hydrodynamic and thermal entrance lengths. Two studies of laminar flow in internally finned tubes were completed by Rustum and Soliman (Rustum, et al., 1988) (Rustum, et al., 1990) and both focused on mixed convection. Experimental results for the pressure drop and heat transfer characteristics for laminar flow were given in their first paper (1988); these showed that the free convection had a great effect on heat transfer and approached the forced-convection predictions as the Rayleigh number decreased. Their numerical analysis was based on the geometrical parameters like fin height and number of fins, and also Prandtl number and...
modified Grashof number. From the analysis, the temperature distribution, secondary flow, wall heat flux, axial velocity, friction factor and average Nusselt number were derived. Available experimental results were compared with analysis result, and a good correlation was found. Other conclusions were that influence of free convective was delayed, and that the enhancement of the friction factor and Nusselt number was suppressed.

In internally finned tubes, the flow can also be turbulent. This condition was analysed by Patankar (Patankar et al., 1979), Webb and Scott (Webb, et al., 1980), and Kim and Webb (Kim, et al., 1993). A mixing length model was used by Patankar (Patankar, et al., 1979) for analysing an internally finned tube. For their analysis, the friction factor and the Nusselt number were derived. The results also showed that the fins were a better transfer surface than the tube wall. Webb and Scott analysed an internally finned tube for three design cases by (Webb, et al., 1980):

- reduced tube material volume for equal pressure drop and heat transfer;
- increased heat transfer for equal pressure drop and heat transfer surface area of heat exchanger; and
- reduced pressure drop for equal heat transfer and heat transfer surface area of heat exchanger.

Their results showed that although material savings were found using axial internal fins, much better savings were obtained when the fins had a helix (twisted) angle. A comparison with the empirical Carnavos correlations was made in the analytical model used by Kim and Webb (Kim, et al., 1993) in their article about axial internal fin tubes. The friction factor and heat transfer equations were derived and predicted the Carnavos friction data within 10% accuracy and the Carnavos heat transfer data within 15% accuracy.

- **Finned Tubes**

A finned tube heat exchanger consists of a tube with fins on the exterior wall. These are used in applications such as water heaters and boilers (Idem et al., 1987). Because of the varied industrial applications for finned tubes, there are many different fields of research related to them.

- **Offset Strip Fins**

Offset strip fins heat exchangers are mostly used in cooling systems such as air conditioning equipment (Carey, 1985). Basically, a recirculating flow forms between two succeeding fins and consequently fully developed flow condition will never establish. This means a continues boundary layer formation which enhances the heat transfer considerably. (Kelkar et al., 1990).

- **Pin Fins**

This type of fin was introduced earlier in this chapter.

- **Other enhancements**

The above selections represent only a portion of the range of enhanced heat exchanger configurations. For example, twisted tape inserts can be placed inside tubes (Royal, et al., 1978) to increase the heat transfer. Other enhanced heat exchangers include the plate-fin arrangement and the louvered fin compact heat exchanger (Xiao, et al., 1992) (Krishnaprakas, 1996) (Tagliafico, et al., 1996) (Atkinson, et al., 1998). Another popular fin pattern for heat transfer enhancement is the wavy fin configuration (Wang, et al., 1999). In this paper, correlations for the airside performance of a wavy fin heat exchanger were derived using samples. A generalized heat transfer coefficient and friction factor was proposed that were within 15% of the sample data.

As part of this thesis focuses on heat and mass transfer in CO₂/H₂O condensers, a brief of literature reviews of that topic is presented here. Recently, researchers have been working on steam condensation and condensation from non-condensable mixtures. Direct contact condensation inside a rectangular channel is investigated experimentally by (Ruile, 1995). In Dehbi work (Dehbi, et al., 1997) theoretical prediction of heat and mass transfer from steam and noncondensable gas mixtures within a vertical tube was calculated by deriving an algebraic equation for the film thickness from momentum balance, considering the interfacial shear stress influences. Then calculate the condensation rate, mass and heat transfer analogy approach were implemented. In 1998, Herranz studied the steam condensation and
diffusion layer model for a Westinghouse containment. They applied a mass and heat transfer analogy to model the thermal resistance and wavy structure of the condensing film. Shortly thereafter, Liu examined the condensation heat transfer coefficient in vertical smooth tubes with different fractions of non-condensable gas (Liu, et al., 2000). They showed that condensation heat transfer is lower when helium exists in the system compared to one with air in it. In 2002, Choi performed experiments on direct contact condensation when steam contains non-condensable gases (Choi, et al., 2002). They investigated water vapour condensation of steam and steam–air mixtures on the subcooled water film flows. The local empirical values for heat transfer coefficients were obtained for different mass fractions of non-condensable gas at the inlet.

Based on the literature survey, it appears that only a very few attempts have been performed on a CFD application in study of condensing flow. Yadigaroglu (Yadigaroglu, 2003) noted that computational dynamics for multi fluids mixtures is still under development and requires additional research. One of the most recent works is by Wilhelmsson (Wihelmsson, et al., 2007). They employed a CFD approach to model water vapour condensation in a channel inside a plate heat exchanger and developed their model based on the governing equations and solved them for the entire single and two-phased fields. Another recent work is by Kljenak (Kljenak, et al., 2006), in which CFX4.4 was employed as the computational fluid dynamics tools. Studied model consists of a big cylindrical vessel in which steam, air and helium gases are injected and conditions on vessel walls causes steam to be condensed on them. The goal was to simulate condensation in a non-homogeneous mixture and get the right condensation rate. A 2-D axisymmetric vessel model was created using CFX4.4 code. The approach modeled the flow in the domain simulation as single-phase while it considered the steam condensation as a sink of energy and mass. It was noted in Klejnak’s work that the modeling of steam condensation is not available in CFD codes and must be applied correctly by users. The condensation model was fed to CFX by used defined function and applying the theoretical correlations, and different theoretical condensation models were identified.

Another recent work involves the mechanistic modeling of water vapour condensation in a mixture of steam and non-condensable gases (Karkoszka, 2007). This thesis presents the analytical and numerical analysis of the water vapour condensation from the multi-component mixture of condensable and non-condensable gases in the area of the nuclear reactor thermal-hydraulic safety. Surface condensation, liquid condensate interaction with gaseous mixtures and spontaneous condensation in supersaturated mixtures have been taken into consideration throughout the thesis. It was found that in all cases that condensation heat and mass transfer rates are significantly dependent on local mixture intensive parameters (such as the non-condensable species concentration). Both analytical and numerical methods were used to find the influence of the light gas and induced buoyancy forces on the condensation heat and mass transfer processes.

Analysis was performed applying the boundary layer approximation and the similarity method to the system of film and mixture conservation equations. Numerical analysis was performed with the in-house-developed code and commercial CFD software.

The survey suggested that the greatest efforts have focused on developing the mathematical modeling for multi-phase flow and that no actual CFD modeling has been done for a condenser itself. The window is open, therefore, for evaluating the derived models and applying them to commercial CFD software (e.g., Fluent) to obtain realistic results as guidance for the improved design of multiphase condensers.

1.5 Objectives

The final goals of this thesis are to evaluate and implement CFD-based methods for realistically predicting heat transfer and fluid flow in some industrial applications related to power generation with the intention of heat transfer efficiency improvement and to provide useful information for designers in addition to evaluate the capability of current CFD codes for challenging industrial applications. There-
fore, different fin geometries featuring typical flow-heat transfer characteristics are investigated and evaluated. Then different numerical methods are evaluated for different applications, and the advantage and disadvantages of each are investigated.

The main focus is on the investigation on flow and heat transfer prediction and improvement in industrial heat exchangers by CFD methods. However a brief study of the principal physical models of all cases is included.
2. Pin Fin Technology and Modeling

2.1 Basic fin theory and definitions

Generally, when the convective heat transfer coefficient is small (a condition commonly encountered when the surrounding fluid is a gas), the rate of heat transfer can be increased considerably by installing an extended surface or a fin on the surface. This part is taken mainly from Lic thesis to keep the consistency of research in the Phd Thesis and give an overall view of pin fins theory.

Here the focus is on the pin fins which provide heat transfer augmentation via the disturbing in laminar boundary layers growth. To show the essential features of fin behavior and introduce the basic definitions, we start with the analysis of a straight fin protruding from a wall. Figure 2-1 shows such a fin with a one-dimensional characteristic.

![Figure 2-1: Analysis of one-dimensional fin (Lienhard IV, et al., 2005)](image)

The temperature of the fin’s root is kept constant at $T_0$. The fin body is cooled (or heated) by ambient air, and its temperature is assumed to be $T_\infty$. The convective heat transfer coefficient, $h$, is supposed to be uniform along the whole length of the fin. Though this assumption is not very accurate, we use it to formulate the fin behavior. The temperature distribution along the fin is necessary to obtain its associated heat transfer. Considering the negligible radiation heat transfer from the fin surface and the one-dimensional conduction through the fin, the energy balance on a thin differential element is shown in the following equation

$$-kA \frac{dT}{dx} + kA \frac{dT}{dx} + \bar{h} \cdot (P \cdot \delta x) \cdot (T - T_\infty) = 0$$

(2-1)

where $A$ represents the fin’s uniform cross-sectional area and $P$ is its circumferential perimeter.
Rearranging equation 2-1 yields the equation 2-2 such that

\[
\frac{d^2(T-T_\infty)}{dx^2} = \frac{\bar{h}P}{kA} (T - T_\infty)
\]  

(2-2)

Boundary conditions for this equation are shown in the equations 2-3(a) and 2-3(b).

\[
(T - T_\infty)_{x=0} = T_0 - T_\infty \tag{2-3(a)}
\]

\[
-kA \frac{d(T-T_\infty)}{dx} \bigg|_{x=L} = \bar{h}_L A (T - T_\infty)_{x=L} \tag{2-3(b)}
\]

For the case of negligible heat transfer at the fin tip (like an insulated tip), the boundary conditions can be considered as equations 2-4(a) and 2-4(b).

\[
(T - T_\infty)_{x=0} = T_0 - T_\infty \tag{2-4(a)}
\]

\[
-kA \frac{d(T-T_\infty)}{dx} \bigg|_{x=L} = 0 \tag{2-4(b)}
\]

To work with dimensionless parameters, we use the following substitutions.

\[
\theta = \frac{T - T_\infty}{T_0 - T_\infty} \tag{2-5(a)}
\]

\[
\xi = \frac{x}{L} \tag{2-5(b)}
\]

\[
\sqrt{\frac{\bar{h}PL^2}{kA}} = mL \tag{2-5(c)}
\]

These equations can be rearranged to give a more general form.

\[
\frac{d^2\theta}{d\xi^2} = (mL)^2 \theta \tag{2-6}
\]

This equation is satisfied by \(\theta = C_1 e^{-\xi mL} + C_2 e^{\xi mL} \) which can be written in the following form to yield the general solution of equation.

\[
\theta = C_1 e^{-\xi mL} + C_2 e^{\xi mL} \tag{2-7}
\]

\(C_1\) and \(C_2\) will be calculated according to boundary conditions, and they are different in different cases. It is obvious that the mL is a useful parameter in the analysis and design of fins. It can be rearranged as equation 2-8.

\[
(mL)^2 = \frac{L}{\frac{kA}{1}} = \frac{internal\ resistance\ in\ x\ direction}{gross\ external\ resistance} \tag{2-8}
\]

When \((mL)^2\) is big, \(\theta \bigg|_{\xi=0} \rightarrow 0\) and tip convection can be neglected. When it is small, the temperature drop along the axis of the fin becomes small.
One of the most important design variables for a fin is the rate at which it removes (or delivers) heat from/to the wall. To calculate this, the Fourier’s law for the heat flow into the base of the fin is written.

\[ Q = -kA \frac{d(T-T_\infty)}{dx} \bigg|_{x=0} \]  

(2-8)

Two other basic measures of fin performance which are particularly useful in a fin design are fin efficiency, \( \eta_f \) and fin effectiveness, \( \varepsilon_f \) (Hewitt, 1990):

\[ \eta_f = \frac{\text{actual heat transfer by a fin}}{\text{heat that would be transferred if the entire fin were at } T=T_0} \]  

(2-9)

\[ \varepsilon_f = \frac{\text{heat flux from the wall with the fin}}{\text{heat flux from the wall without the fin}} \]  

(2-10)

Though \( \eta_f \) provides some useful information about fin design, there is not a particular \( \eta_f \) which represents the best design. On the other hand, it seems that the effectiveness should be as high as possible which is always achievable by extending the length of the fin. However, this solution is a losing proposition. For a reasonable design, \( \varepsilon_f \) should be as large as possible and generally, when \( \varepsilon_f \leq 2 \), there is no use for fins.

The fin effectiveness can be increased by the following considerations:

- Selection of high conductive materials (such as copper and aluminum, however, aluminum is more usual, because of its lower price and weight).
- Increase in the circumference to section area ratio. This is the reason for using thin fins in engineering applications.
- Using fins in cases that the convection heat transfer coefficient \( (h) \) is low.

Thus fin design is still an open-ended subject of optimizing, subjected to many factors. Lienhard has summarized some of them (Lienhard IV, et al., 2005):

“1- The weight of material added by the fin.
2- The possible dependence of \( h \) on \( (T - T_\infty) \), flow velocity past the fin, or other influences.
3- The influence of the fin (or fins) on the heat transfer coefficient, \( h \).
4- The geometric configuration of the channel that the fin lies in. the cost and complexity of manufacturing fins.
5- The pressure drop introduced by the fins”.

The last parameter that we are dealing with in this section is fin thermal resistance which is defined as the ratio of a temperature difference to the resultant heat transfer rate. This means that the fin acts as a thermal resistance between the root and the surrounding ambient. For a straight fin with negligible heat transfer at its tip, we have:

\[ Q = \frac{(T_0-T_\infty)}{(\sqrt{kAhP \cdot \tanh(mL)})^{-1}} = \frac{(T_0-T_\infty)}{R_{t,\text{fin}}} \]  

(2-11)

where:

\[ R_{t,\text{fin}} = \frac{1}{\sqrt{kAhP \cdot \tanh(mL)}} \text{ for a straight fin} \]  

(2-12)

Equation (2-12) implies rate of the heat which the fin can remove from the wall. The relation between the fin thermal resistance and other fins parameter can be derived from foregoing equations and written as equation 2-14.
\[ R_{t,\text{fin}} = \frac{1}{\eta_f A_{\text{surface}} h} = \frac{1}{\varepsilon_f A_{\text{root}} h} \]  \hspace{1cm} (2.13)

2.2 Pin fin study

In this study, three different cases are considered. The heat exchanger domain consists of three connected channels: an entrance section, a pin fin section, and an exit section. The pin fin section consists of eight rows of in-line pin fins with axes perpendicular to the flow, as shown in figure 2-3. Three different pin shapes are considered: cylindrical, rectangular, and drop-shaped. The main geometrical dimensions that characterize the heat exchanger are the pin height (H), the diameter of the cylindrical portion of the pin (D [note that for rectangular pin fin this parameter is considered as fin width]), the streamwise pin spacing (\(\Delta\)) and the pin-tail length (Z). Figure 2-3 depicts more details on these parameters. The pin width (D) was kept constant at a value equal to 5 mm and was considered as a reference length scale. The total pin fin area is equal for all pin fins. The entrance duct was constructed in front of the heat exchanger pin fin section to ensure a fully developed laminar flow condition at the entrance to this section. An exit duct was also used to ensure well-mixed conditions at the exit plane. Both the entrance and exit duct are assumed to be adiabatic.

![Heat exchanger domain](image)

**Figure 2-2:** Heat exchanger domain.

![Studied fins](image)

**Figure 2-3:** Studied fins: Drop-Shaped, Rectangular, and Cylindrical.
3. Governing equations and numerical solution procedure

3.1 Introduction

The finite volume method (FVM) is based on decomposition of the physical domain into small cells which are called control volumes. The equations governing fluid flow are integrated formally over all the (finite) control volumes of the solution domain. Resulted integral equations are discretized into a system of algebraic equations by substituting finite difference approximations in them in order to represent different terms such as diffusion, convection, and sources. Finally these algebraic equations are solved iteratively. More complete details can be found in various CFD text books (e.g. (Versteeg et al., 2007) (Anderson, 1995)). Discretisation schemes exhibit different properties; among these, the First and Second Order Upwind, QUICK (third order), Power Law and Third-Order MUSCL are most frequently used in CFD codes.

Algorithms for pressure-velocity coupling in steady flows include SIMPLE, SIMPLEC and PISO (Versteeg et al., 2007). In most instances, this thesis applied the SIMPLE algorithm unless a convergence problem occurred. In such cases, SIMPLEC or PISO algorithms were used to determine a converged solution. In all cases, the physical properties of fluids were assumed to be dependent on temperature. This assumption aligns the results more closely with reality, especially in cases with large temperature variations.

In two-phase flow regimes, there are more complexities involved and special considerations should be taken into account. These issues are discussed in a later section of this chapter.

3.2 Governing equations of fluid flow and heat transfer

Considering the following conservation laws, which are proved that govern the fluids dynamic behavior, the mathematical governing equation can be derived:

1. mass conservation;
2. momentum conservation;
3. energy conservation.

The law of mass conservation states that mass cannot be created in a fluid system, nor can it disappear from one. Any variation of mass would imply a displacement of fluid particles. This fact is represented mathematically by the following continuity equation.

\[
\frac{\partial \rho}{\partial t} + \text{div}(\rho \mathbf{U}) = 0
\]  

(3.1)

Navier–Stokes equations govern the motion of viscous fluids. They are obtained by applying the Newton's second law of motion to fluid flow with some extra relevant assumptions for stress terms and considering pressure gradient. The Navier–Stokes equations are differential equations that represent the correlation between rates of change of different flow variable (e.g. velocity and pressure). A solution for these equations is called the flow field or velocity field, which describes the velocity of a fluid at a given point in space and time. Other fluid flow properties such as flow rate can be determined based on measured velocity field. The general form of these equations is (Anderson, 1995):
\[
\frac{\partial \rho U_i}{\partial t} + \text{div}(\rho U_i U_j) = -\frac{\partial p}{\partial x_j} + \text{div}(\mu \, \text{grad} U_j) + S_{Mj}
\] (3-2)

where \(S_{Mj}\) represents the source term and small contributions in the viscous stress term. Source term depends on body force type. For example, the gravity body force is modeled by: \(-\rho g\).

The equation representing the conservation of energy has a form similar to that for momentum. It is based on the first law of thermodynamics, which implies sum of the net added heat to a system and the net work done on it equally increases the system energy (Versteeg, et al., 2007).

\[
\frac{\partial \rho l}{\partial t} + \text{div}(\rho l U) = -P \text{div} U + \text{div}(k \, \text{grad} T) + \phi + S_i
\] (3-3)

The dissipation function \(\Phi\) describes the effects due to viscous stresses and can be described in terms of velocity gradients.

The conservative governing equations with two extra equations of state \((I = i(\rho, T) \& P = P(\rho, T))\) result in a system of seven equations with seven unknowns. This means that system of equations can be solved provided that suitable initial and boundary conditions are supplied.

In laminar flows, direct numerical simulations (DNS) can be used to integrate the Navier-Stokes equations numerically and find the unknown parameters. In turbulent flows, the governing equations are exactly same as those ones which described before for laminar flows. However, in turbulent flow regimes the solution is much complex. In these cases, there are two approaches to solve equations: The first, as mentioned above, is DNS which is an expensive method to implement and on the other hand is really applicable for simple geometries. The solution procedure is same as one which is used in laminar flows, however all fluctuations in pressure and velocity must be taken into account. The second approach is known as Reynolds-averaged Navier-Stokes (RANS) equations. With this approach, fluctuations in pressure and velocity are treated using terms of mean pressure and velocity. The main advantage of these equations is that it is much easier to solve them; however, to close systems of equations, we should provide some extra equations which are not really governing equation and they are some approximations which eventually introduce errors into solutions.

3.3 The instantaneous equations

The governing equations and turbulence models used for analyzing the flow and heat transfer in turbulent conditions are introduced in the following section. In the industrial instances presented in this work, forced convection flows are considered. In these cases, the ratio between the Grashof number and the Reynolds number \((Gr/Re^2)\) is much less than one, which means the buoyancy forces will be much smaller than the inertia forces and flow will be determined by both the inertia and friction forces (Lienhard IV, et al., 2005). Main heat transfer mechanism in the fins and base for the fins involves both convection and conduction. This means energy equation should be solved in both solid and fluid regions.

Based on the above assumptions for an incompressible and three-dimensional steady state turbulent flow, governing equations are simplified to following correlations:

\[
\frac{\partial U_i}{\partial x_i} = 0
\] (3-4)
\[
\frac{\partial \vec{u}_i \vec{u}_j}{\partial x_j} = -\frac{\partial \bar{P}}{\partial x_i} + v_i \nabla^2 U_i + \frac{\partial}{\partial x_j} \left( -\bar{u}_i \bar{u}_j \right) \tag{3-5}
\]

\[
\frac{\partial \bar{u}_i \bar{T}}{\partial x_j} = \alpha \nabla^2 T + \frac{\partial}{\partial x_j} \left( -\bar{u}_i \bar{\theta} \right) \tag{3-6}
\]

where \( \bar{P} \) is the modified kinematics pressure \((p/\rho)\) and \( \bar{u}_i \bar{u}_j, \bar{u}_i \bar{\theta} \) are Reynolds stresses and turbulent heat fluxes respectively. It is necessary to model these two parameters in order to get a closed system of equations. In the solid region, the same form of energy equation can be used, but the convective terms should be removed.

The most popular models to approximate \( \bar{u}_i \bar{u}_j \) and \( \bar{u}_i \bar{\theta} \) are 1- eddy-viscosity turbulence models 2- Reynolds stress transport model (RSM) (Moshfegh, et al., 2004). In the first, the turbulent transport phenomenon is analogized to viscous transport and Boussinesq assumption is used for \( \bar{u}_i \bar{u}_j \) and \( \bar{u}_i \bar{\theta} \) modeling:

\[
\bar{u}_i \bar{u}_j = -2v_i S_{ij} + \frac{2}{3} \delta_{ij} k \tag{3-7}
\]

\[
\bar{u}_i \bar{\theta} = \frac{u_i \sigma_t \delta_{ij}^\prime \delta_{ij} \delta_{ij} \sigma_t^\prime}{\delta_{ij} \delta_{ij} \sigma_t \delta_{ij}^\prime \delta_{ij} \sigma_t^\prime} \tag{3-8}
\]

In equations 3-7 and 3-8, \( k \) stands for turbulent kinetic energy, \( \nu_t \) represents the eddy viscosity, \( \sigma_t \) represents the turbulent Prandtle number , and \( S_{ij} = 0.5 \left( \frac{\partial U_j}{\partial x_i} + \frac{\partial U_i}{\partial x_j} \right) \).

In this category, several models have been proposed to state the \( \nu_t \) and, therefore, two unknowns \( \bar{u}_i \bar{u}_j \) and \( \bar{u}_i \bar{\theta} \). One important model is two-equation model (Moshfegh, et al., 2004) which implies there is direct proportion between velocity and length scales in mean flow and turbulence flow. The proportion can be expressed in terms of turbulent kinetic energy \( k \) and dissipation rate \( \varepsilon \) as follows:

\[
u = k^{0.5} , \quad l = k^{1.5} \varepsilon^{-1} \tag{3-9}
\]

It is obvious from these equations that it is required to provide two more equations for \( k \) and \( \varepsilon \) to close system of equations. The most important models in this category are the standard \( k-\varepsilon \) model, the renormalization group (RNG) \( k- \varepsilon \) model, the realizable \( k-\varepsilon \) model and the \( k-\omega \) model. More details about each can be found in (Versteeg, et al., 2007). In these equations, energy dissipation rate, \( \varepsilon \), is given by:

\[
\varepsilon = \nu \frac{\partial u_i^t \bar{u}_j^t}{\partial x_j} \tag{3-10}
\]

The second model, RSM, is also known as “second-moment model,” and in this model, two unknowns \( \bar{u}_i \bar{u}_j \) and \( \bar{u}_i \bar{\theta} \) are solved, when averaging Navier-Stokes equations. This model is not implemented in the current work; however, more details can be found in literatures (Moshfegh, et al., 2004) (Versteeg, et al., 2007) (Wilcox, 1998).

In the current work, Moshfegh results on comparison between different turbulence models are considered (Moshfegh, et al., 2004). In his work, different turbulence models are evaluated and compared with experimental data. The studied case involved a rectangular channel where circular pin fin heat sinks are installed on it floor. Comparison between results from experiments and results from various
turbulence models for same \( T_b \) (base temperature) and \( \Delta p \) are given in Table 3-1. Results show that the predictions of RSM model and two-equation models are close to each other. Our studied case closely aligns with this study, and considering the element of experience, the \( k-\varepsilon \) models are implemented in our pin fin cases as a robust and simple turbulent model. The original intent here was not to compare the different turbulent models but to choose a fair, proven model and use it to generate acceptable results in order to focus on other important issues of the flow itself.

Table 3-1: Comparison of different turbulence modeling (Moshfegh, et al., 2004).

<table>
<thead>
<tr>
<th>Case 2</th>
<th>Coarse mesh</th>
</tr>
</thead>
<tbody>
<tr>
<td>( T_b ) [°C]</td>
<td>35.8</td>
</tr>
<tr>
<td>CFD/Exp</td>
<td>-</td>
</tr>
<tr>
<td>( \Delta p ) [N/m²]</td>
<td>2.38</td>
</tr>
<tr>
<td>CFD/Exp</td>
<td>-</td>
</tr>
</tbody>
</table>

Another issue in flow modeling involves transient cases. At values of the Reynolds number above \( Re_{cri} \) (critical Reynolds), certain complicated events take place which result in a basic change of the flow character. In its final state, the flow behavior is random and chaotic, otherwise known as turbulent flow. It should be noted that there is no comprehensive theory of transition (Anderson, 1995) (Versteeg, et al., 2007) (Wilcox, 1998). Fluent cannot presently compute either boundary layer instability or subsequent transition to turbulence using a RANS approach (e.g., \( k-\varepsilon \), RSM), so it often ignores transition entirely and classifies flows as either laminar or fully turbulent. However, the transition region often comprises only a very small fraction of the size of the flow domain, and in those cases it is assumed that the errors made by neglecting its detailed structure are only small ones. In this thesis, the focus is on continuous stable conditions; therefore, boundaries are considered, in a way, to have a laminar flow or a fully turbulent flow to avoid numerical problems accompanied by transient conditions.

3.4 Simulation of pin fin model

A commercial finite volume analysis package called FLUENT© was used to perform numerical analysis on the model. The models were constructed using Gambit software, and the models data were passed to the Fluent software for various analyses. The governing equations solved were the Navier-Stokes equations combined with the continuity equation, the energy equation, and constitutive property relationships as described in the previous section. Once the analyses were completed, the resulting data were easily evaluated by the Fluent postprocessor.

3.4.1 Numerical mesh

Model is meshed by specifying the hexahedral 8-node element spacing along the boundary and sweeping later to cover the entire model volume. Meshing was also refined in critical areas to ensure coverage for satisfactory resolution, both near the no-slip walls where the velocity and temperature gradients were expected to be high, and also between the pins to capture flow acceleration due to the decrease in the cross-section area. Grid independence was verified, and sensitivity analysis was performed to certify that the number of used cells was optimal. Different cell densities (the number of cells divided by the volume) were modeled and results were compared. A result was considered valid if the model outlet temperature matched the calculated outlet temperature using an energy balance method to within 2%.
3.4.2 Solution technique
All flows were specified as steady state and incompressible. The standard k-ε turbulence model with
standard wall function was set for each model. The segregated 3D solver with an implicit formulation
was set to solve the models.

3.4.3 Model validation
Several checks were performed in order to verify the generated results. Three orders of magnitude of
convergence were maintained for each solution in order to ensure the accuracy of the results. The con-
tour plots for velocity, temperature, and pressure were observed separately to ensure that the results
satisfied the boundary conditions.

The resulting file generated by Fluent upon the completion of each run was carefully examined and
analyzed. The conservation of mass was verified by comparing the inlet and outlet mass flow rates to
ensure that the mass balance was achieved. The energy flow into and out of the system at the flow
boundaries were ensured to be equal to the total energy added at the hot wall. Computational errors in
both pressure drop and heat transfer measurements were commonly encountered. This sometimes re-
sulted in a diverging solution and was corrected by increasing the reference pressure and/or relaxing
the modified inertia criteria.

For validation of the numerical model, the cylindrical case with different Reynolds number was used.
The simulations results were compared with theoretical correlation for the Nusselt number (Lienhard
IV, et al., 2005) and results are presented in Figure 3-1. This theoretical correlation was developed for
a single cylindrical fin in a cross flow.

\[
\bar{N}_u_D = 0.3 + \frac{0.62Re_D^{1/2}Pr^{1/3}}{[1 + (Re_D^{1/2}Pr^{1/3})^{1/4}]} \tag{3-11}
\]

The resulting maximum relative errors varied between 4.6 to 47.6%. However, it should be noted that
in our case, we used a series of fins rather than a single fin, which could be one source for the ob-
served discrepancies. The validation of the numerical results was limited to the average Nusselt num-
ber in the pin fin section. The differences were considerable in some cases, especially at the transient
Reynolds number range. However there is a good agreement between results across most of the range.

![Figure 3-1: Comparison of Nusselt number in CFD modeling and theoretical model.](image)

22
3.4.4 Test approach

3.4.4.1 Test matrix

A range of studies was conducted based on the three-dimensional model of pin fins with different morphologies. The pins’ surface areas for all computation simulations were kept constant with varying Reynolds numbers. These were chosen across a wide range to cover both laminar and turbulent flow regimes. Table 3-2 shows a sample test matrix. Depending on the Reynolds number, the appropriate laminar or turbulent model was chosen in Fluent.

Table 3-2: Test configuration.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>ReD</td>
<td>500, 1000, 3000, 5000, 10000, 50000</td>
</tr>
<tr>
<td>Fin Morphology</td>
<td>Cylindrical, Rectangular, Drop-shaped</td>
</tr>
</tbody>
</table>

3.4.4.2 Data analysis

Crucial results such as total heat transfer rate, pressure drop, inlet and outlet bulk temperature and mass flow rate were obtained. The Nusselt number and friction factor of the pin fins array were calculated using heat transfer relationships. A complete calculation of the energy balance was performed to verify the results, obtained directly from Fluent.

3.5 Two-phase flow theory and modeling

3.5.1 Surface condensation theory

Condensation occurs when vapour is cooled sufficiently below the saturation temperature $T_{\text{sat}}$. Usually this happens when vapour is brought into contact with a solid surface having a temperature $T_s$ below the saturation temperature of the vapour. However, condensation can also take place within a gas or on the liquid free surface. When condensation occurs in a gas, the liquid droplets are usually suspended in that gas. Because it is easier in industry to operate and control surface contact condensers, it is common to consider only film condensation in the design of heat transfer equipment, and based on what is known about oxy-fuel CO$_2$ capturing, we have studied condensation on solid surfaces only, with focus on film condensation.

During film condensation, surface becomes wet and subsequently a liquid film develops on it. This liquid film slides down under the influence of gravity. As more vapours condenses on the film in the flow direction, the thickness of the film increases. The film covers the solid surface and forms a liquid wall between that solid surface and the condensing vapour. This liquid film acts as a resistance to heat transfer. Therefore, the heat released from vapour condensation at the vapour-liquid interface must be transmitted through this heat resistance layer before it can reach the solid surface and be passed to the cooling medium on the other side. Figure 3-2 shows film condensation mechanism. This is normally observed in a pure substance surface condensation.
However, the presence of even a small quantity of non-condensable gas has a significant influence on the resistance to heat transfer in the region of the liquid-vapour interface. Experimental studies show that the presence of non-condensable gases in the mixture has an unfavorable effect on condensation heat transfer. As an example, the presence of less than 1% (by mass) of air in steam can reduce the condensation heat transfer coefficient by more than half (Cengel, 2003). A non-condensable gas is carried within the vapour towards the interface, where it accumulates. Thus, special consideration should be given when condensation from a CO₂/H₂O mixture is studied. In this case, a large portion of the gas stream is occupied by CO₂, which is considered a non-condensable gas under normal conditions. This typifies a multi-component (n > 2) mixture condensation where n is the number of components. The CO₂/H₂O flue gas is a binary (n = 2) mixture and its phase equilibrium characteristics are important in flue gas condenser design and operation. Condensation of water vapour from a CO₂/H₂O flue gas mixture on a vertical smooth surface is shown in Figure 3-3. The severe reduction in the condensation heat transfer coefficient when a non-condensable gas exists in the mixture may be explained as follows: during the condensation of the water vapour from a mixture that contains non-condensable gas, only the non-condensable gas remains in the vicinity of the interface surface.

This gas layer acts as an obstacle between the vapour and the surface and makes it difficult for the vapour to reach the surface, which reduces the efficiency of the condensation process. Experimental studies show that condensation efficiency in this case strongly depends on the nature of the mixture flow and its velocity. A high-velocity flow will more likely remove the still non-condensable gas layer adjacent to the liquid-gas interface, and thus increase the heat transfer rate.

Within the experimental and numerical work on surface condensation, most of the researches have been performed primarily on condensation from steam and non-condensable mixtures. However the main goal in the current research was to use numerical methods to simulate both heat and mass transfer. This work develops a model based on available two-phase flow models and applies it using a numerical scheme with the help of Fluent software to study water vapour condensation from a flue gas containing mainly CO₂ and H₂O.
3.5.2 Physical model for binary system condensation

In the mixture of CO$_2$/H$_2$O, the partial pressure, $P_A$, of component H$_2$O is that pressure which would be exerted by H$_2$O alone in the mixture appropriate to the concentration of H$_2$O in the flue gas at the same temperature. Here A represents H$_2$O to simplify the proceeding equations (sometimes A is referred to as a general representative of one component in the mixture). Since $P = \sum P_A$, then the partial pressure, $P_A$, is proportional to the mole fraction of H$_2$O in the vapour phase.

$$P_A = x_A P \quad (3-12)$$

The best known correlation which relates the partial pressure of vapour phase to the concentration of component A in the liquid phase is Raoult’s law, and it states that the partial pressure $P_A$ is related to the mole fraction $x_A$ and the saturation pressure of pure component A at the same temperature, such that

$$P_A = x_A P_{sat} \quad (3-13)$$

where $x_A$ represents the mole fraction of component A in the liquid phase.

3.5.3 Mass transfer considerations

There is a strong relationship between condensation and mass transfer in multi-components gases. In the studied case, water vapour diffusion drives the condensation heat and mass transfer. During the mass transfer process, water vapour travels from a region of high concentration to a region where it has a low concentration. Just as thermal energy diffuses as it moves from a region of high temperature to one of low temperature (following the temperature gradient), so the mass transfer follows the con-
centration gradient. Fick’s law of diffusion states that the diffusion mass flux of a species in a multi-component stream is directly proportional to its concentration gradient. This law can be expressed as

\[ \overrightarrow{J_i} = -\rho \cdot D_{im} \cdot \frac{\partial m_i}{\partial x} \]  

(3-14)

where \( \overrightarrow{J_i} \) is the total mass flux of species \( i \) and gets the unit of \( (\text{kg/m}^2 \cdot \text{s}) \). \( D_{im} \) is the effective diffusivity which is used to represent the diffusion of species \( i \) into a mixture \( m \). The mass diffusivity has a unit of \( \text{m}^2/\text{s} \). When diffusion occurs in a mixture with only two species, this coefficient is called the binary diffusion coefficient, \( D_{12} \). Diffusivity coefficient is a function of composition, temperature, and pressure, and for gases is typically on the order of \( 10^{-5} \) (\( \text{m}^2/\text{s} \)) near room temperature (Lienhard IV, et al., 2005). There are several empirical correlations to calculate the diffusivity coefficient for a binary mixture. The Chapman-Enskog correlation is based on kinetic theory and takes into account all the molecular effects precisely. The correlation is:

\[ D_{AB} = \left( \frac{1.8583 \times 10^{-7} \gamma^2}{\rho \sigma_{AB}^4 \Omega D} \right) \sqrt{\frac{1}{M_A} + \frac{1}{M_B}} \]  

(3-15)

Units of \( \rho, T \) and \( D_{AB} \) are \( \text{atm} \), \( \text{K} \) and \( \text{m}^2/\text{s} \) respectively. \( \sigma_{AB} \) is the average molecular diameter in \( \text{Å} \) and is equal to \( (\sigma_A + \sigma_B)/2 \). \( M_A \) and \( M_B \) are molecular weight of components A and B. \( \Omega D \) can be obtained from following correlation, based on data taken from Lienhard IV (Lienhard IV, et al., 2005).

\[ \Omega D = \frac{a + c \cdot x^d}{b + x^d} \]  

(3-16)

Coefficient Data:

\( \begin{align*}
    a &= 7.50644892705E-001 \\
    b &= 4.63723707964E+000 \\
    c &= 4.63618614657E+000 \\
    d &= -1.24747368172E+000
\end{align*} \)

\( x \) is a special parameter as following:

\[ x = \frac{k_B T}{\varepsilon} \]  

(3-17)

This parameter can be calculated considering the mixture temperature and information presented in Table 3-3.

Table 3-3: Required data for mass diffusivity calculation (Lienhard IV, et al., 2005).

<table>
<thead>
<tr>
<th>Species</th>
<th>( \varepsilon/\kappa_0 )</th>
<th>( \sigma(\text{Å}) )</th>
<th>( M(\text{kg/kmol}) )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Air</td>
<td>78.6</td>
<td>3.711</td>
<td>28.96</td>
</tr>
<tr>
<td>H(_2)O</td>
<td>363</td>
<td>2.655</td>
<td>18.02</td>
</tr>
<tr>
<td>CO(_2)</td>
<td>195.2</td>
<td>3.941</td>
<td>44.01</td>
</tr>
</tbody>
</table>

When there are more than two species in the gas stream, different equations are needed. There are some simplified cases. If a low concentration of species \( i \) diffuses into a homogeneous mixture of \( n \) species, then:
\( J_j = 0 \) for \( j \neq i \), and

\[
D_{ij}^{-1} = \sum_{i=1}^{n} \frac{s_i}{D_{ij}}
\]

(3-18)

where \( D_{ij} \) is the binary diffusivity coefficient for species \( i \) and \( j \) alone. If a mixture is composed mainly of one species and includes only small portions of several other species, then the diffusion coefficient of each dilute gas is approximately the same as it would be if the other dilute gases were not present; that is,

\[
D_{im} \cong D_{IA}
\]

(3-19)

3.5.4 Influence of non-condensable gases on condensation

The presence of a small amount of non-condensable gas in a condensing vapour has a major effect on the resistance of heat and mass transfer in the liquid-vapour interface. The non-condensable gas is carried towards the interface and accumulates there. This accumulation causes the partial pressure of gas at the interface to become greater than its partial pressure in the binary mixture, an effect which produces a driving force for non-condensable gas to diffuse again toward the bulk. This diffusive motion is contrary to water vapour diffusion toward liquid-vapour interface. Moreover, when the vapour which is mixed with a non-condensable gas is condensing, only the non-condensable gas remains in the vicinity of the liquid-gas surface. This gas layer acts as a barrier between the vapour and the condensing surface and makes it hard for the vapour to come into contact with the surface. Therefore, vapour should diffuse through the non-condensable gas layer first before reaching the surface. These effects reduce the condensation process efficiency. Significantly, these conditions are worse for the CO₂/H₂O condenser, where the concentration of CO₂ is much higher than the water vapour. The accumulation of CO₂ in the vicinity of the condensation layer creates a barrier to the remaining water vapour in the flue gas stream, and decreases the effectiveness of condensation process.

3.5.5 Numerical modeling of two-phase flow

Before analysing the techniques for multiphase modeling, some basic definitions will be provided. A phase is considered to be a class of matter which has a definable boundary, and it has a particular dynamic response to the surrounding flow field. Generally, phase is identified as having a solid, liquid or gaseous state of matter, but it can also refer to other forms such as particles with different sizes. A multiphase flow is known as a simultaneous flow of:

- materials with different states or phases (i.e., gas, liquid or solid); or
- materials with different chemical properties but in the same state or phase (i.e., liquid-liquid, such as oil-water).

In contrast, a multi-component flow is referred to as a mixture whose components are mixed at the molecular level and all components have the same velocity and temperature. However, any phase can be composed of either a single component or a mixture of different species.

3.5.5.1 Comparison of different approaches for multiphase modeling

Two approaches are used presently in the multiphase flows modeling: the Euler-Lagrange approach and the Euler-Euler approach (Fluent, 2005). Therefore, the first step in solving the multiphase problem is to assess which of the flow regimes best represents our case flow. Then we can select the appropriate model for numerical selection.
In the Euler-Lagrange approach, the dominant flow regime is fluid phase which time-averaged Navier-Stokes equations are solved for it to obtain the comprehensive flow field. The dispersed phase which could be bubbles or particles is traced in the calculated flow field. However the basic supposition in this approach is that volume fraction of dispersed phase is negligible compared to volume fraction of fluid phase. Some examples of its application are fuel combustion and spray dryer modeling.

In contrast, in the Euler-Euler approach, all phases interpenetrate into each other and there the phasic volume fraction concept is used which implies some of all volumes is equal to one and volumes quantity depends on both time and space. Based on these assumptions, time-averaged Navier-Stokes equations are applied to each phase separately. There are three Euler-Euler models available for multiphase flow modeling in the FLUENT (Fluent, 2005):

1. volume of fluid (VOF) model,
2. mixture model,
3. Eulerian model.

The volume of fluid (VOF) model is used wherever the position of the interface between immiscible fluids is of interest. Same set of momentum equations is used for all fluids; however the volume fraction of each fluid is calculated through the time and space. This model is mostly used to model steady state or transient behavior of liquid-gas interface.

The mixture model is used wherever phases move at either same or different velocities and there is strong coupling between phases. This model cannot be used to model melting and solidification.

The Eulerian multiphase model is used for almost any combination of gases, liquids or solids. The main assumption is that these are separate phases, but they interact with each other. Compressible flow, inviscid flow, melting and solidification are not allowed in this model.

Wet steam forms whenever steam expands rapidly and passes the vapour-saturation equilibrium line. Fluent has adapted a Eulerian-Eulerian method to simulate the wet steam flow that is known as the wet steam model and is suitable for homogeneous-condensation of spherical droplets. More information about each of mentioned models can be found in Fluent user guide. (Fluent, 2005)

Our evaluation of the above models and comparison of their characteristics indicated that the Eulerian multiphase model and the wet steam model are the most suitable for condensation modeling. However, some modifications are needed and UDF should be applied in special cases.

3.5.6 Governing equations

The governing equations for the conversation of energy, momentum and mass given by (Versteeg, et al., 2007) can be formulated using tensor notation, as expressed in the following sections. The flue gas mixture and condensation layer are considered to be Newtonian fluids. Fluent software is used to solve the continuity, momentum and energy equation for each species in a structured or unstructured mesh according to the finite volume method. The problem equations system is closed using the standard (or another) k-ε turbulent model.

3.5.7 Continuity equation

Conservation equation for a species in a multiphase mixture is represented by following equation:

$$\frac{\partial}{\partial t} (\rho y_i) + \nabla \cdot (\rho y_i \mathbf{v}) = -\nabla \cdot \mathbf{J}_i + S_i$$  (3-20)

Equation is called continuity equation where $S_i$ is the source term for species mass change (kg/s m³) and $\mathbf{J}_i$ as stated in equation 3-14 is the diffusive flux of species i.
\[
\frac{\partial}{\partial t} (\alpha_q \rho_q \mathbf{u}_q) + \nabla \cdot (\alpha_q \rho_q \mathbf{u}_q \mathbf{u}_q) = \sum_{p=1}^{n} \dot{m}_{pq} \tag{3-21}
\]

where \( \alpha_q \) is the volume fraction for the \( q \)th phase and is defined as following:

\[
Volume\ Fraction = \alpha = \frac{\text{Volume of the phase in a cell/domain}}{\text{Volume of the cell/domain}} \tag{3-22}
\]

### 3.5.8 Momentum equation

Relation between resultant force on the fluid element with rate of momentum change for that element is represented by momentum equation as following:

Transverse term + Convection term = Pressure force + Body force + Shear force + Inter-phase forces + and momentum exchange + other external forces

This is shown in the mathematical form as:

\[
\frac{\partial}{\partial t} (\alpha_q \rho_q \mathbf{u}_q) + \nabla \cdot (\alpha_q \rho_q \mathbf{u}_q \mathbf{u}_q) = -\alpha_q \rho \mathbf{g} + \alpha_q \rho \mathbf{g} + \sum_{p=1}^{n} (R_{pq} + \dot{m}_{pq} \mathbf{u}_q) + \alpha_q \rho \mathbf{F}_q \tag{3-23}
\]

where \( \dot{m}_{pq} \) represents the mass transfer from \( q \)th phase to the \( p \)th phase and depends on physical state of the phases interface. Mass transfer source may also be caused by heterogeneous reaction. The momentum equation can be simplified for different flows based on the flow conditions.

### 3.5.9 Multiphase species transport equation

The general multiphase species transport equation for species \( i \) belonging to a mixture of the \( q \)th phase is expressed as follows:

\[
\frac{\partial}{\partial t} (\alpha_q \rho_q y_q^i) + \nabla \cdot (\alpha_q \rho_q \mathbf{u}_q y_q^i) = -\nabla \cdot (\alpha_q y_q^i \mathbf{F}_q) + \alpha_q y_q^i \mathbf{S}_i + \sum_{p=1}^{n} (\dot{m}_{pq}^i y_q^i - \dot{m}_{pq}^i) \tag{3-24}
\]

where \( y_q^i \) represents the mass fraction of the species \( i \) in the \( q \)th phase and \( S_i \) is the rate of increase (could be a negative value) of the mass of component \( i \) per unit volume of the mixture.

### 3.5.10 Energy equation

Energy conservation of a multiphase fluid requires an extra equation for enthalpy which is written separately for each phase. This equation, based on Fluent analysis, is represented in the following form (Fluent, 2005):

\[
\frac{\partial}{\partial t} (\alpha_q \rho_q h_q) + \nabla \cdot (\alpha_q \rho_q \mathbf{u}_q h_q) = \alpha_q \rho \mathbf{g} \cdot \nabla T + \alpha_q \rho \mathbf{g} \cdot \mathbf{H} + \sum_{p=1}^{n} (R_{pq} + \dot{m}_{pq} h_q) + \alpha_q \rho \mathbf{F}_q \cdot \nabla T \tag{3-25}
\]
\[
\frac{\partial}{\partial t} (\alpha_q \cdot \rho_q \cdot h_q) + \nabla \cdot (\alpha_q \cdot \rho_q \cdot \tilde{u}_q \cdot h_q)
= -\alpha_q \frac{\partial p_q}{\partial t} + \nabla \cdot (\alpha_q \cdot \tilde{\tau}_q \cdot \nabla \tilde{u}_q) - \nabla \tilde{q}_q + S_q
\]
\[
+ \sum_{p=1}^{n} (Q_{pq} + m_{pq} h_{pq} - m_{qp} h_{qp})
\]

(3-25)

Here \(h_q\) is the specific enthalpy of the \(q^{th}\) phase, \(\tilde{u}_q\) is the heat flux, \(S_q\) is added to include enthalpy sources, \(Q_{pq}\) represents any net heat exchange between the \(p^{th}\) and \(q^{th}\) phases, and \(h_{pq}\) is enthalpy exchange between two phases. \(\tilde{\tau}_q\) is the phase stress-strain tensor (Fluent, 2005).
4. Results and Discussion

4.1 Introduction

Detailed results and discussions about the simulations procedure and achieved findings may be found in the listed appendix papers. Nevertheless, this chapter presents the total results as well as some additional details of interest which are not included in the papers or discussed before in Lic thesis, in order to make the thesis more comprehensive. Moreover, to give a flavor of the individual papers, the physical features in the studied cases are reviewed.

4.2 Pin fin morphologies

These investigations were comprehensively presented in Lic Thesis and here a summary of main results are presented. An investigation was made to identify the optimum fin form based on a thermo-hydraulical point of view. The heat exchanger domain consisted of three connected channels: an entrance section, a pin fin section, and an exit section. The pin fin section contained eight rows of in-line pin fins with an axis perpendicular to the flow. Three different pin shapes were studied: cylindrical, rectangular and drop-shaped. Figure 4.1 illustrates the computational domain.

![Figure 4-1: Computational domain including the generic fin forms.](image)

4.2.1 Nusselt number

The Nusselt number was calculated for each test configuration. The graph in Figure 4-2 shows the relation between Nusselt number and Reynolds number on an logarithmic scale for all pin fin configurations. The data indicate that the Nusselt number can be correlated with the Reynolds number as a power law function, which is shown in equation 4-1.
Moreover, it is also apparent that the Nusselt number is sensitive to flow regime and, clearly, this sensitivity is greater for rectangular fins than for cylindrical or drop-shaped fins. These observations led to the conclusion that during the flow regime change, there is a sharp increase in NuD for all configurations. The drop-shaped pin fin has the minimum NuD among all morphologies.

The correlation for Nusselt numbers were derived as follows:

**Cylindrical**, Turbulent flow regime:

Shifed power fit: \( \text{Nu}_D = a \cdot (Re_D - b)^c \)  

Coefficient Data: \( a = 4.9120403, \ b = 4946.4338, \ c = 0.29401445 \)  

7000 < Re < 60000

**Rectangular**, Turbulent flow regime:

Hoel model: \( \text{Nu}_D = a \cdot b^{Re_D} \cdot Re_D^c \)  

Coefficient Data: \( a = 127.88763, \ b = 1.0000086, \ c = -0.041831821 \)  

7000 < Re < 60000

**Drop-shaped**, Turbulent flow regime:

Logarithm fit: \( \text{Nu}_D = a + b \cdot \ln(Re_D) \)  

Coefficient Data: \( a = -252.63549, \ b = 33.607692 \)  

5000 < Re < 60000

### 4.2.2 Friction factor

The friction factor results were plotted for three configurations in a similar manner. The graph in Figure 4-3 shows the friction factors for different Reynolds numbers. It can readily be seen that the friction factor varied greatly in connection to changes in ReD. At high Reynolds numbers, all the graphs tended to converge toward a common value. There was a sharp increase in friction factor for non-circular morphologies during the changing of the flow regime. This increment was smoother for circular pins. It should be noted that the H/D ratio for drop-shaped pin fins is smaller than that for the others (although the wetted surface area was equal for all). This behavior can be justified by evaluating figure
It can be seen that in drop-shaped pin fins, flow is accelerated in the pin fin section, which could consequently cause more friction and pressure drop.

**Figure 4-3:** Friction factor.

**Figure 4-4:** Velocity contour (ReD = 10000, Horizontal Surface, Y=75 mm).

Same as Nusselt number, there may be cases where a simpler mathematical model is desired for programming. In order to derive such a model, relationships for the behavior of friction factor as a function of Reynolds and morphologies were examined.

**Cylindrical**, Turbulent flow regime:

Vapour Pressure Model: \( f = e^{a + \frac{b}{Re_D} + c \ln(Re_D)} \)  

Coefficient Data: \( a = 1.28229327222, b = 210.809018002, c = -0.4517080322 \)  

5000 < Re < 60000

**Rectangular**, Turbulent flow regime:

Hoerl model: \( f = a b^{Re_D} Re_D^c \)  

Coefficient Data: \( a = 544.25107, b = 1.0000162, c = -0.97799842 \)  

3000 < Re < 60000

**Drop-shaped**, Turbulent flow regime:

Hoerl model: \( f = a b^{Re_D} Re_D^c \)  

Coefficient Data: \( a = 697.01874, b = 1.0000214, c = -1.0321644 \)  

3000 < Re < 60000
4.2.3 Heat transfer

One of the unique advantages of a numerical study is the ability to obtain detailed local heat transfer coefficient values at any point in the test array. These data provide opportunities to evaluate the relative contributions of the pin fin surface to the total heat transfer problem and to identify other regions of interest.

Additionally, it is possible to observe the changes in the heat transfer coefficient based on the location of the pins. Ultimately, these observations can lead to improvement in the heat exchanger performance. In this section, the numerical solution for the different configurations was used to probe the local heat transfer behavior within the array in detail.

As previously stated, pin fins were introduced in the planar duct to enhance overall heat transfer performance. They achieve this through the addition of their own surface area and by increasing flow turbulence levels, thereby giving rise to better transport rates. In the current section, results for two different Reynolds numbers (1000 and 50000) are presented. The first one addresses the laminar flow regime, and the second one shows the turbulent flow regime.

Figure 4-5 provides a plot indicating the contour of the heat transfer coefficient on the iso horizontal surface, exactly in the middle of the pins’ body as the flow moves from inlet to outlet. It is evident that the pins affected the temperature distribution. It is also obvious that in the drop-shaped pin fins, the flow reached its fully developed conditions more quickly than in the other pins. The heat transfer coefficient has the highest value in the drop-shaped pins and the lowest in the rectangular. The reason is that there is a strong recirculation flow between pins in the rectangular design. This recirculation was reduced in the cylindrical and drop-shaped arrays. Recirculation acts as a wall preventing the fresh air contribution in heat transfer. Figure 4-6 shows the same temperature contour in a vertical surface. Here it is even clearer that the heat transfer coefficient is higher among drop-shaped pin fin arrays. In addition, this clearly illustrates the beneficial effect that flow turbulence has on the base plate heat transfer performance. Because of the high conductivity of copper, there is no difference between the main base temperature and the pin fins' surface temperature, which is an enhancing tool for heat transfer. Figures 4-7 to 4-8 consist of different plots showing the temperature distribution in different parts of solution domain. In the first plot, the temperature distribution appears nearly same for all three cases, while there is a small difference between outlet temperatures. The difference is less than 1°C. Noticeably, air temperature levels are higher at the last pin. The subsequent figures (4-9 to 4-12) are drawn for the same morphologies, but for different Reynolds numbers. In this case, the flow regime is fully turbulent.
Figure 4-5: Temperature distribution, Horizontal surface (ReD =1000, Y= Cte)

Figure 4-6: Temperature distribution, Vertical surface (ReD =1000, X= Cte)

Figure 4-7: Temperature distribution, First pin (ReD =1000, Z = Cte).

Figure 4-8: Temperature distribution, last pin (ReD = 1000, Z = Cte).

The figures show that for drop-shaped fins, the temperature distribution is much smoother. Worth noting is that the fin temperature has the highest average value for the rectangular morphology, and the lowest average value for the drop-shaped array. This means that the temperature difference between fin surface and impingement air is larger in drop-shaped pins, which yields a better convection heat transfer.

Another interesting observation is that if we do not consider the pressure loss effects and simply look for more cooled fins, the rectangular pins could be a good choice. But figures show that generated mesh needs more refining to exhibit more reliable data. It can be seen that the pressure loss is higher in the rectangular pin array. The drop-shaped pin fins exhibit the least pressure loss since in this configuration; flow particles follow a smoother path line.
Figure 4-9: Temperature distribution, Horizontal surface (ReD = 50000, Y = Cte).

Figure 4-10: Temperature distribution, Vertical surface (ReD = 50000, Y = Cte).

Figure 4-11: Temperature distribution, First pin (ReD = 50000, Z = Cte).

Figure 4-12: Temperature distribution, last pin (ReD = 50000, Z = Cte).
4.3 Industrial application in power transformers cooling

Selected results from findings in the previous section were implemented in a significant industrial application: power transformers cooling. In the first case, a radiator block was simulated numerically and results were investigated (figure 4-14). Then a more in-depth study was performed on a single radiator in the block, and suggestions based on pin fin studies were presented.

Each radiator block consisted of 18 individual cooling radiators, and power transformers were equipped with several radiator blocks based on their individual power rating. Technical data about the radiator blocks are presented in the attached papers. The radiators were steel plate heat exchangers installed vertically next to transformers. Every individual radiator consisted of seven channels for oil flow, and each channel was equipped with two internal wings throughout to increase the heat transfer rate from the radiator. Two identical steel plates were pressed and welded together to form the radiator. At the upper and lower parts of the radiator, semicircular channels were provided as places for oil flow entry and exit. Oil was directed from the transformer and flowed to an upper header which directed it to a semicircular channel. This channel directed the oil into channels inside the radiator. Finally, oil was collected in a lower semicircular channel and moved out. The cooling capacity of each radiator block was defined according to the number and dimensions of its cooling elements (radiators).

4.3.1 Power transformer cooling system

4.3.1.1 Radiators block

Figure 4-14 illustrates the radiator block which was simulated, including all radiators and both upper and lower headers. Since the geometry significantly affects the flow behavior inside the radiator elements, these headers and their connections to the radiators were modeled with great care. All important geometrical aspects were included in the model, and each radiator element was considered as a rectangular cube.

Temperature distribution on the outer walls of the radiator block is shown in figure 4-15. The temperature is almost constant in the upper header, but in the lower header it is quite different. In the radiators located at the right side of the block, the exit temperature is lower. Following discussions analyse this behavior; however, the most influential factor is the pressure distribution in the radiator block and the resultant recirculation flows which cause less oil to reach the last radiators. Subsequently, the oil velocity is lower in that area, and there is more time available for heat exchange between the oil and the
surrounding air. Recirculation flows creates dead zones in the headers and strongly affect the cooling capacity of the radiators. The heat transfer mechanism in these regions is mainly due to heat conduction inside the oil and, as a result, the cooling efficiency is reduced.

The temperature distribution in the middle section of the radiator block is presented in figure 4-16. The result of simulation is quite representative for this case. The temperature is almost constant in the upper header, while in the lower header it has different profile.

![Simulated radiator block](image1)

![Temperature contour on the outer surface of radiator](image2)

In the right side radiators, the exit temperature is lower than in the left side radiators. The average temperature reduction is about 12°C.

The cooling capacity of the block, derived using Fluent, was about 8602 watts. This was the total heat rate from the oil in the blocks to the surrounding air. The studied power transformer was equipped with 12 radiator blocks. Therefore, the total cooling capacity may be considered as $8602 \times 12 = 103244$ watts. Power transformer documents provided by the manufacturer indicate that cooling capacity should be calculated according to the following correlation:

$$P_{55} = P_{\text{total}} \times \left(\frac{55}{\theta_{0,max}}\right)^{\frac{1}{2}}$$

(4-7)

where from transformer documents the following data were extracted:

$P_{55} = 112$ kW : Required cooling capacity for a 55°C rated transformer (55°C average winding temperature rise by resistance)

$\theta_{0,max} = 50.3$°C : Maximum operational winding temperature rise

$x = 0.7$ : Coefficient for ONAN case

Applying these values to the equation 4-7 yields $P_{\text{total}} \approx 98.6$ kW. Therefore the difference is less than 5%, which is acceptable considering the assumptions applied during simulation.
In figure 4-17, the velocity vectors in the middle section of the radiator block are presented. The figure shows that flow recirculation starts at the middle of the upper header, and thus only a small amount of inlet oil flow passes through the last radiators.

Figure 4-16: Temperature contour of oil (°C) in the vertical middle surface of radiators block.

The velocity magnitudes are quite small, which is normal in cooling systems driven by buoyancy forces. This is one benefit of numerical modeling: it is possible to calculate the velocity, as in reality, there is no oil flow measuring in the transformers.

Figure 4-17: Contour of velocity vector in the vertical middle surface of radiators block.

The pressure distributions in the upper and lower headers are presented in figures 4-18 and 4-19, respectively. The static pressure increases as the oil flows through the header, and at the end of the upper
header there are disturbances in the pressure distribution resulting from recirculation flow. A comparison between the two graphs shows that the static pressure difference between upper and lower headers also increases through the header. Thus the oil flow is less in the right side of the radiators. Subsequently, it is expected that the velocity distribution through the radiators varies greatly.

Figure 4-18: Pressure distribution in the inlet header.

Figure 4-19: Pressure distribution in the outlet header.

4.3.1.2 Cooling radiators

A radiator module, as one important part in the power transformer cooling system, was investigated in depth. Figure 4-20 shows a typical radiator that is used in cooling system construction. The radiator model on the right side of the figure shows a half part, as a symmetric condition is present for radiators and this property may be used to simplify the modeling. Two main cases are considered: one, a radiator with simple oil channels and two, a radiator having oil channels fitted internally with pin fins. More details about modeling techniques may be found in the attached paper. However, here the important results are highlighted.

4.3.1.2.1 Validity of models

Using the Newton–Raphson iterative procedure for theoretical model and finite volume numerical procedure, the governing non-linear equations were solved to find the oil outlet temperature in different cases. Figure 4-21 shows the comparison between calculated temperatures drop values and available experimental data from the manufacturer (Iran Transfor Corporation). Although experimental data and modeling results differ slightly, the diagram confirms that the constructed model is sufficiently capable of predicting the thermal behavior of the studied radiator.

Reasons for slight variations between the modeling and the experimental results can include:

1. errors within the parameters measurement (experimental errors);
2. intrinsic numerical errors due to numerical operations such as error truncation (modeling errors);
3. errors due to modeling simplification and assumptions, addressed in Appendix Paper 3 (modeling errors).
To minimize the CFD modeling errors, the mesh dependency of the solution was examined by solving the flow and temperature fields for different mesh configurations made of different cells. These profiles were compared in several sections for all configurations to ensure that the maximum difference in the flow field properties between the coarser and finer meshes were less than 1%, and the final mesh lead to mesh-independent solutions.

4.3.1.2.2 Discussion on results

Oil flow rate through a radiator is governed by available buoyancy forces and the resistance in the flow paths. There is no experimental data available for oil flow through the radiators, as it has a small magnitude and there is no clear reason for manufacturers to measure this flow.

Figures 4-22 and 4-23 show the average oil flow rate and velocity in the radiator channels at a horizontal surface placed exactly in the middle of the radiator element for a radiator with simple channels. The oil flow rate shows a linear relation with inlet oil temperature. A 1% increase in temperature will subsequently increase the oil flow rate by about 1.6%.
Channels equipped with internal pin fins have less oil flow in comparison to the simple channels. Fin installation causes the area for flow passage to decrease, and thus pressure drop through the oil channels increases. This means that the pressure loss through the channels with pin fins is higher. The reason is that the flow passage cross-section area decreases in the sections where fins are inserted. This is a positive effect, as oil has more time to exchange heat with ambient air. As the inlet temperature increases, this effect decreases. Figures 4-24 and 4-25 show a comparison of oil flow and temperature between two different channels versus the inlet temperature. The temperatures levels are lower in the pin fin case; however, the maximum temperature drop in comparison to the simple case is 5%. Since the other cooling media on the outside of the radiator is air, any attempt to increase the convection heat transfer coefficient inside the oil channels has a minor effect on the heat transfer. Considering this fact, fins also provide a bridge between radiator plates and heat is dissipated through the fins by conduction. The effect of fins on the internal convective heat transfer is not an effective parameter in the whole heat transfer. Figure 4-26 shows that the installation of pin fins increases the cooling capacity by about 5 to 10% in the simple radiator.
4.3.2 Application in CO₂/H₂O condenser

This section investigates the computational results that were obtained by applying the methods described in Chapter 3. A schematic of the oxy-fuel process and the 2D model considered for condensation study is presented in figure 4-27. A primary evaluation showed that it was necessary to develop the condensation model based on a simple geometry and then develop it for use with the more complex geometries. The study began with the solution of governing equations on a vertical surface. Then, a comparison of obtained results with available experimental data from literature was performed and final discussions on different parameters ensued.

First, a model was run using a binary mixture of air and water vapour. This was done first to evaluate the model and to compare the results with available data. The heat transfer coefficient is presented in figure 4-28 as a function of mass fraction of air at the inlet. A comparison of average heat transfer coefficients with available correlations shows that the computational results match the trends given by these expressions. This means if the average values are considered, the diffusive mass flux calculated...
in the modeling corresponds to the physical results. The model was evaluated and corrected until there was a good conformity between experimental equations and results obtained from numerical modeling. Then the flue gas mixture was changed to CO₂/H₂O and appropriate material properties were supplied.

Figure 4-28: Comparing model results with experimental data ($\Delta T_{w-g} = 18^\circ C$, $P = 1$ bar).

Condensation rate depends on several parameters and among them, the inlet velocity, inlet mole fraction, and inlet temperature are the most important ones from a process point of view. Modeling results show that the condensation rate has a greater sensitivity to inlet velocity than average heat transfer coefficient, considering the fact that an ideal diffusion coefficient is used for modeling (Figure 4-29 and Figure 4-30). As it seen in Figure 4-30, there is a direct relationship between inlet velocity and the rate of condensation. Also, lower inlet temperature causes a higher condensation rate. This condition is natural, as water vapour reaches a saturation temperature with less heat removal.

As noted in Chapter 3, the presence of CO₂ as a non-condensable gas in the flue gas stream causes the formation of a concentration boundary layer. This concentration boundary layer, as well as the thermal and velocity boundary layers, affects the rate of water condensation.

The condensation rate decreases along the plate length, as these boundary layers grow in the flow direction. Two condensation models were developed, one based on heat and mass transfer analogy and one based on diffusion boundary layer theory. These models are described in detail in Appendix Paper 7.

Figure 4-29: Average heat transfer coefficient versus inlet velocity and CO₂ mass fraction

Figure 4-30: Condensation rate as a function of inlet velocity and inlet temperature
The heat transfer coefficient is presented in Figure 4-31 as a function of mass fraction of air at the inlet. As observed, the trend of both numerical condensation models is similar to the experimental correlation results (Dehbi, et al., 1991). Even though experimental data and modeling results differ slightly, the given diagram confirms that presented models are relatively capable of predicting the condensation behavior of such condensers.

![Figure 4-31: Verification of model validity (air/water vapour mixture)'](image)

The prediction of condensation rate is illustrated in Figure 4-32. The trend of the condensation rate (blue line) is consistent with theory (red line). As the CO₂ mass fraction increases, the condensation rate decreases sharply. The reasons are that first, there is less water content in the flow gas and second, the diffusion of water vapour toward the cooling surface became more difficult.

![Figure 4-32: Comparison of condensation rate for two condensation models.](image)

Figure 4-33 shows the results obtained using a model based on diffusion boundary layer theory. It can be seen that at velocities greater than 1.5 m/s, the velocity effects are negligible. However, when the CO₂ mass fraction is lower at the inlet, velocity can be an affecting parameter as well. The trend was found to be the same for lower CO₂ fractions.
The final case considered was a vertical plate equipped with pin fins. Figure 4-34 shows the comparison of the total heat transfer coefficient in the simple condensing plate and the surface with pin fins. Clearly, results show that the difference in coefficient values is negligible. This means that the developed models are not really accurate for predicting the exact heat transfer coefficient in complicated geometries. The reason is that the model looks only into the cell adjacent to the wall, and the geometry is not accounted for. Moreover, the correlations that were implemented were developed for a vertical positioning, and some surfaces on fins are horizontal.

4.4 CFD capabilities

The numerical method was successfully implemented in single phase process applications, and results showed good agreement with available experimental data or theoretical correlations. The current situation of a cooling system for a power transformer was studied and based on simulation results, and proposals were made for performance improvement. However, it was found that the application of numerical methods to two-phase processes is a challenging issue. In such cases, the effort and consuming time is much higher than in the single phase case. Also data validation was more difficult since for such applications there are fewer experimental data and theoretical correlations.
This approach was implemented for a basic case involving a CO₂/H₂O condenser, and the condensation process was investigated. It was shown that models based on numerical approaches are capable of predicting both the trends in the condensation process and the effect of the CO₂ presence in the flue gas as a non-condensable gas. However, experience during this research showed that this kind of simulation requires quite a bit of expertise. In addition, the modification of currently available models to suit the condensation process (itself a very complex process) would require both a good understanding of the physical process and a good knowledge of Fluent. The accuracy of the numerical code results depend on the empirical correlations specified to model the condensation process. In industry, there is a practice to model the process with some correlations (available in the open literature) and then to tweak various parameters in order to generate a good agreement with the experimental results. Such a fine-tuning is necessary in numerical modeling, as the general correlations may not yield accurate results for a specific set-up. Based on these research findings, it must be asserted that designing a condenser based solely on numerical results may be a difficult and expensive task.
5. Conclusion

5.1 General discussions
This worked started with an evaluation of pin fins as extended surfaces for heat transfer enhancement. A numerical model based on governing physical equations was built. This model was verified using a theoretical energy balance equation. Then it was implemented in a power transformer cooling system and verified according to operational data from the manufacturer. Then, the verified model was applied to a more complicated two-phase flow modeling process to evaluate the pin fin effect in condensation efficiency. In summary, this thesis has focused on the following areas:

- Numerical analysis of the fluid flow and heat transfer in a duct equipped with different forms fins in order to identify the optimum fin shape;
- Investigation of the applicability of CFD methods in various industrial applications;
- Full-scale simulation of power transformer cooling using optimization based on the experiences gained during this work;
- Simulation of flow behavior inside a full-scale condenser design for final use in CO\textsubscript{2} capturing application;
- Basic study of numerical simulation for water vapour condensation from a binary mixture, relevant to the CO\textsubscript{2} capturing process.

The impact of the fin form on the flow and heat transfer was found to be very significant. Moreover, the inlet boundary condition and flow velocity have a significant influence on the thermo-hydraulic behavior of the heat exchanger. The sensitivity to this condition is greater in the low Reynolds flows.

A general review of the results shows that of the three tested models, a drop-shaped pin fin array has the best heat transfer in combination with the lowest pressure loss. With equal pin fin surface area for all morphologies, cylindrical pins have the maximum volume, which increases the weight; this factor should be considered for optimization. Rectangular pins have the minimum weight, which is beneficial from a weight point of view.

Theoretical and numerical studies show that, in principle, it is possible to model the surface condensation using Fluent. However, this requires high-quality experimental data for insertion into the model as boundaries. Also, this would necessitate the use of a Eulerian model, one of the most advanced models in Fluent and requiring quite a bit of experience to handle and set to suit the condensation process (which itself is a very complex process). This is not an easy job when it comes to assessing a real condenser that involves complex geometry, and in most cases it is not possible to produce structured mesh for it. In the industry, the practice is to model the process with certain correlations and then tweak the parameters to get a good agreement with the experimental results. This is necessary in most cases, as the general correlations may not yield accurate results for a specific set up. The situation would be the same for a numerical simulation. It is required to make the model similar to an existing experimental setup in order to verify results which are not an easy case.

It would be advisable to understand that choosing a condenser for applications such as oxy fuel plants (as noted in the introduction) based on Fluent results may be a difficult task, especially if experimental data is not available, since it would become difficult to validate the empirical correlation used. Therefore, this would be a highly expensive method for studying this kind of flow regime. However, it is possible to study the flow field and temperature profile in the real condenser. Then a simple plane
condensation model such as the one presented in this study can be used for parametric study using real data (operation pressure, mixture gas composition, and inlet temperature and inlet mass flow rate) for the flue gas.

5.2 Future work

The trend in industry for designing and optimizing new heat exchangers quickly and inexpensively makes the CFD an important tool in many industrial applications. Fields of future interest which may be identified through this research are presented in this section.

When a pin fin simulation is performed, it is important to study the different fin arrangements in the flow and to provide general tables and correlations for designers. This information already exists for simple and standard fins in the textbooks and standards, but with the help of the CFD, it would be easy to provide such information for a wider variety of complicated fin geometries. Moreover, the effect of fin material (in addition to its geometry) plays an important role in heat transfer, and different materials can be used as conjunctions between pin fin and tube. An exploration of this area may be especially important from a production point of view.

Regarding the specific industrial cases studied for this thesis, the following issues might be considered for further investigation:

In the power transformer cooling system, it would be of great value to investigate the oil channels forms and to determine an optimized form with fin technology in order to design the best cooling radiator from performance point of view. This study would have a great advantage for industry, since there is a sharp demand for power transformer production as a result of world demand for more electricity. Currently, most of these power transformers use a conventional cooling system with an older design; any attempt to improve that design would result in enormous energy saving.

For the CO₂ capturing application, various areas of exploration exist. Flue gas in oxy-fuel plants contains elements other than CO₂ and H₂O, depending on the fuel that is used. Therefore condensation simulation in a multi-component stream can deliver more reliable results.

Additionally, the provided simulation models and methodologies can be used for other applications where redesign and optimization is required. Some examples include selective catalytic reduction (SCR) in new power plants, new heat exchangers in CO₂ capturing applications, and performance improvement in applications which use internal or external fins.

In closing, more research work is needed to clearly create a general numerical code that can be used with commercial available CFD software to minimize the effort on UDF preparation for each specific case.
6. References


d Activities at ChalmersUniversity [Conference] // 2nd Oxy-Combustion Workshop at
Windsor. - CT, USA : [s.n.], 2007.

Atkinson K.N. [et al.] Two- and three dimensional numerical models of flow and heat transfer over

Bergles A.E. Some perspectives on enhanced heat transfer – second generation heat transfer

Carey V.P. Surface tension effects on convective boiling heat transfer on compact heat exchangers
970-974.


Cengel Yunus A. and Boles Michael A. Thermodynamics, An Engineering Approach [Book]. -


Charlotte Wilhelmsson Jinliang Yuan, Bengt Sundén Water condensation and two-phase flow
modeling for a plate heat exchanger channel [Conference] // ASME International Mechanical
Engineering Congress and Exposition. - Seattle, USA : [s.n.], 2007.

Charlotte Wilhelmsson, Jinliang Yuan and Bengt Sundén Water condensation and two-phase flow
modeling for a plate heat exchanger channel [Conference] // ASME International Mechanical
Engineering Congress and Exposition. - Seattle, USA : [s.n.], 2007.

Choi K. Y., Chung H. J. and No H. C. Direct – Contact Condensation Heat Transfer Model in
RELAP5/MOD3.2 with/without Non - Condensable Gases for Horizontally Stratified Flow
[Conference] // International Topical Meeting on Nuclear Reactor Thermal-Hydraulics

Chou F.C., Lukes J.R. and Tien C.L. Heat transfer enhancement by fins in the microscale regime

Christopher L. Chapman, Seri Lee and Bill Schmidt L. Thermal Performance of an Elliptical Pin
Department of Physics, Div. of Nuclear Reactor Technology ; KTH. - Stockholm : [s.n.], 2007.


7. Papers Summary

**Paper 1:** “Investigation on Numerical Modeling of Water Vapour Condensation from a Flue Gas with High CO2 Content” 

Contributions from Hamid Nabati: Main author, underlying theory study, modeling and simulations, discussion on results in collaboration with Jafar Mahmoudi.

In this paper, the numerical modelling of condensation process of water vapour from a flue gas with high CO2 concentration is studied. To simplify the study and focus on the physical model, a simple vertical plate was chosen. Two condensation models are developed. A numerical approach is considered to implement these models. The main objective in the current paper was to study the capability of numerical model in prediction of complex process. Results showed that both developed models are capable to predict the trends in condensation process. However, the model based on boundary layer theory shows closer value to experimental correlation. Experiences during the modeling showed that surface contact condensers modeling with numerical approaches and Fluent© requires the Eulerian model. This Eulerian multiphase model is an advanced model of Fluent and requires quite a bit of experience to handle. In addition, modification of these model to suit condensation process, which itself is a very complex process, would require both, good understanding of the physical process and good knowledge of model inside the Fluent. The accuracy of the Fluent results depend on the empirical correlations specified to model the condensation process. In the industry, there is a practice to model the process with some correlations available in the open literature and then tweak various parameters to get a good agreement with the experimental results. Such a tuning is necessary in numerical modeling as well for most of the cases, as the general correlations may not yield accurate results for a specific set up. It is advisable that designing a condenser just based on Numerical results may be a difficult and expensive task. However, results from this study can be developed further to be used in designing of condenser which is suitable for oxy-fuel power plants.

**Paper 2:** “Numerical and Theoretical Investigation on Cooling Performance of Radiators in the Power Transformers” 

Contributions from Hamid Nabati: Main author, data collection, modeling and simulations, parameters identification and analysis in collaboration with Jafar Mahmoudi.

In the third paper, we have studied the heat transfer and fluid flow in a cooling system of power transformers. In order to simplify the study, a single radiator element is chosen which is composed of two welded identical steel plates. Also based on symmetrical property of radiator, just one quarter of it, was modelled and simulated. It is well known that phenomenon like Foucault currents generate unwanted loss in form of heat in the transformers. From engineering and design points of view, it is desired to establish efficient cooling in transformers efficiently in order to increase their operation life. The main objective in the current paper is to study the current situation of cooling system and its characteristics to identify the weakness points of installed cooling system. Also a new configuration for oil
channels based on pin fin technology was studied to evaluate its capability for cooling efficiency improvement in this special case. Result can be used for optimization of power transformer cooling system in future studies. To accomplish these goals, a basic model based on theoretical correlation and a numerical study was performed for radiator model in the ONAN state. The physical properties of the cooling oil were calculated from technical drawing obtained from Iran Transfo Company. The finite volume method was used to solve the fluid and heat transfer governing equations in the steady state. Results show that there is a good agreement between available data, theoretical model and CFD model for oil channel in the current configuration. Heat transfer in ONAN state is within acceptable range, but the oil entrance and outlet passages should be redesigned to get better flow behaviour. The radiator height has a direct effect on outlet temperature; however the maximum reduction in the outlet temperature would 7%, when the height is doubling. The radiator equipped with internal pin fins shows better heat transfer behaviour, while the pressure drop through the channels remains in acceptable range.

**Paper 3:** “Heat Transfer and Fluid Flow Analysis of Power Transformer's Cooling System Using CFD Approach”


*Contributions from Hamid Nabati:* Main author, data collection, modeling and simulations, parameters identification and analysis in collaboration with Jafar Mahmoudi and Ali Ehteram.

This paper presents the results of numerical modeling of temperature distribution and flow pattern in a block radiator used in power transformer cooling system. Each block radiator consists of eighteen plane radiators which are parallel together and a typical power transformer (like a 30MVA) has 6 radiator blocks in each side which means it has 12 blocks totally. The numerical study using Fluent software has been conducted to find the explanation of low cooling efficiency in this special power transformer currently used in industry. Our main desire is the study of relation between radiator block characteristics and cooling behavior of system which can be used for its optimization in future studies. The results indicate that recirculation occurs whenever pressure increase at the end of a radiator block that consequently prevents the enough oil flow through the last radiator of block. Experimental data taken from company and technical data extracted from transformer documents have been used for model calibration and results verification.

**Paper 4:** “Evaluation of CFD Method for Efficiency Improvement in CO2 Capturing with Application of Pin Fin Technology”


*Contributions from Hamid Nabati:* Main author, data collection, modeling and simulations and results analysis in collaboration with Jafar Mahmoudi.

This paper presents the results of a primary numerical study on heat transfer in a heat exchanger, considering different shape pin fins. This study was conducted to select the optimum pin shape to maximize the heat transfer and minimize the pressure drop across the heat exchanger. Results from this study then are used in a feasibility study on design optimization for a condenser required for oxy-fuel process. So, different available numerical model for multiphase modeling are investigated first. Then a primary design concept is proposed and water vapour condensation from a flue gas containing CO2 and H2O is studied upon it. The numerical model is applied to predict the flow field and heat transfer inside the condenser to investigate on condensation rate and CO2 separation efficiency. Then based on this primary study result, effect of pin fins on condensation is discussed.