Cfd modeling of compact offset strip-fin high temperature heat exchanger

Sundaresan Subramanian

University of Nevada, Las Vegas

Follow this and additional works at: https://digitalscholarship.unlv.edu/rtds

Repository Citation

https://digitalscholarship.unlv.edu/rtds/1855
The Thesis prepared by
Sundaresan Subramanian

Entitled
CFD Modeling of Compact Offset Strip-Fin High Temperature Heat Exchanger

is approved in partial fulfillment of the requirements for the degree of
Master of Science in Mechanical Engineering

Examination Committee Chair

Dean of the Graduate College
ABSTRACT

CFD Modeling of Compact Offset Strip-fin High Temperature Heat Exchanger

by

Sundaresan Subramanian

Dr. Yitung Chen, Examination Committee Chair
Associate Professor, Department of Mechanical Engineering
University of Nevada, Las Vegas

This thesis deals with the development of a numerical model to predict the overall performance of an advanced high temperature heat exchanger (HTHX) design, up to 1000°C, for the production of hydrogen by the sulfur iodine thermo-chemical cycle. The present study considers an offset strip-fin type compact high temperature heat exchanger made of liquid silicon impregnated carbon composite (SiC). The ceramic matrix composite material (CMC) is manufactured by impregnating the silicon into the pores of the carbon composites. The prototype heat exchanger is designed to operate at a thermal power of 50 MW. The design is an offset strip-fin, hybrid plate compact heat exchanger. The two working fluids are helium gas and liquid salt (FLINAK). The offset strip-fin is chosen as a method of heat transfer enhancement because of its ability to induce periodic boundary layer restart mechanism between the fins that has a direct effect on heat transfer enhancement. The effects of the fin geometry on the flow field and heat transfer are studied in three-dimensions using numerical techniques, and the results are then compared with the results from the analytical calculations. The pre-processor GAMBIT is
used to create a computational mesh, and the CFD software package FLUENT that is based on the finite volume method is used to produce the numerical results. The equations governing the flow and heat transfer are solved numerically using finite volume techniques, additional transport equations are also solved when the flow is turbulent.

The fluid flow inside the heat exchanger channels is considered to be incompressible and steady. The standard K-ω model is used for modeling turbulence while the conjugate heat transfer model is used for solving the energy equation. The heat exchanger channel is characterized by the presence of the fins mounted in a staggered fashion along the flow direction. For the present study both rectangular edged and curved edged fin channels are considered. The model developed in this thesis will be used to investigate the heat exchanger design parameters in order to find an optimal design. Also numerical simulation results were performed and compared to study the effect of the temperature dependent physical properties.

Comparison of the overall performance between two fin shapes (rectangular versus curved edges) is performed using numerical techniques. The model developed in this paper will be used to investigate the heat exchanger design parameters in order to find an optimal design.
TABLE OF CONTENTS

ABSTRACT .................................................................................................................................iv

LIST OF FIGURES ................................................................................................................viii

LIST OF TABLES ........................................................................................................................x

NOMENCLATURE ...................................................................................................................xi

ACKNOWLEDGEMENTS .......................................................................................................xiv

CHAPTER 1 INTRODUCTION .................................................................................................1

1.1 Motivation and scope .......................................................................................................1
1.2 IHX requirements .............................................................................................................3
1.3 Research objectives .........................................................................................................5
1.4 Literature review ..............................................................................................................5
1.5 Outline of thesis ............................................................................................................11

CHAPTER 2 DESCRIPTION OF THE PROBLEM AND GEOMETRY ..................................12

2.1 Offset strip-fin channel dimensions .........................................................................15
2.2 Governing equations ....................................................................................................16
2.3 Modeling approach ......................................................................................................17

CHAPTER 3 NUMERICAL METHOD AND ALGORITHM .....................................................18

3.1 Finite volume method .................................................................................................18
3.2 Solution algorithm ........................................................................................................19
3.3 Discretization ................................................................................................................21
3.4 Modeling periodic flows in a heat exchanger channel ..............................................22
3.5 Turbulence modeling ...................................................................................................25
3.6 Grid considerations for turbulent flow simulations ....................................................31
3.7 Defining turbulence boundary conditions ..................................................................31
3.8 Solution strategy for turbulent flow simulations .......................................................31
3.9 Boundary conditions ....................................................................................................32
3.10 Physical properties model ........................................................................................35
3.11 Setting under-relaxation factors for solution ..............................................................41

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
LIST OF FIGURES

Figure 1.1 Schematic diagram of the plant concept ................................................................. 2
Figure 1.2 Plan view of the offset strip fin channel ................................................................. 6
Figure 2.1 Flow channel of the candidate offset strip fin heat exchanger ......................... 14
Figure 2.2 One module of the candidate offset strip fin heat exchanger ......................... 14
Figure 3.1 Variation of thermal conductivity of helium with temperature ....................... 37
Figure 3.2 Variation of density of helium with temperature ............................................... 38
Figure 3.3 Variation of dynamic viscosity of helium with temperature ......................... 39
Figure 3.4 Variations of density of FLINAK with temperature ....................................... 40
Figure 3.5 Variation of Dynamic Viscosity of FLINAK with temperature ....................... 41
Figure 4.1 Geometry of the calculation domain (solid part) .............................................. 42
Figure 4.2 Static pressure distribution (LS part), Pa ......................................................... 44
Figure 4.3 Static pressure distribution (He part), Pa ......................................................... 45
Figure 4.4 Static pressure distribution (LS part), Pa ......................................................... 46
Figure 4.5 Static pressure distribution (He part), Pa ......................................................... 46
Figure 4.6 Geometry of the one section (with fillets) ....................................................... 53
Figure 4.7 Non-rounded geometry (without fillets) ......................................................... 54
Figure 4.8 Static pressure distribution (He part), Pa ......................................................... 55
Figure 4.9 Static temperature distribution (He part), K ...................................................... 55
Figure 4.10 Static pressure distribution (LS part), Pa ....................................................... 56
Figure 4.11 Static temperature distribution (LS part), K .................................................... 56
Figure 5.1 Computational domain considered in the periodic flow simulations ............. 59
Figure 5.2 Computational mesh for the 2-D numerical simulations ................................ 60
Figure 5.3 Velocity vectors and pressure contours for rounded fin edge channel .......... 61
Figure 5.4 Velocity vectors and pressure contours for rectangular fin edge channel .... 61
Figure 5.5 Stream line plots showing vortices at different Re values ............................... 63
Figure 5.6a Surface heat transfer coefficient along fin wall (Gap = 1 mm) ....................... 64
Figure 5.6b Surface heat transfer coefficient along fin wall (Gap = 3 mm) ....................... 65
Figure 5.7 Variation of pumping power with gap for periodic fluid flow with a constant fin length ............................................................... 66
Figure 5.8 Variation of pumping power with gap for periodic fluid flow with a constant pitch ......................................................................................... 67
Figure 5.9 Computational domain considered for the flow across the full channel length ........................................................................................................ 68
Figure 5.10 Velocity and Temperature contours for the baseline case with curved fin channels ........................................................................................................ 69
Figure 5.11 Plot for pressure drop across 37 module heat exchanger channel .............. 70
Figure 5.12 Plot for temperature drop across 37 module heat exchanger channel .......... 70
Figure 5.13 Variation of pumping power with gap for full channel numerical simulation with a constant pitch .................................................. 72
Figure 5.14 Variation of thermal power with gap for full length channel numerical simulation with a constant pitch .................................................. 72
Figure 6.1 Helium channel mesh ................................................................................................... 76
Figure 6.2 Liquid salt channel mesh ............................................................................................. 77
Figure 6.3 Geometry of the one section (with fillets) ................................................................. 87
Figure 6.4 Cross section of the channel geometries considered in the analytical correlations ........................................................................................................... 90
Figure 7.1 Heat exchanger length vs Overall thermal power .................................................. 98
Figure 7.2 Heat exchanger module showing the thickness parameter under study .......... 99
Figure 7.3 Fin thickness vs Thermal power for helium channel ........................................... 99
Figure 7.4 Fin thickness vs Thermal power for liquid salt channel ...................................... 100
Figure 7.5 Fin thickness vs Pressure drop for helium channel ............................................... 101
Figure 7.6 Fin thickness vs Pressure drop for liquid salt channel .......................................... 101
Figure 7.7 Heat exchanger module showing the fin height parameter under study .......... 102
Figure 7.8 Channel height vs thermal power both channels ................................................. 103
Figure 7.9 Channel height vs Pressure drop for both channels ............................................ 104
Figure 7.10 Heat exchanger module showing the pitch in x-direction parameter under study .......................................................... 105
Figure 7.11 X-direction pitch vs Thermal power for both channels ..................................... 105
Figure 7.12 X-direction pitch vs Pressure drop for both the channels .................................. 106
Figure 7.13 Heat exchanger module showing the gap between fins in flow direction parameter under study .......................................................... 107
Figure 7.14 Gap length vs Thermal power for both channels .............................................. 107
Figure 7.15 Gap length vs Pressure drop for both channels ................................................. 109
Figure 7.16 Heat exchanger module showing the fin length parameter under study .......... 109
Figure 7.17 Fin length vs Thermal power for both channels ............................................... 110
Figure 7.18 Fin length vs Pressure drop for both channels .................................................... 111
LIST OF TABLES

Table 2.1 IHX operating conditions................................................................. 12
Table 2.2 Baseline Heat Exchanger Channel Dimensions...................................... 15
Table 5.1 Results of the 2-D Numerical calculations for the baseline designs........... 60
Table 5.2 Summary of the 2-D simulation results for the helium side performance...... 71
Table 6.1 Numerical solution results for the helium channel for the two
  baseline geometries with incompressible laminar model .................................. 78
Table 6.2 Numerical solution results for the liquid salt channel for the two
  baseline geometries ............................................................................................ 78
Table 6.3 Numerical results for the helium channel for the two baseline
  geometries with temperature dependent physical properties ........................... 79
Table 6.4 Numerical results for the LS channel for the two baseline
  geometries with temperature dependent physical properties ........................... 79
Table 6.5 Heat Exchanger Overall Performance from numerical calculations
  for rectangular fin edge case .............................................................................. 80
Table 6.6 Heat Exchanger Overall Performance from numerical calculations
  for curved fin edge case .................................................................................... 80
Table 6.7 Numerical solution results for the helium channel for the two
  baseline geometries ............................................................................................ 83
Table 6.8 Numerical solution results for the liquid salt channel for the two
  baseline geometries ............................................................................................ 83
Table 6.9 Heat exchanger overall performance comparison between the laminar
  and turbulence numerical models calculations for curved fin edge case ........... 84
Table 6.10 Heat exchanger overall performance comparison between the laminar and
  turbulent numerical models calculations for rectangular fin edge case .......... 85
Table 6.11 Comparison of numerical solution results for the He side laminar model ... 87
Table 6.12 Comparison of the numerical solution results for the LS side ................. 88
Table 6.13 Numerical results comparison for He side (turbulent case) ................. 88
Table 6.14 Numerical results comparison for LS side (turbulent case) ................. 89
Table 6.15 Summary of the comparison between three-dimensional numerical
  and analytical calculation for rectangular fin edge helium channel ............... 92
Table 6.16 Summary of the comparison between three-dimensional numerical
  and analytical calculation for curved fin edge helium channel ..................... 92
Table 6.17 Summary of the comparison between three-dimensional numerical
  and analytical calculation for rectangular fin edge liquid salt channel .......... 92
Table 6.18 Summary of the comparison between three-dimensional numerical
  and analytical calculation for curved fin edge liquid salt channel ............... 93
Table 7.1 Numerical solution results for the two baseline geometries ................. 97
NOMENCLATURE

\( C_p \) \quad \text{coefficient of heat capacity, J/kgK}

\( D_h \) \quad \text{hydraulic diameter, m}

\( E \) \quad \text{total energy}

\( f \) \quad \text{fanning friction factor}

\( G_k \) \quad \text{turbulence kinetic energy due to mean velocity gradients}

\( G_w \) \quad \text{generation of kinetic energy}

\( h \) \quad \text{channel height, mm}

\( h_f \) \quad \text{heat transfer coefficient of fluid, W/m}^2\text{K}

\( j \) \quad \text{colburn factor}

\( k \) \quad \text{thermal conductivity of the fluid, W m}^{-1}\text{K}^{-1}

\( k \) \quad \text{turbulent kinetic energy}

\( l \) \quad \text{fin length parameter, mm}

\( \text{LMTD} \) \quad \text{logarithmic mean temperature difference, K}

\( \text{Nu} \) \quad \text{nusselt's number}

\( P \) \quad \text{Pressure, Pa}

\( \text{Pr} \) \quad \text{Prandtl number}
\( P_y \)  
Pitch in flow direction, mm

\( P_x \)  
Pitch in spanwise direction, mm

\( \Delta P \)  
pressure drop, kPa

\( q \)  
heat flux, W/m\(^2\)

\( Re \)  
Reynolds number

\( t \)  
fin thickness, mm

\( t \)  
time, s

\( T \)  
temperature, K

\( T_b \)  
bulk temperature of the fluid, K

\( T_f \)  
temperature of fluid, K

\( T_w \)  
wall temperature, K

\( T_s \)  
surface temperature, K

\( U \)  
velocity, m/s

\( u \)  
velocity vector, m/s

\( u_i \)  
x, y, z -components of velocity in tensor notation at \( i = 1, 2, 3 \), m/s

\( v \)  
y -component of velocity, m/s

\( w \)  
z -component of velocity, m/s

\( x_i \)  
x, y, z -coordinates at \( i = 1, 2, 3 \)

\( x \)  
horizontal Cartesian coordinate, m

\( y \)  
vertical Cartesian coordinate, m
\( Y_k \)  
\( Y_\omega \)  
\( z \)

- dissipation of \( k \)
- dissipation of \( \omega \)
- lateral Cartesian coordinate, m

Greek Symbols

\( \delta_{ij} \)  
\( \beta \)  
\( \mu \)  
\( \mu_t \)  
\( \rho \)

- Kronecker delta
- constant for pressure gradient
- dynamic viscosity, kg/ms
- turbulent dynamic viscosity, kg/ms
- density, kg/m\(^3\)
ACKNOWLEDGMENTS

I wish to thank all my committee members, and the various faculty members at UNLV who have encouraged and helped me over the past two years when I was working in this project.

Acknowledgement is due to Dr. Yitung Chen, my advisor and thesis committee chairman, and Dr. Anthony E. Hechanova, program manager and research advisor for their encouragement, inspiration and guidance. I owe them my gratitude for accommodating me in their research group and assigning such a challenging project. I would also like to acknowledge Dr. Per F. Peterson of the University of California, Berkeley for his collaboration in this project. I also extend my thanks to Dr. Robert Boehm and Dr. Jichun Li for providing their consent to be part of the thesis examination committee. I would also like to thank Dr. Hsuan-Tsung (Sean) Hsieh for reviewing my thesis, and fruitful suggestions about the thesis. Acknowledgement is due to the US Department of Energy for funding this project under the contract DE-FG04-01AL67356.

Special thanks go to Mr. Clayton DeLosier and Mr. Valery Ponyavin for their invaluable inputs and co-operation as a part of this research team. I would also like to thank Mr. Roald Akberov, my ex-colleague at NCACM. Without his inputs, it would have been extremely tough for me to work in this project.

I wish to express gratitude to my parents, my brother, my sister, and my friends for their encouragement and support.
CHAPTER 1

INTRODUCTION

1.1 Motivation and scope

Compact heat exchangers are used in a wide variety of applications such as: automobile radiators, air-conditioning systems, condensers, electronic cooling devices, recuperators, and regenerators, and cryogenic exchangers. As it is advantageous to have a lightweight and less space consuming heat exchangers, compact type heat exchangers are the most sought after. The need to design an intermediate high temperature heat exchanger (IHX) that acts as a interface between the nuclear reactor and the thermo-chemical loop that deals with the production of hydrogen as a part of the Nuclear Hydrogen Initiative (NHI) program sponsored by the US Department of Energy (DOE) gave the motivation for this work.

With the inevitable depletion of fossil fuels, hydrogen has been identified as a fuel storage medium for the future. Hydrogen can be an attractive energy carrier if it can be produced cleanly and in a cost-effective manner. Nuclear energy can be used as an abundant source of energy for the production of hydrogen through high temperature processes (up to 1000 °C). The Sulfur Iodine (S-I) Cycle, a high temperature process is a baseline candidate thermo-chemical process. It consists of three chemical reactions that result in the dissociation of water. These reactions are as follows:
\[ I_2 + SO_2 + 2H_2O \rightarrow 2HI + H_2SO_4 \quad (120^\circ C \text{ min.}) \]  \hspace{1cm} (1.1)

\[ H_2SO_4 \rightarrow H_2O + SO_2 + \frac{1}{2}O_2 \quad (850^\circ C \text{ min.}) \]  \hspace{1cm} (1.2)

\[ 2HI \rightarrow H_2 + I_2 \quad (450^\circ C \text{ min.}) \]  \hspace{1cm} (1.3)

Net Reaction: \[ H_2O \rightarrow H_2 + \frac{1}{2}O_2 \]  \hspace{1cm} (1.4)

Theoretically, only water and heat need to be added to the cycle. From the above chemical reactions one can see that the splitting of the water molecule by this method requires a temperature of at least 850°C. All of the reactants, other than water, are regenerated and recycled. Figure 1.1 shows a concept for driving the S-I process using process heat from a modular helium reactor (MHR). The intermediate heat exchanger (IHX) consists of heat exchanger modules housed within a vessel, along with the primary coolant circulator. Alternatively, the intermediate heat transfer fluid could be a high-temperature low-pressure liquid salt, depending upon tradeoffs among pumping power, heat exchanger mechanical design, materials performance, cost, and safety.

**Figure 1.1. Schematic diagram of the plant concept**
1.2 IHX Requirements

There are essentially four basic requirements for the advanced IHX (C.F. McDonald [1]), namely: (1) construction to assure leak tightness under all modes of operation, (2) on-line leakage monitoring capability, (3) higher temperature capability, and (4) compact heat exchanger capability for ease of installation. This research was concentrated upon choosing a compact heat exchanger that meets the requirements of 3 and 4.

Compact heat exchangers are characterized by extended surfaces with large surface area to volume ratios. Extended surfaces are provided in order to enhance the heat transfer rate; and there are many methods which are adopted for heat transfer enhancement. Some of the most commonly applied methods are providing fins, coiled tubes, and swirl flow devices. Plate fins and tube fins constitute the fin type of heat transfer enhancement. The plate fin type in turn has several other forms such as plain fins, offset strip fins, wavy fins, perforated fins, and pin fins. Of the above mentioned types of heat transfer enhancement methods the offset strip fin is the most widely applied.

In this project there was a necessity to design and develop a compact heat exchanger that could operate at high temperatures, in order to aid in the thermo-chemical process for hydrogen production. This thesis is concerned with the design of an offset strip fin heat exchanger for a high temperature application that is made from a liquid silicon impregnated carbon composite. The Ceramic Matrix Composite (CMC) material is manufactured by impregnating the silicon into the pores of the carbon composites. The prototype heat exchanger is designed to operate at a thermal power of 50 MW. This type of heat transfer enhancement method was chosen because the fins are able to create a
boundary layer restart mechanism; since, they behave as thin flat plates aligned along the
direction of flow, which in turn increases the heat transfer.

It is well known that the heat transfer coefficients in the entrance region of a duct are
substantially larger than those at locations farther downstream. This is because the
entrance region is characterized by thin thermal boundary layers; whereas, in the
downstream region transport of heat occurs across the entire duct cross section. The fact
that higher heat transfer coefficients are attainable in the entrance region has motivated
the design of heat exchanger flow passages which consist, in effect, of successive
entrance regions. The walls of such passages are periodically interrupted along the
streamwise direction. Each interruption enables the velocity and temperature distributions
to become more cross-sectionally homogeneous, and a new boundary layer is restarted
when the passage wall is resumed downstream of the interruption. This enhancement is
accompanied by an increase in pressure drop. Past research in this area was reviewed
intensively, and it was found that most of the work was numerical or experimental in
nature. The correlations that have been developed are based upon the experimental data
and are valid only for limited parameters.

As in most heat exchanger problems the working fluid, heat transfer rate, and mass
flow rate are usually known. If the concept of heat transfer enhancement is also known
then the problem is simplified to an optimization procedure. For which the perfect
correlations for the fanning friction factor \( f \) and the colbourn factor \( j \) are necessary.

Computational modeling of these flows has been used in the past and has become
increasingly popular, because it can provide detailed information about the heat transfer
mechanism, if the model is accurate. Numerical methods are more flexible and much
cheaper as it gives opportunity for testing new methods before they are executed through experiments, which would prove to be costly.

1.3 Research Objectives

The Department of Energy (DOE) gave the motivation for this work in support of the NGNP program that deals with the production of hydrogen. Some of the research objectives that have been outlined and which would benefit the development of a high temperature compact heat exchanger are:

- Choosing the best suited heat transfer correlations for the laminar, turbulent, and transition region
- Find a perfect correlation for the hydraulic diameter
- Obtain experimental data to validate the CFD results
- Optimization of the design
- Choosing the most suited materials and working fluids whose thermo-physical properties are temperature dependent
- Models need to be developed for several flow conditions and boundary conditions

1.4 Literature review

Over the past few decades a large amount of work has been conducted in the study of heat transfer and pressure drop characteristics of compact heat exchangers. Though various types of interrupted fin surfaces have been done in the past, this thesis focuses on the strip fin type compact heat exchanger. In the parallel plate fin heat exchanger there are two variations, they are the inline and staggered type. Although the problem outlined
mainly regards the staggered type, information regarding the inline type has also been considered. Staggered fins can be obtained by offsetting the strip fins (Figure 1.2) hence the name offset strip fin.

Offset strip fin heat exchangers are used as evaporators in the refrigeration and air conditioning industry (Carey [2]). The fins cause a recirculating flow between two successive fins (Rowley and Patankar [3]). Enhanced heat transfer is obtained due to the fins preventing the flow from becoming fully developed and the restarting of the boundary layer produces a higher heat transfer (Kelkar and Patankar [4]). The following paragraphs will discuss laminar and turbulent flow in offset strip fins. Rowley and Patankar [3] studied the laminar flow for a circular tube with circumferential internal fins was studied, and it was found that these fins often decreased the heat transfer coefficient because of the distribution of flow. The heat transfer was enhanced when the flow was turbulent. The work done by Dejong et al. [5] showed that steady laminar flow heat
transfer behavior was determined by boundary layer growth and that at higher Reynolds numbers vortex shedding must be taken into account. Various experimental and numerical studies were done to obtain the characteristics of the offset strip fin heat exchanger (Kelkar and Patankar [4], Maughan and Incropera [6], [7]). Maughan and Incropera [6], [7] presented two papers about convection heat transfer in a horizontal parallel plate with fins. The first paper (Maugan and Incropera, [6]) considered the numerical results while the second paper looked at the experimental results (Maugan and Incropera [7]). Kelkar and Patankar [4] used a numerical method to analyze the fluid flow and heat transfer for a tube with staggered fins. For fluids with low Prandtl number the arrangement of the fins had an influence on the heat transfer.

Kays and London [8] present the most needed experimental data. Most of the research work that has been performed in this area has been done with the assumption of zero fin thickness. Many different correlations for heat transfer have been reviewed during the literature survey. The other most important observation that was made is that only some research workers have given the transition region its due, as most papers fail to present a separate heat transfer correlation for the transition region.

Weiting [9] presented experimental data for 22 heat exchanger geometries with air as the working fluid and developed correlations for heat transfer in the laminar and turbulent region by the multiple regression method. Correlations for the laminar and turbulent regions and the critical Reynolds number were developed in the paper. Their relationships indicate that the flow passage aspect ratio is significant only in the laminar flow regime and the fin thickness parameter is significant only in the turbulent regime.
Also their correlations are applicable only for working fluids such as air with Prandtl numbers less than one.

Sparrow and Liu [10] performed numerical solutions for the fluid flow and energy equations using a finite difference method for a laminar airflow through arrays of inline and staggered plate segments, considering them as a single periodic unit. They assumed uniform temperature and velocity at the inlet and isothermal boundary conditions on the plate surfaces. The plates are assumed to be sufficiently thin so that thickness effects can be neglected. No slip conditions were assumed along all the solid boundaries and symmetric conditions were also considered as boundary conditions. Comparisons were made between the overall performance of inline and staggered fin arrays using constant mass flow rate and constant pumping power. In both cases the staggered fin configurations seemed to provide a higher effectiveness with a higher pressure drop.

Patankar et al. [11] conducted a numerical analysis on the heat transfer and fluid flow in channels whose walls are periodically interrupted along the streamwise direction. The momentum, mass, and energy equations were solved with constant fluid properties and for negligible viscous dissipation and compression work. The plates were assumed to be of negligible thickness, smooth edged, and isothermal. The results were employed to compare the heat transfer rates in heat exchangers composed either of interrupted-wall channels or of parallel plate channels. The heat transfer augmentation was markedly felt on relatively short heat exchanger channels and at high Reynolds numbers. The heat transfer predictions lie about 20-35% above the data, whereas the $f$ predictions lay 10-20% below the data.
Joshi and Webb [12] have made an attempt to identify the transition region by conducting flow visualization experiments and they also modified the correlations of Weiting [9]. The authors developed elaborate analytical models, which are based upon the predicted f and j characteristics. They also came up with a separated definition for the Reynolds number (Re*) in the transition region. Thus, the laminar, turbulent, and transition regions were identified.

Tinaut et al. [13] developed a prediction model for a water/engine oil compact heat exchanger, which predicts the heat exchanger performance and effects on various geometric parameters. Globally the expressions proposed have been found to be acceptable when comparing the results of the model to the experimental data.

Manglik and Bergles [14] studied 18 offset strip fin surfaces and analyzed the effect of the non-dimensional parameters on them, and arrived upon a correlation to describe all three regions. They reanalyzed all other different thermal hydraulic relationships and identified the asymptotic behavior in the laminar and turbulent regimes.

L.W. Zhang et al [15] investigated the heat transfer mechanism for both the inline and staggered array of strip fins. Finite fin thickness was assumed and correlations for the transition region were derived for different Reynolds number values. They studied the time-dependent flow behavior due to vortex shedding by solving two-dimensional and three-dimensional unsteady equations. The effect of vortices on the local Nusselts number and the overall heat transfer is studied. The results were compared to those of Sparrow and Liu [10].

Muzychka and Yovanovich [16] have made a thorough review of the other research work done and the different correlations for heat transfer. They have also compared their
model with the data available in the Kays and London [8]. The analytical models that they derived are based on Weiting [9], and Joshi et al [12] but simpler. Those models were found to be in correlation with the experimental data over a full range of Reynolds numbers. They have also shown how the correlations for friction factor by Manglik and Bergles [14] unpredicted experimental data at some Reynolds number regions. Their models were in correlation with the data presented by Kays and London [8].

Saidi and Sunden [17] conducted numerical analysis of the instantaneous flow and heat transfer for the offset strip fin geometry in self sustained time-dependent oscillatory flow. They also studied the effect of bubble on heat transfer at the fin surfaces and investigated the intermediate Reynolds number region unlike many other researchers.

Patankar and Prakash [18] presented a numerical analysis for the flow and heat transfer in an interrupted plate passage, which is an idealization of an offset fin heat exchanger channel, and compared the overall results with available experimental data. Their calculation method was based on the periodically fully developed flow through one periodic module, and the effect of plate thickness in the offset strip fins was studied. They also assumed stable laminar wake and used a constant heat flux boundary condition with the additional specification that each row of fins were at a fixed temperature. Their calculations have shown that by varying the fin thickness at fixed Reynolds number based on the hydraulic diameter, the flow pattern changes, resulting in overall heat transfer and friction loss. It was observed that only when the plate is sufficiently thick, the recirculation zones extend to the next plate. From their investigations it was concluded that a thick-plate situation leads to significantly higher pressure drop, while
the heat transfer does not sufficiently improve, despite the increased surface area and increased mean velocity.

Suzuki et al. [19] performed a two-dimensional numerical computation for unsteady flow and thermal fields around a three-row in-line fin array with the assumption of constant fluid properties. They studied the mechanism of heat transfer caused by flow instability and analyzed the instantaneous flow and thermal fields. Explanation was given for the effect of wake, which is an important mechanism on heat transfer enhancement.

Mochizuki et al. [20] did some experimental work on the offset strip fin cores and concluded that both $f$ and $j$ factors are enhanced with reduction in the fin length/plate spacing ratio at a given Reynolds number. The $f$ and $j$ correlations are derived for the offset strip fin cores.

Sara and Acharya [21] performed a numerical study to analyze the unsteady three-dimensional flow and conjugate heat transfer in a channel with in line and staggered arrays of periodically mounted square posts. They observed the flow to become unsteady for the staggered case, even at low Reynolds numbers.

1.5 Outline of thesis

Chapter 2 explains the details of the problem and the geometry and the governing equations associated. Chapter 3 numerical algorithm, the boundary conditions, and the physical properties while Chapter 4 focuses on the validation of the use of the numerical model used for solving the governing equations. Chapter 5 and 6 discusses the results for the different 2-D and 3-D geometries, respectively. Chapter 7 explains the optimizations studies performed while chapter 8 concludes the current research.
CHAPTER 2

DESCRIPTIONS OF THE PROBLEM AND GEOMETRY

The proposed design for the intermediate high temperature heat exchanger is a hybrid plate-type ceramic compact heat exchanger. The heat exchanger has a counter-flow arrangement and the flow channel is characterized by the presence of periodic interruptions in the form of rectangular or curved fins. The working fluids are helium gas (hot side) and liquid salt (cold side); the prescribed operating conditions of the heat exchanger are shown in Table 2.1 with the candidate working fluids dealt in this thesis highlighted.

Table 2.1. IHX operating conditions

<table>
<thead>
<tr>
<th>Primary/intermediate fluids</th>
<th>Helium/Helium</th>
<th>Helium/ Liquid salt</th>
</tr>
</thead>
<tbody>
<tr>
<td>Primary/intermediate pressures (MPa)</td>
<td>7.0/7.0</td>
<td>7.0/0.1</td>
</tr>
<tr>
<td>Primary inlet/outlet temperatures (°C)</td>
<td>1000/632</td>
<td>1000/632</td>
</tr>
<tr>
<td>Cold side inlet/outlet temperatures (°C)</td>
<td>560/975</td>
<td>560/975</td>
</tr>
</tbody>
</table>
The heat exchanger is designed to produce a thermal power of 50 MW, and the overall dimension of the heat exchanger is 1m x 1 m x 1m. One module of the heat exchanger under study is shown in Figure 2.1. The liquid salt considered is a fluoride based salt, which is commercially referred to as FLINAK (Fluoride-Lithium-Sodium-Potassium). The respective baseline channel dimensions are chosen considering the higher pumping power required for pumping gases, as opposed to a liquid. The considered design for the High Temperature Heat Exchanger (HTHX) has the offset arrangement of fins on both the hot and cold sides of the heat exchanger. The purpose of the heat exchanger is to decrease the temperature of the helium (He) from 1000 °C (1273.15 K) to 632 °C (905.15 K) by means of transferring the heat to the coolant fluid, liquid salt (LS), through the solid material of the wall. The considered liquid salt, FLINAK, has a melting temperature of about 100 K below the designed inlet temperature of 560 °C (833.15 K); thus, providing liquidity of the substance through the lengthy paths of the heat exchanger (0.9 m long, 1 mm high). By accepting heat from the He side through the solid material, the LS is designed to be heated in the micro channels almost up to the inlet temperature of helium, 975 °C (1248.15 K). In industrial applications all the fins of heat exchangers are normally designed to be of equal size and shape to maintain the geometric periodicity.
Figure 2.1 Flow channel of the candidate offset strip fin heat exchanger

The baseline design consists of 37 such periodic modules along the flow direction and they are also repeated numerously in the spanwise direction. The typical dimension of one module heat exchanger is shown in Figure 2.2.

Figure 2.2 One module of the candidate offset strip fin heat exchanger
2.1 Offset strip-fin Channel dimensions

Figure 2.2 shows the three-dimensional section of the geometry that was considered in the present study. Each fin has length (l), thickness (t), height (h), pitch in the x-direction (P_x), and pitch in the y-direction (P_y). Table 2.1 summarizes the flow channel dimensions for the baseline heat exchanger design. The dimensions that were chosen for the baseline heat exchanger geometry were based upon some initial sensitivity studies that were performed [22] based on the thermal design.

Table 2.2. Baseline Heat Exchanger Channel Dimensions

<table>
<thead>
<tr>
<th>Geometric parameters</th>
<th>Helium side (mm)</th>
<th>Liquid salt side (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fin length (l)</td>
<td>10</td>
<td>10</td>
</tr>
<tr>
<td>Channel height (h)</td>
<td>2</td>
<td>1</td>
</tr>
<tr>
<td>Fin thickness (t)</td>
<td>0.75</td>
<td>1.25</td>
</tr>
<tr>
<td>Pitch in flow direction (P_y = l + gap - length)</td>
<td>12</td>
<td>12</td>
</tr>
<tr>
<td>Pitch in span wise direction (P_x)</td>
<td>3</td>
<td>3</td>
</tr>
</tbody>
</table>

Since the heat exchanger geometry is different from those reviewed in the literature and due to the other constraints the analytical correlations that are derived in the literature do not suit the heat exchanger calculations under study. Hence, it was decided to use CFD techniques to perform the overall performance and optimization analysis of the heat exchanger. The fluid flow and heat transfer of the heat exchanger were performed using
FLUENT, a finite volume code based on a set of governing equations and boundary conditions.

2.2 Governing Equations

The fundamental equations governing the physical process of Newtonian viscous fluid flow are the continuity and momentum equations. For the convenience of utilizing a tensor notation the \( x, y, z \) coordinates will be denoted as \( x_1, x_2, x_3 \), and the \( u, v, w \) components of velocity as \( u_1, u_2, u_3 \).

Neglecting body forces, the continuity and momentum equations can be written in Cartesian tensor form as follows:

\[
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i) = 0
\]  \hspace{1cm} (2.1)

\[
\frac{\partial \left( \rho u_i \right)}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_l}{\partial x_l} \right) \right] + \rho g_i
\]  \hspace{1cm} (2.2)

where \( i, j = 1, 2, 3 \).

The momentum equation written in the above form is known as the Navier-Stokes equation. This set of equations is a general set of equations that, along with some additional model equations can be used for calculations of any Newtonian viscous fluid flow in Cartesian coordinates. While the governing equation for solving the temperature field is provided by

\[
\frac{\partial}{\partial t} (\rho E) + \frac{\partial}{\partial x_i} (\rho u_i (\rho E + p)) = \frac{\partial}{\partial x_i} \left( k \frac{\partial T}{\partial x_i} \right)
\]  \hspace{1cm} (2.3)

where \( E \) is the total energy.
2.3 Modeling Approach

Initially the fluid flow and heat transfer analysis was performed using the two-dimensional model. A three-dimensional approach was then used in order to understand the complex flow physics and to get some idea about the three-dimensional approach that needs to be adopted. This would also help in validating the use of the code for these types of applications. The modeling and results will be discussed in detail in future chapters 5 and 6 of the thesis.
CHAPTER 3

NUMERICAL METHOD AND ALGORITHM

The flow inside an offset strip-fin heat exchanger channel is characterized by a complex flow field, which is affected by blockage and recirculation zones enhanced by sharp rectangular edges and narrow gaps. In this case the actual physical device is replaced by a discrete number of points that represent the entire geometry of the cell where the distributions of pressure, velocity, etc. are to be found. The approach requires defining the mathematical equations that govern the physical process. These equations will be solved only at the discrete points representing the geometry. FLUENT, a computer program, based on a finite-volume method is among the more powerful packages of existing commercial software for solving fluid flow and heat transfer problems [23]. The purpose of this research study is to determine the effect of each geometrical parameter on the performance of the overall heat exchanger, and to determine some useful methods on improving the current baseline heat exchanger design.

3.1 The finite volume method

The Finite Volume Method (FVM), often called control volume methods, are formulated from the inner product of the governing partial differential equations with a unit function, \( I \). This process results in the spatial integration of the governing equations.
The integrated terms are approximated by either finite differences or finite elements, discretely summed over the entire domain. Physically, the conservation of mass, momentum, and energy are assured in the formulation of FVM via the FDM itself. For the program used in this study, the approximation is done by the finite difference scheme.

3.2 Solution method

The segregated solver was used to solve the governing integral equations for the conservation of the mass, momentum, and energy equations and other scalars, such as turbulence. The solution is obtained by using a control-volume-based technique which consists of:

- Division of the domain into discrete control volumes using a computational grid
- Integration of governing equations on the individual control volumes to construct algebraic equations for the discrete dependent variables ("unknowns"), such as velocity, pressure, temperature, and conserved scalars
- Linearization of discretized equations and solution of the resultant linear equation system to yield updated values of the dependent variables

3.2.1 Segregated Solution Algorithm

In this approach the non-linear governing equations are solved sequentially using the iterative technique. Each step in iteration is as follows:

1. Fluid properties are updated, based on the current solution (or the initial conditions) if just starting;
2. The u, v, and w momentum equations are each solved in turn using current values of pressure and face mass fluxes, in order to update the velocity field

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
3. Since the velocities obtained in step 2 may not satisfy the continuity equation locally the "Poisson-type" equation for the pressure correction is derived from the continuity equation and the linearized momentum equations. This pressure correction equation is then solved to obtain the necessary corrections to the pressure and velocity fields and the face mass fluxes, such that the continuity is satisfied.

4. Where appropriate the equations for scalars such as turbulence, and energy are solved using the previously updated values of other variables;

5. A check for the convergence of the equation set is made.

The governing equations which are discrete and non-linear are linearized to produce a system of equations for the dependent variables in every computational cell. The resultant linear system is then solved to yield an updated flow-field solution. The governing equations were linearized by the "implicit" method with respect to the set of dependent variables. By this method the unknown value in each cell is computed using a relation that includes both existing and unknown values from neighboring cells. Therefore, each unknown appearing in more than one equation such as the velocity \( \nu \) and these equations will be solved for simultaneously to give the velocity and pressure values.

A point implicit Gauss-Siedel linear equation is used in conjunction with an algebraic multigrid (AMG) method to solve the resultant scalar system of equations for the dependent variables in each cell. Thus, the solver solves for a single variable field like pressure, at one time considering all cells. The next variable, like velocity, is solved using the same technique.
3.3 Discretization

FLUENT uses a control-volume-based technique to convert the governing equations to algebraic equations that can be solved numerically. This control volume technique consists of integrating the governing equations about each control volume, yielding discrete equations that conserve each quantity on a control-volume basis. The integral of the governing equations is applied to each control volume, or cell, in the computational domain and discretized. By default the solver stores the discrete values of the scalar quantities at the cell centers. However, when the face values of the scalar quantities are required the values are interpolated from the cell center values. This is accomplished using an “upwind” scheme. Upwinding means that the face values are derived from quantities in the cell upstream, relative to the direction of the normal velocity.

A first order upwind scheme was used for acquiring first-order accuracy. When this scheme is selected, the face value of a scalar quantity is set equal to the value in the upstream cell. Since the flow is aligned to the grid here i.e., flow in a hexahedral grid, the first order discretization scheme was found to be acceptable.

3.3.1 Discretization of the Momentum Equations

Since the pressure fields and face mass fluxes are not known a priori and must be obtained as apart of the solution. FLUENT uses a co-located scheme, whereby pressure and velocity are both stores at cell centers. An interpolation scheme is formulated to compute the face values of pressure from the cell values.
Pressure Interpolation Scheme

The default scheme in the standard solver interpolates the pressure values at the faces using momentum equation coefficients, since the variation between the cell centers is considerably smooth without any large gradients.

3.3.2 Discretization of the Continuity Equation

In the sequential scheme as described earlier the continuity equation is used as an equation for pressure. But for incompressible flows pressure does not appear explicitly since density is not directly related to pressure. The SIMPLE (Semi-Implicit Method For Pressure-Linked Equations) is used to introduce pressure into continuity equations.

Pressure-Velocity Coupling

The SIMPLE algorithm uses the relationship between velocity and pressure corrections to enforce mass conservation and to obtain the pressure field. The momentum equation when solved using a guessed pressure field to obtain the face flux. If the resulting face flux does not satisfy the continuity equation a correction face flux is added to obtain the corrected face flux. Thus, satisfying the continuity equation.

The SIMPLE algorithm substitutes the flux correction equations into the discrete continuity equation to obtain the discrete equation for the pressure correction $p'$ in the cell. The pressure-correction equation may be solved using the AMG method.

3.4 Modeling periodic flows in the heat exchanger channel

Periodic flow occurs when the physical geometry of interest and the expected pattern of the flow solution have a periodically repeating nature. In this type of heat exchanger geometry, a pressure drop occurs transitionally across periodic boundaries, resulting in a
"fully-developed" or "streamwise-periodic" flow. FLUENT provides the ability to calculate such a periodic flow.

3.4.1 Theory

Definition of Periodic Velocity

The assumption of periodicity implies that the velocity components repeat themselves in space as follows:

\[ u(r) = u(r + L) = u(r + 2L) = \ldots \] (3.1)

\[ v(r) = v(r + L) = v(r + 2L) = \ldots \] (3.2)

\[ w(r) = w(r + L) = w(r + 2L) = \ldots \] (3.3)

Where \( r \) is the position vector and \( L \) is the periodic length vector of the domain considered.

3.4.2 Definition of Streamwise-Periodic Pressure

For viscous flows the pressure is not periodic in the sense of equations 3.1-3.3. Instead, the pressure drop between the modules is periodic

\[ \Delta p = p(r) - p(r + L) = p(r) - p(r + 2L) = \ldots \] (3.4)
Since the segregated solver was used the local pressure gradient can be decomposed into two parts: the gradient of a periodic component, $\nabla p^r(r)$ and the gradient of a linearly-varying component, $\beta \frac{L}{r}$:

$$\nabla p(r) = \beta \frac{L}{r} + \nabla p^r(r)$$

Where $\nabla p^r(r)$ is the periodic pressure and $\beta(r)$ is the linearly-varying component of the pressure. The periodic pressure is the pressure left over after subtracting out the linearly-varying pressure. The linearly-varying component of the pressure results in a force acting on the fluid in the momentum equations. Because the value of $\beta$ is not known a priori it must be found iteratively until the mass flow rate that has been defined is achieved in the computational model. This correction of $\beta$ occurs in the pressure correction step of the SIMPLE algorithm, where the value of $\beta$ is updated based on the difference between the desired mass flow rate and the actual one. The number of sub-iterations used to update $\beta$ is decided by the user.

3.4.3 Calculation Method

In order to calculate a spatially periodic flow field with a specified mass flow rate or pressure derivative, a grid was created with translationally periodic boundaries that are parallel to each other and equal in size. If the mass flow rate specification option is
chosen and the solver calculates the appropriate value of $\beta$. The value of $\beta$ is iterated until the desired mass flow rate is obtained.

3.5 Turbulence modeling

The flows inside the type of heat exchanger channels outlined in this work are sometimes characterized by fluctuating velocity fields. These fluctuations mix transported quantities such as momentum, and energy and cause these transported quantities to fluctuate as well. Since these fluctuations are of small scale and high frequency they are too computationally expensive to simulate directly in calculations. Instead, the exact governing equations can be time-averaged, ensemble-averaged, or otherwise manipulated to remove all scales, resulting in a modified set of equations that are computationally less expensive to solve.

3.5.1 Choosing a Turbulence Model

FLUENT offers two major approaches for the turbulence modeling, the Reynolds-averaged Navier-Stokes (RANS) and the Large Eddy Simulation (LES). While the former is well developed and consumes less computational time the latter is in its infancy stage and requires very large computational times. The Reynolds-averaged Navier-Stokes (RANS) equation represents transport equations for the mean flow quantities only, with all the scales of turbulence being modeled. The approach of permitting a solution for the mean flow variables greatly reduces the computational effort. Since the mean flow is considered to be steady, the governing equations will not contain time derivatives and a steady-state solution can be obtained economically. The Reynolds-averaged approach is generally adopted for practical engineering calculations such as flow inside heat exchanger channels. The turbulence modeling was performed using the standard $k-\omega$
model, an inbuilt module in the commercial code FLUENT (2003), which is based on the
RANS approach.

The standard k-ω model is an empirical model based on model transport equations for
the turbulence kinetic energy (k) and the specific dissipation rate (ω). As the model has
undergone modification over the years production terms have been added to both the k
and ω equations, which have improved the accuracy of the model in predicting shear
flows. The transport equations of the k-ω turbulence model are as follows:

\[ \frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} \left( \frac{\Gamma_k}{\partial x_j} \right) + G_k - Y_k + S_k \quad (3.5) \]

And

\[ \frac{\partial}{\partial t} (\rho \omega) + \frac{\partial}{\partial x_i} (\rho \omega u_i) = \frac{\partial}{\partial x_j} \left( \frac{\Gamma_\omega}{\partial x_j} \right) + G_\omega - Y_\omega + S_\omega \quad (3.6) \]

In these equations \( G_k \) represents the generation of turbulence kinetic energy due to
mean velocity gradients. \( G_\omega \) represents the generation of \( \omega \). \( \Gamma_k \) and \( \Gamma_\omega \) represent the
effective diffusivity of k and \( \omega \), respectively. \( Y_k \) and \( Y_\omega \) represent the dissipation of k
and \( \omega \) due to turbulence. \( S_k \) and \( S_\omega \) are the user defined source terms, and all of the
above terms are calculated as described below.

3.5.2 Modeling the Effective Diffusivity

The effective diffusivities for the k-ω model are given by

\[ \Gamma_k = \mu + \frac{\mu_s}{\sigma_k} \]
\[ \Gamma_w = \mu + \frac{\mu_t}{\sigma_w} \]

Where \( \sigma_k \) and \( \sigma_\omega \) are the turbulent Prandtl numbers for \( k \) and \( \omega \), respectively. The turbulent viscosity, \( \mu_t \), is computed by combining \( k \) and \( \omega \) as follows:

\[ \mu_t = \alpha^* \frac{\rho k}{\omega} \]

### 3.5.3 Low-Reynolds-Number Correction

Since the flow inside the heat exchanger channel is in the transitional regime it is fit to enable the Low-Reynolds-Number correction equation. The coefficient \( \alpha^* \) damps the turbulent viscosity causing a low-Reynolds-number correction. It is given by:

\[ \alpha^* = \alpha^*_w \left( \frac{\alpha_0^* + \frac{\text{Re}_t}{R_k}}{1 + \frac{\text{Re}_t}{R_k}} \right) \]

Where

\[ \text{Re}_t = \frac{\rho k}{\mu \omega} \]

\[ R_k = 6 \]

\[ \alpha_0^* = \frac{\beta_1}{3} \]

\[ \beta_1 = 0.072 \]

Note that in the high-Reynolds-number form of the \( k-\omega \) model, \( \alpha^* = \alpha^*_w = 1 \)

27
3.5.4 Modeling the Turbulence Production

Production of $k$

The term $G_k$ represents the production of turbulence kinetic energy. From the exact equation for the transport of $k$, this term may be defined as:

$$G_k = -\rho \mu_i \mu_j \frac{\partial u_j}{\partial x_i}$$

Production of $\omega$

The production of $\omega$ is given by

$$G_\omega = \alpha \frac{\omega}{k} G_k$$

Where $G_k$ is given by the above equation. The coefficient $\alpha$ is given by

$$\alpha = \frac{\alpha_0}{\alpha^\prime} \left( \frac{\alpha_0 + \text{Re}_e/R_\omega}{1 + \text{Re}_e/R_\omega} \right)$$

Where $R_\omega = 2.95$. $\alpha^\prime$ and $\text{Re}_e$ are given by the above equations.

3.5.5 Modeling the Turbulence Dissipation

Dissipation of $k$

The dissipation of $k$ is given by

$$Y_k = \rho \beta^\prime f_\beta k \omega$$

Where

$$f_\beta = 1$$

Where

$$x_k = \frac{1}{\omega^3} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}$$
And

\[ \beta^* = \beta^*_1 \left[ 1 + \zeta^* F(M_e) \right] \]

\[ \beta^*_1 = \beta^*_\infty \left( \frac{4/15}{1 + \left( \frac{Re_p}{R_p} \right)^4} \right) \]

\[ \zeta^* = 1.5 \]

\[ R_p = 8 \]

\[ \beta^*_\infty = 0.09 \]

**Dissipation of \( \omega \)**

The dissipation of \( \omega \) is given by:

\[ Y_\omega = \rho f_\beta \omega^2 \]

Where

\[ f_\beta = 1 \]

\[ \chi_\omega = \frac{\Omega_{ij} \Omega_{kl} S_{li}}{\left( \beta^*_\omega \omega \right)^3} \]

\[ \Omega_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i} \right) \]

The strain rate tensor, \( S_{ij} \) is defined in the equations above
3.5.6 Model Constants

\[ \alpha^* = 1, \alpha_\omega = 0.52, \alpha_0 = 0.111, \beta^* = 0.09, \beta_i = 0.072, R_\beta = 8 \]

\[ R_s = 6, R_\omega = 2.95, \zeta^* = 1.5, \sigma_k = 2.0, \sigma_\omega = 2.0 \]

3.5.7 Near Wall Treatments for Wall-Bounded Turbulent Flows

Turbulent flows are significantly affected by the presence of walls. Obviously, the mean velocity field is affected through the no-slip condition that has to be satisfied at the wall. Very close to the walls, viscous damping reduces the tangential velocity fluctuations, while kinematic blocking reduces normal fluctuations. However, towards the outer part of the near-wall region, the turbulence is rapidly augmented by the production of turbulence kinetic energy, due to the large gradients in the mean velocity.

Traditionally, there are two approaches to modeling the near-wall region. In one approach the viscosity-affected inner region (viscous sub-layer and buffer layer) is not resolved. Instead, semi-empirical formulae called “wall-functions” are used to bridge the viscosity-affected region between the wall and the fully-turbulent region. The use of wall functions obviates the need to modify the turbulence model to account for the presence of the wall.

In another approach the turbulence models are modified to enable the viscosity-affected region to be resolved with a mesh all the way to the wall, including the viscous sub-layer and are called the “near-wall modeling” approach. Since flows in these channels are in low-Reynolds-number regimes the wall function approach was found to be inadequate. Thus, the near-wall modeling approach was considered.
3.6 Grid Considerations for Turbulent Flow Simulations

Successful computations of turbulent flows depend greatly on mesh generation, due to the fact that strong interaction of the mean flow and turbulence, the numerical results for the turbulent flows tend to be more susceptible to grid dependency than laminar flows. Since the transitional flows option was enabled in the Viscous Model panel to suit the flow inside the heat exchanger channel, low-Reynolds-number variants were used, and in such a case the mesh guidelines followed were similar to those used for the enhanced wall treatment.

3.7 Defining Turbulence Boundary Conditions

The boundary conditions which need to be input into the code are the ones at the inlet boundaries (velocity inlet). It is critical to specify correct or realistic boundary conditions at the inlets, because the inlet turbulence can significantly affect the downstream flow.

3.8 Solution Strategy for Turbulent Flow Simulations

Compared to the laminar flow simulations the turbulent flow was more challenging in many ways. Since the equations for mean quantities and turbulent quantities are strongly coupled in a highly non-linear fashion it took a lot more computational effort to obtain an accurate turbulent solution than it did to obtain an accurate laminar flow solution. Some of the extra care taken is as follows:

- Additional care was taken to create an optimal mesh, since the flow was wall bounded, and the wall was expected to significantly affect the flow.
- Hexahedral mesh was used, since the mean quantities were expected to have larger gradients than in laminar flows.
- Realistic inlet boundary conditions were set

3.9 Boundary Conditions

3.9.1 Velocity Inlet Boundary Conditions

Velocity inlet boundary conditions are used to define the flow velocity, along with all other relevant scalar properties of the flow, at the flow inlets. The total (or stagnation) properties of the flow are not fixed, so they will rise to whatever value necessary to provide the required velocity distribution. This type of boundary condition at inlet is intended to be used in incompressible flow. It requires the specification of velocity magnitude and direction, the velocity components, or the velocity magnitude normal to the boundary. In this case the velocity normal to boundary specification method was used. There are several ways in which the code allows the definition of the turbulence parameters for turbulent calculations. The method of specifying the turbulent intensity and hydraulic diameter was used for turbulence modeling purposes. Since the flow was found to be in the laminar regions for most cases an intensity of 1% was used.

3.9.2 Pressure Outlet Boundary Conditions

The pressure outlet boundary condition requires the specification of gauge pressure at the outlet. The value of the static pressure is used only when the flow is sub-sonic, as in this case. All other flow quantities are extrapolated from the interior. A set of the “backflow” conditions are also specified, should reverse flow occur at the exit during the solution process. Specifying realistic values of backflow quantities reduced convergence.
difficulties. To set the static pressure the appropriate gauge pressure should be entered. Backflow temperature and turbulence parameters were set normal to the boundary with a realistic value. At the pressure outlets FLUENT uses the boundary condition pressure input as the static pressure of the fluid at the outer plane, and extrapolates all other conditions from the interior of the domain.

3.9.3 Symmetric Boundary Conditions

Symmetric boundary conditions can be used when the physical geometry of interest, such as the outlined heat exchanger channel, and the expected pattern of the flow/thermal solution are symmetric. When using this type of boundary condition in such regions no additional boundary conditions are required. FLUENT assumes zero flux of all quantities across a symmetric boundary. There is no convective flux across a symmetry plane, and the normal velocity component across the symmetry plane is zero. All the normal gradients are set to zero for a symmetry plane.

3.9.4 Periodic Boundary Conditions

Since the heat exchanger channel is characterized by a geometry periodically repeating in the flow direction, and the expected flow pattern would have a periodically repeating nature, the usage of such type of boundary conditions is valid. For a periodic boundary without any pressure drop, there is only one input needed i.e., the geometry is translationally periodic but when there is a pressure drop the input of mass flow rate across the periodic module is required. FLUENT treats the flow at a periodic boundary as though the opposing periodic plane is a direct neighbor to the cells adjacent to the first periodic boundary. Thus, when calculating flow through the periodic boundary adjacent
to a fluid cell, the flow conditions at the fluid cell adjacent to the opposite periodic plane are used.

3.9.5 Thermal Boundary Conditions

When choosing to solve an energy equation, it is required to define the thermal boundary condition at the walls. Since the wall zone here is a “two-sided wall” (a wall that forms the interface between two regions, such as the fluid/solid interface) a conjugate heat transfer problem is encountered. The code allows us an option to choose whether or not the two sides of the wall are “coupled”. When the “coupled” option is chosen no other additional thermal boundary conditions are required, because the solver will calculate heat transfer directly from the solution in the adjacent cells. But when performing two-dimensional numerical simulations the “Temperature Boundary Conditions” was chosen, which requires the specification of the wall surface temperature.

The heat transfer to the wall is computed as

\[ q = h_f (T_w - T_f) \] (3.7)

The fluid-side heat transfer coefficient is computed based on the local flow-field conditions. The heat transfer to the wall boundary is calculated as

\[ q = \frac{k_s}{\Delta n} (T_w - T_s) \] (3.8)

3.9.6 Fluid conditions

A fluid zone is a group of cells for which all active equations are solved. The only required input for a fluid zone is the type of fluid material. In our case there are two fluid regions. The material properties of the respective fluids are first defined in the material
properties panel of FLUENT. Since during the turbulent flow simulation the second fluid region was always laminar, an additional input called "Laminar-Zone" option was activated to disable the turbulence modeling in that region.

3.9.7 Solid Conditions

A "solid" zone is a group of cells for which the heat conduction problem is solved; no flow equations are solved. The only required input for the solid zone is the material of solid which is input into the material conditions panel of the FLUENT code.

3.10 Physical properties model

An important step in the set up of the numerical model is the definition of the physical properties. For the solid materials since the segregated solver is used; only the thermal conductivity value is required for calculations. While for the fluid materials the values of density, thermal conductivity, viscosity, and specific heat capacity are required for the calculation purposes. The physical properties may be dependent or independent of temperature depending upon the type of approach chosen.

When there is a large temperature difference between the fluid and the surface the assumption of constant fluid transport properties may cause some errors, because the transport properties of most fluids vary with temperature. These property variations will then cause a variation of velocity and temperature throughout the boundary layer or over the flow cross section of the duct. For most liquids, such as the liquid-salt Flinak, the specific heat, thermal conductivity, and density are nearly independent of temperature, but the viscosity decreases markedly with increasing temperature. It is also important to note that the Prandtl number of liquids also varies with temperature, similar to that of
viscosity. In the case of gases, like helium, the density, thermal conductivity, and viscosity all vary at the same rate with respect to temperature. The specific heat varies only slightly with temperature and the Prandtl number does not vary significantly, which was shown by Kakac et al. [24].

Hence, in order to study the influence of temperature dependent physical properties on the numerical simulations simple polynomial equations were formed. With those equations having the physical properties defined only as a function of temperature. The polynomial functions derived are in the following form

\[ \phi(T) = A_1 + A_2 T + A_3 T^2 + \ldots \]  

(3.9)

To define a physical property as a polynomial function in FLUENT the number of coefficients (A) are specified and the values are entered in the control panel.

3.10.1 Helium gas properties

When the constant physical property model was assumed the material properties such as the density, viscosity, specific heat, and thermal conductivity were taken at a temperature of 1050 K. Thus, when solving the energy equation the properties were assumed not to be changing as the temperature changed in the flow channel. The properties of helium gas at 1050 K are summarized as follows:

- Density \( (\rho) = 3.082 \text{ Kg/m}^3 \)
- Dynamic Viscosity \( (\mu) = 4.91 \times 10^{-5} \text{ Kg/m* s} \)
- Specific Heat \( (C_p) = 5193 \text{ J/Kg*K} \)
- Thermal Conductivity \( (k) = 0.38679 \text{ W/m*K} \)
When the temperature change along the channel was considered to be influencing the material properties algebraic equations were derived by the method of curve fitting from the data obtained within a certain temperature range. But since in most cases the specific heat capacity does not change with respect to temperature the value was kept constant, as in the above model. The equations that were used and the graphs used to form the algebraic equations for a temperature range of 800-1300 K are shown in Figures 3.1-3.5.

**Thermal Conductivity Using Kelvin**

\[ k(T) = 0.0003T^4 - 0.0624 \text{ (W/m*K)} \]  

(3.10)

![Graph showing variation of thermal conductivity of helium with temperature](image)

Figure 3.1 Variation of thermal conductivity of helium with temperature

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
Density Using Kelvin

\[ \rho (T) = 3.0161e^{-6}T^2 - 0.00948T + 9.809 \text{ (Kg/m}^3) \] (3.11)

Figure 3.2 Variation of density of helium with temperature

Dynamic Viscosity Using Kelvin

\[ \mu (T) = -5.4685e^{-12}T^2 + 4.0755e^{-8}T + 9.7485e^{-6} \text{ (Kg/m*s)} \] (3.12)
3.10.2 FLINAK properties

When the constant physical property model was used, similar to the helium side the material properties of FLINAK (46.5% LiF – 11.5% NaF – 42% KF) was assumed for a constant temperature of 1050 K and summarized as follows:

Density ($\rho$) = 1939.85 Kg/m$^3$

Dynamic Viscosity ($\mu$) = 0.0021214 Kg/m*s

Specific Heat ($C_p$) = 1882.8 J/Kg*K

Thermal Conductivity ($k$) = 4.50 W/m*K

In a similar way to that of the helium side the temperature dependent properties for the density and dynamic viscosity were derived in the form of algebraic equations obtained from curve fitting techniques for a temperature range of 748-1148 K were done. But the specific heat capacity and the thermal conductivity were assumed to always stay constant.
Density Using Kelvin

\[ \rho (T) = 2783 - 0.803T \text{ (Kg/m}^3) \] (3.13)

The Density of Flinak

\[ \mu (T) = 5.0351e-13T^4 - 2.1034e-9T^3 + 3.3161e-6T^2 - 0.0023468T + 0.63319 \text{ (Kg/m*s)} \] (3.14)

Figure 3.4 Variation of Density of FLINAK with temperature

Dynamic Viscosity Using Kelvin
3.11 Setting the under-relaxation factors for the solution

The segregated solver uses under-relaxation to control the update of the computed variables after each iteration. The solver initially has a set of default under-relaxation factors for all variables that are set to near optimal values for most cases. The calculations were performed with the default under-relaxation factor for all parameters except the temperature. Since the residuals of energy started increasing the under relaxation factor for energy was reduced from 1.0 to 0.8. Once the residuals started to stabilize it was then increased to 0.9 which resulted in faster convergence.
CHAPTER 4

VALIDATION

4.1 Mesh dependency investigation

The main objectives of the mesh independence investigation:

1) Verification of the mesh correctness for the problem (3-D simulations of HTHX);
2) Examination of the solution sensitivity from the mesh changes (refinement and coarsening); and,
3) Selection of the optimal mesh for the problem calculations.

4.1.1 Calculation geometry

One section of the geometry with an inlet and outlet at the 1st fin was chosen for the investigations shown in Figure 4.1.

Figure 4.1. Geometry of the calculation domain (solid part)
4.1.2 Input parameters for the FLUENT simulation

All of the input parameters were taken from the calculations of the whole heat exchanger geometry (37 sections). Inlet and outlet parameters were used from 18th section of the whole geometry. The parameters were:

- inlet parameters
  
  He side - $U=16.978 \text{ m/s}$  
  $T=1145.67 \text{ K}$  
  LS side - $U=0.1692 \text{ m/s}$  
  $T=1090 \text{ K}$

- outlet parameters:
  
  He side - $P=7.06 \text{ MPa}$  
  LS side - $P=0.1 \text{ Mpa}$

4.1.3 Mesh and calculations

Two different kinds of meshes were chosen:

1. Mesh with thickening in the wall zones (which was used for the calculations).
2. Uniform mesh.

4.1.4 Mesh with thickening in the wall zones

Three mesh types were chosen for this investigation: (coarse mesh, normal mesh and fine mesh). The coarse mesh contained 3540 nodes and 2508 hexahedral cells. The normal mesh consisted of 22419 nodes and 18392 hexahedral cells and the fine mesh consisted of 73182 nodes and 69954 hexahedral cells. For the uniform mesh the four mesh types were chosen for the investigations were: coarser mesh (12849 nodes,
10080 hexahedral cells), coarse mesh (39661 nodes, 33480 hexahedral cells), normal mesh (89665 nodes, 78720 hexahedral cells) and fine mesh (288049 nodes, 263520 hexahedral cells). The results obtained from the grid dependency analysis of the different kinds of meshes are discussed below.

4.1.5 Results for the mesh with thickening in the wall zones

Figures 4.2 and 4.3, show the comparisons of static pressure in the center of the LS and He channels. It can be clearly seen that the mesh thickening on the wall zones does not have any sort of affect on the liquid salt side and about 5% of an effect on the helium side.

![Image](image_url)

Figure 4.2. Static pressure distribution (LS part), Pa
4.1.6 Results for the uniform mesh

The dependency studies of the uniform mesh showed that the normal mesh had more consistent results in the static temperature results than compared to the fine mesh. But the static pressure behavior was found to be independent of the mesh type. The inconsistency in the finer meshes may be because the volumes along the walls are too small. The Figures 4.4 and 4.5 show the static pressure behavior along the center of the helium and liquid salt channels of the heat exchanger.
Figure 4.4. Static pressure distribution (LS part), Pa

Figure 4.5. Static pressure distribution (He part), Pa
The conclusions from the investigations are:

a) For the fine meshes the temperature behavior is not correct close to the walls. The reason is very large mesh distortions in the wall zones. The mesh volumes can be even negative; therefore, it is very important to check the mesh before calculations and exclude the abnormal (too tiny) volumes from the mesh structure.

b) Static pressure behavior is quite appropriate for each kind of mesh (difference not more than 4%).

c) Static temperature behavior is better with a uniform mesh than compared with mesh that has thickened wall zones.

d) Static temperature behavior is quite appropriate for LS part but not appropriate for He part (maximum difference is 20%). For the uniform mesh the solution for the He part is more mesh independent when compared with a mesh that has thickening in the wall zones.
4.2 Validation of conjugate heat transfer model in FLUENT

In order to use the Conjugate Heat Transfer (CHT) option in FLUENT, the capability of FLUENT to accurately simulate CHT must be validated. One of the ways of validating heat transfer results is to compare results with analytical solutions. For laminar flow the Nusselt numbers for flows through channels of basic shapes such as a circle, rectangle or infinite width channels are known quantities.

4.2.1 Test 1

A counter-flow heat exchanger having two parallel channels of a constant height and an infinite width separated from each other by a solid material was selected for Test 1. The fluids moving opposite one another in these channels are He and LS. The heights of the channels are identical to those of the real offset strip fin HTHX (He: 2 mm, LS: 1 mm). The thickness of the separating solid material is also identical to the real offset strip fin HTHX design, i.e., 1 mm. For the laminar fully-developed flow the Nusselt number is a known quantity and can be taken from Kays and London [8] for both the constant wall temperature (Nu=7.54) and constant wall heat flux (Nu=8.235) boundary conditions. Thus, the convective heat transfer coefficients can be calculated (He: $h_1=1593$ W/m²K, LS: $h_2=8400$ W/m²K).

It is possible to represent heat transfer from a hot fluid (He) to a cold fluid (LS) using an electric circuit analogy. The resistances to heat transfer can be placed into a network that would include in our case two convective and one conductive thermal resistances (Incropera and De Witt [25]). The convective thermal resistances become (He: $1/h_1=6.28 \cdot 10^{-4}$ m²K/W, LS: $1/h_2=1.19 \cdot 10^{-4}$ m²K/W). The conductive thermal resistance: $L/k_{LS}=1 \cdot 10^{-4}$ m²K/W. The average temperature on the He side is 1089.15 K, on the LS
side – 1040.65 K. On the LS side if we take these temperatures as $T_{\infty,1}$ and $T_{\infty,2}$, then the wall surface temperatures can be calculated (He: $T_{\infty,1}=1053.19$ K, LS: $T_{\infty,2}=1047.46$ K). In our numerical solution we would like to get identical wall surface temperatures.

Two periodic flows must be considered in order to get fully-developed conditions in the channels. FLUENT allows imposing only one pair of periodic boundary conditions; thus, this method of reaching a fully-developed state in FLUENT is not applicable. Another way is to take a very long channel and impose uniform temperature and velocity at the inlet so that fully-developed conditions will be reached after a certain entrance length. This is the approach that was selected. The length of the channel could be potentially taken as the length of the actual HTHX, i.e. 0.9 m. But at the channel heights of 1 mm and 2 mm it would require a very lengthy domain and a great number of nodal points. A fully developed condition could also be reached on a shorter domain, say, equal to 0.06 m. In the actual HTHX the temperature would decrease by 184 °C per 0.45 m length on the He side and increase by 207.5 °C per the same length on the LS side. This is equivalent to an increase of 12 °C per 0.03 m on the He side and a decrease by 14 °C per the same length on the LS side. Thus the inlet temperatures are taken as follows: He: $T_{in}=1101$ K, LS: $T_{in}=1027$ K. In the middle of the channels the numerical simulation must give the desired wall surface temperatures, $T_{s,1}$, $T_{s,2}$. The computed values had only a small difference ($T_{s,1}=1047$ K, $T_{s,2}=1042$ K vs $T_{s,1}=1053.19$ K, $T_{s,2}=1047.46$ K obtained analytically). The discrepancy is most likely due to a nonlinear fluid temperature variation along the He and LS channels. Even though the absolute values of $T_{s,1}$, $T_{s,2}$ do not match precisely the temperature drop through the wall is quite close. It makes up about 5 °C versus 5.73 °C from the analytical solution.
4.2.2 Test 2

In Test 1 the analytical solution for $T_{s,1}, T_{s,2}$ was derived based on the Nusselt number for the constant heat flux boundary condition ($\text{Nu}=8.235$). To see a difference from selecting the Nusselt number for the constant temperature boundary condition ($\text{Nu}=7.54$) the analytical solution was derived again and gave ($T_{s,1}=1052.83 \text{ K}, T_{s,2}=1047.53 \text{ K}$). The values differed only by about 1% to the values for the constant heat flux boundary conditions, and the temperature drop through the wall makes up 5.3 °C; thus, the numerical value of 5 °C is correct.

4.2.3 Test 3

In Test 1 the network of thermal resistances included only three thermal resistances, with only one resistance (convection) on each fluid side. The energy equation that FLUENT solves numerically for each fluid side has both convective and conductive terms, and the influence of conductive terms on the surface wall temperature values could also be significant. Therefore, in our analytical solution it would also be reasonable to include the influence of conduction from the fluid to the wall in order to see if this would eliminate the previous discrepancy in the wall surface temperatures or if it would make the discrepancy even larger. The conduction acts in parallel with convection; and therefore, the thermal resistances in the fluid become smaller due to additional heat transfer paths by conduction. The analytical solution for the case of constant heat flux boundary conditions ($\text{Nu}=8.235$), which included in parallel conductive thermal resistances on each fluid side, gave ($T_{s,1}=1054.21 \text{ K}, T_{s,2}=1047.29 \text{ K}$). These values had only less than 1% variation to the values obtained in Test 1 thus; the influence of conduction through the fluid on the overall heat transfer can be neglected.
4.2.4 Test 4

In this test the fluid flow on the LS side was not calculated. Instead, it was considered that the material of the wall along the He channel is exposed to the “atmosphere” of LS having bulk temperature \( T_{\infty,2} \) equal to the average temperature of the LS (\( T_{\infty,2}=1040.65 \) K) and a convective heat transfer coefficient \( h_2=8400 \text{ W/m}^2\text{K} \), calculated based on the Nusselt number for a constant heat flux boundary condition (\( Nu_2=8.235 \)). In FLUENT this is associated with the convective boundary condition. The fluid flow in the channel must reach a fully-developed state before the exit since the channel length is 450 hydraulic diameters. Likewise, it is expected that the thermal boundary layer also reaches a fully-developed state before the exit. The obtained solution showed that the velocity boundary layer reaches a fully-developed state in 0.1-0.2 meters (50-100 hydraulic diameters). This can be evidenced by a stabilized pressure gradient and an unchanging shape of the velocity profile. However, the shape of the temperature profile, as well as the temperature distribution within the solid material does not reach a constant condition. The difference in the wall temperature and axial fluid temperature decreases from 82 °C in the middle of the channel to 30.5 °C at the end of the channel. The temperature gradient through the wall also keeps decreasing throughout the channel length without reaching a constant value. The exit temperature on the external wall (\( T_{x,2}=1042.7 \) K) exposed to the LS “atmosphere” had only a 1% variation to the bulk temperature of the LS (\( T_{\infty,2}=1040.65 \) K). The exit temperature on the internal wall adjacent to the He (\( T_{x,1}=1044.5 \) K) and the exit bulk temperature of He (\( T_{\infty,1}=1063.5 \) K) can be easily obtained from FLUENT. The exit heat transfer coefficient on the He side becomes \( h_1=900 \text{ W/m}^2\text{K} \), and the exit Nusselt number (\( Nu_1=4.7 \)) is almost two times less than the
anticipated value of $\textit{Nu}_j=8.235$ for a constant wall heat flux situation or $\textit{Nu}_j=7.54$ for a constant wall temperature situation (Kays and London [8]). Such a low value of the Nusselt number can be likely explained by an indirect influence of the large wall thickness, which is only twice less than the He channel height. Similarly to the previous validation study the found values of the bulk temperatures ($T_{\infty,1}=1063.5$ K, $T_{\infty,2}=1040.65$ K) and the heat transfer coefficients ($h_1=900$ W/m$^2$-K, $h_2=8400$ W/m$^2$-K) along with the thermal conductivity of LSI ($k_{\text{LSI}}=10$ W/m-K) could be used to find the wall surface temperatures, $T_{s,1}$, $T_{s,2}$ (Incropera and De Witt [25]) analytically. The analytical values were only about 1% different from the results of FLUENT ($T_{s,1}=1044.4$ K, $T_{s,2}=1042.7$ K versus $T_{s,1}=1044.5$ K, $T_{s,2}=1042.7$ K found numerically). It can be concluded that FLUENT accurately handles the CHT conditions.
4.3 Validation of the periodic boundary conditions for future calculations

The investigations to check the validity of the use of periodic boundary conditions for upper and lower walls were completed. The calculations were performed for one module due to the constraints in computational time. The three geometries used were: original geometry (the ones used in previous numerical modeling), non-rounded geometry (Figure 4.7) and rounded geometry (Figure 4.6). The calculated results for the original geometry and non-rounded geometry should be the same and the comparisons should confirm the correctness of using periodic boundary conditions for upper and bottom walls.

Figure 4.6 Geometry of the one section (with fillets)
All of the input parameters were taken from calculations of the whole heat exchanger geometry (37 modules). Inlet and outlet parameters were used from 18th section of the whole geometry:

- inlet parameters
  
  He side - $U=16.978 \text{ m/s}$;
  $T=1145.67 \text{ K}$;
  
  LS side - $U=0.1692 \text{ m/s}$;
  $T=1090 \text{ K}$;

- outlet parameters:
  
  He side - $P=7.06 \text{ Mpa}$;
  
  LS side - $P=0.1 \text{ Mpa}$.
The results were compared with the middle axes of the LS and He channels. The plots are shown in Figures 4.8 to 4.11.

Figure 4.8 Static pressure distribution (He part), Pa

Figure 4.9. Static temperature distribution (He part), K

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
Figure 4.10. Static pressure distribution (LS part), Pa

Figure 4.11. Static temperature distribution (LS part), K

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
The plots of the static temperature and static pressure on the helium channel and liquid salt channel showed non-similarity in the results when compared to original geometry and the rounded geometries. But the temperature, velocity, and pressure distributions are the same for original and non-rounded geometries; therefore, the solutions are correct and the applied periodic boundary conditions predicted the results reasonably well. Thus, with the updated boundary condition the magnitude of the effect of the fillets on the numerical results were studied.
CHAPTER 5

TWO-DIMENSIONAL NUMERICAL CALCULATIONS

5.1 Two-dimensional numerical simulations for the helium side

The concepts of 2-D studies for fluid flow and heat transfer were only performed on the helium side of the baseline heat exchanger channel design. The same cannot be applied to the molten salt side due to a low ratio of the channel height to its width (1:1.75), which shows that the neglecting of the roof and the floor of the channels would produce only trivial results. The 2-D numerical simulations were performed only to get some idea about the complexity of the flow in these kinds of heat exchanger channels.

The two approaches followed during the two-dimensional studies are:

- Simulation of fluid flow using the periodic boundary conditions (using one module of the helium channel)
- Simulation of the fluid flow and heat transfer for the full-length of the helium channel (37 modules)

The two-dimensional numerical simulation for one periodic channel was performed for the baseline channel dimensions for the helium side provided in table 2.

5.1.1 Fluid-flow across a periodic module

A periodic module was used in order to solve for both the hydrodynamics and the heat transfer with constant material properties. Periodic boundary conditions were used at AH and DE (Figure 5.2) as the modules repeat themselves in the flow direction and symmetric boundary conditions were used at AB, CD, GH, and FE in the span wise
directions. For the heat transfer case constant wall temperature boundary conditions were used along the fin walls BC and GF. The mass flow rate for the periodic module was recalculated for the 2-D problem as 0.052 Kg/s and using this FLUENT is able to calculate the pressure gradient across one module. The Figure 5.1 shows the computational domain considered for this two-dimensional numerical model.

![Figure 5.1 Computational domain considered in the periodic flow simulations](image)

While generating the computational mesh utmost care was taken to capture the boundary layers and the gradients along the channel walls as expected for these kinds of channel flows. Figure 5.2 shows the computational mesh that was used for both approaches of the two-dimensional numerical simulations.
The summary of the results obtained after the numerical simulations for the helium channel with curved fin edges and the rectangular fin edges are given in Table 5.1:

Table 5.1 Results of the 2-D CFD calculations for the baseline designs

<table>
<thead>
<tr>
<th>VARIABLE</th>
<th>RECTANGULAR FIN EDGES</th>
<th>CURVED FIN EDGES</th>
</tr>
</thead>
<tbody>
<tr>
<td>$U_{\text{ave}}$</td>
<td>15.20</td>
<td>15.19</td>
</tr>
<tr>
<td>(Average Velocity, m/s)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$U_{\text{max}}$</td>
<td>21.4</td>
<td>21.6</td>
</tr>
<tr>
<td>(Maximum Velocity, m/s)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$\Delta P$</td>
<td>20.7</td>
<td>14.8</td>
</tr>
<tr>
<td>(Pressure Drop, kPa)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$f$</td>
<td>0.065</td>
<td>0.048</td>
</tr>
<tr>
<td>(Friction Factor)</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

From the highlighted results it is quite clear that the pressure drop for the channels with curved fin edges is about 30% smaller when compared to that of the rectangular fin edges. This noticeable increase in the pressure drop can be attributed to the presence of sharp constrictions in the rectangular fin edge channels (Figure 5.3); while the flow is blocked smoothly in the case of channels with curved fin edges (Figure 5.4).
Figure 5.3 Velocity vectors and pressure contours for rounded fin edges channel

Figure 5.4 Velocity vectors and pressure contours for rectangular fin edge channel

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
The literature review made it quite clear that even though the pressure drop across each module is periodic, the temperature drop is not periodic. Proceeding further to solving the energy equation the convective heat transfer boundary conditions were used instead of the constant wall temperature boundary conditions for solving the energy equation. The results which were obtained were different from the previous temperature profiles. Also, the heat transfer coefficients along the fin walls were plotted and the plots obtained were pretty consistent and the results were compared by varying the dimensions of the fins. The formation of vortices at the trailing edge of the fins were analyzed and compared using different cases of Reynolds numbers and the stream line plots were made for each case. Parametric studies with one module were used to investigate the influence of the Reynolds number on the length of the recirculation zone, i.e. “reattachment” length. As the Reynolds number increased the reattachment length and the magnitude of the vortices increased which had a detrimental effect in increasing the pressure drop along the flow channel.

Similar studies were not performed for the liquid salt side because of the low Reynolds number flow occurring in the flow channel. Figures 5.5 show that the reattachment length is significantly dependent on the Reynolds number, and such a big influence occurs only at lower Reynolds number regimes. Interestingly, Patankar and Prakash (1981) studied a similar case and showed that with their fin dimensions, having straight edges and lacking the longitudinal gap between fins, the influence was significant throughout the whole range of Reynolds numbers. Their results gave an increase in the reattachment length by approximately a factor of two by going from Re=100 to Re=500. The results of the present study give a similar doubling of the reattachment length by
going from 0.8 mm at Re=100 to 1.6 mm at Re=500. With increasing the Reynolds number, results of Patankar and Prakash give a continuous increase in the reattachment length. While the numerical results of the present study showed that the increase slows down at Re=500. At Re=1000 and Re=2500 the reattachment lengths are 1.8 and 2.2 mm respectively, which is an increase of only 12.5% and 22% with respect to the 1.6 mm reattachment length at Re=500.

Figure 5.5 Stream line plots showing vortices at different Re values

![Stream line plots](image)

(a) Re= 100  
(b) Re= 500

(c) Re= 1000  
(d) Re= 2500

Figures 5.6a and 5.6b show the variation of the heat transfer coefficient on the surface of the second fin (along BC curve in Figure 5.2) for the cases with a fin length equal to
10 mm, and a longitudinal gap between fins of 1 mm and 3 mm, respectively. The plots show a similar behavior of the surface heat transfer coefficient $h$, where

$$h = \frac{q}{(T_w - T_b)}$$

(23)

For both cases it is due to the flow velocities. The largest discrepancy between the profiles is only 11%. The fin length average value of the convective heat transfer coefficient makes up 3417 W/ (m$^2$ K) and 3367 W/ (m$^2$ K). The empirical correlation of Manglik and Bergles give the average value of convective heat transfer coefficient as 2894 W/ (m$^2$ K). The correlations do not account for the longitudinal gap between rows of fins. The observed difference in the enhancement of heat transfer is seemingly due to the gap.

![Figure 5.6a. Surface heat transfer coefficient along fin wall (Gap = 1 mm)](image)

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
The flow was considered laminar for all of the calculations. The highest Reynolds number used in these computations was slightly larger than 2000 which is in the transition region. At these values of Re the real flow is expected to be mostly laminar; although, it is possible that transition to turbulence may occur somewhat at $Re = 4000$. Also, the real flow may display instabilities and vortex-shedding from the trailing edges of the plates.

In addition, a parametric study was conducted using the similar boundary conditions for the effect of variation in the gap on the pumping power, the length of the fins were kept constant at 10 mm. Figure 5.7 shows the plot of pumping power with respect to variation in gap. It can be seen that the decrease in gap between the fins in flow direction did reduce the length of the recirculation zones but with an increased pumping
power requirement. In order to have negligible or less recirculation regions the fins need to be off-set in such a way that there is no gap, but it is accompanied with an increase in the pressure drop value; thus, affecting the overall heat exchanger performance.

Figure 5.7. Variation of pumping power with gap for periodic fluid flow with a constant fin length

The numerical simulations were performed with both rectangular as well as curved fin edges, and the pitch (length of fin + gap length) in the flow direction was kept constant at 12 mm. The fin length and gap were varied accordingly in different combinations. For the calculation of pumping power, based on the pressure drop the efficiency of the heat exchanger was assumed to be 0.85. The plot in figure 5.8 shows the variation in pumping power with gap for a constant pitch in the flow direction.
5.1.2 Fluid flow and heat transfer across full channel length of helium side

Numerical simulation was performed for the 37 modules of the helium side heat exchanger channel in order to study the hydrodynamics and heat transfer for both rectangular as well as curved fin edge cases. The inlet boundary condition had a velocity value of 16.97 m/s and at the pressure outlet boundary the gauge pressure was input as 7 MPa. Constant wall temperatures were specified for every fin wall, but the temperatures were different for each wall of the channel, such that the wall temperatures increase along the channel length. Post processing of results was done using TECPLCT. Constant material properties were used for the numerical simulation with the laminar flow model. Initially the numerical simulations were performed for the baseline helium channel dimensions for both curved and rectangular fin edges. The computational domain considered for the numerical simulations is as shown
The dotted line in Figure 5.9 shows the symmetric computational domain which was used for the numerical simulations. A journal file generating code was written using PASCAL, which creates the journal file that can be run in GAMBIT. This generates the 2-D computational domain for the 37 modules of the heat exchanger and the mesh file was created with about 40,000 nodes using the same concept as the one used in the previous approach for hydrodynamic and heat transfer simulations. Normal velocity inlet and pressure outlet boundary conditions were used at the entrance and exit, respectively. The boundary conditions at the fins were no-slip and constant wall temperature conditions, but the fin temperatures were varying along the flow direction. Numerical computations were performed for both rectangular and curved fin edges; while varying other geometrical parameters. Figure 5.10 below shows the velocity and temperature contour of the helium channel for the baseline case, while Figures 5.11 and 5.12 show the pressure and temperature drop, respectively across the helium channel.
Figure 5.10. Velocity and Temperature contours for the baseline case with curved fin channels
Table 5.2 shows the comparison between the two baseline designs of heat exchanger channels under study. It can be clearly observed that the overall performance of the

Figure 5.11. Plot for pressure drop across 37 module heat exchanger channel

Figure 5.12. Plot for Temperature drop across 37 module heat exchanger channel
curved fin edge channels is better than that of the rectangular fin edged channels. Similar thermal capacity is obtained for a much smaller pressure drop value.

Table 5.2 Summary of the 2-D simulation results for the helium side performance

<table>
<thead>
<tr>
<th>Properties</th>
<th>Curved fin edges</th>
<th>Rectangular fin edges</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet Velocity (m/s)</td>
<td>16.9786</td>
<td>16.9786</td>
</tr>
<tr>
<td>Outlet Velocity (m/s)</td>
<td>16.71</td>
<td>16.71</td>
</tr>
<tr>
<td>Inlet Temperature (K)</td>
<td>1273.15</td>
<td>1273.15</td>
</tr>
<tr>
<td>Outlet Temperature (K)</td>
<td>946</td>
<td>952</td>
</tr>
<tr>
<td>Pressure Drop (kPa)</td>
<td>10.39</td>
<td>12.25</td>
</tr>
<tr>
<td>Thermal Power (MW)</td>
<td>44.4</td>
<td>43.6</td>
</tr>
<tr>
<td>Friction factor $f$</td>
<td>0.015</td>
<td>0.018</td>
</tr>
</tbody>
</table>

The study of variation of pumping power with respect to the variation of the gap in the flow direction was studied with the help of numerical simulations. The results obtained from the two different approaches, namely the full channel simulations and the periodic fluid flow calculations are shown in Figure 5.13. The numerical simulations were performed with both rectangular as well as the curved fin edges and the pitch (length of fin + gap) in the flow direction were kept constant at 12 mm. The fin length and gap were varied accordingly. The efficiency of the heat exchanger was assumed to be 0.85 for the calculation of pumping power based on the pressure drop. It was observed that the pumping power required increased as the gap was decreased, and the variation in pumping power with respect to the gap was more significant for the rectangular fins.
Figure 5.13. Variation of pumping power with gap for full channel numerical simulation with a constant pitch.

Figure 5.14. Variation of thermal power with gap for full length channel numerical simulation with a constant pitch.
From Figure 5.14 it can be observed that the variation in the gap-length did not show a considerable influence on heat transfer as was the case on the fluid flow. From Figures 5.13 and 5.14 it is evident from the 2-D point of view that the overall performance of the helium side heat exchanger channel was better when curved fin edges were used. This can be attributed to the fact that the flow area is almost kept constant thus resulting in a reduced pressure drop when compared to that of rectangular fin edges.

From the results of the two-dimensional numerical simulations it can be concluded that the use of the empirical correlations in calculations for this type of heat exchanger geometry needs to be done only if the correlations are modified further to include these effects. Hence, only a three-dimensional study would provide a better idea about the validation of the analytical correlations. The next chapter would explain in detail about the three-dimensional analysis carried out for the fluid flow and heat transfer for both fluids.
6.1 Three-dimensional numerical modeling of the baseline heat exchanger designs

A 3-D CFD model of the compact high temperature heat exchanger (HTHX) design, based on the baseline dimensions, with both curved and rectangular fin edges was performed. In ceramic heat exchangers the fins can easily be shaped. By shaping the fin edges the hydrodynamic resistance to the fluid flow through the micro channels is expected to be significantly lowered, leading to a better performance of HTHX in terms of required pumping power. Pressure losses associated with the heat exchanger are usually due to two sources: (1) friction along the wall, and (2) contractions and expansions of the flow area. The first source of pressure loss also represents a source of heat transfer augmentation from a fluid to a (rough) wall. The second source of pressure loss does not lead to heat transfer enhancement in HTHX; therefore, it could be reduced to an acceptable extent or eliminated. One of the possible shapes of the fin edges is a round shape. If properly designed, the heat exchanger with rounded fin edges would maintain the equal flow area throughout the entire micro channel; thus, minimizing pressure losses leaving only those that are caused entirely by friction.

Being theoretically sound the designs of heat exchangers with rounded edges are the most difficult to handle when it comes to the estimation of their performance. Available
in the open literature are empirical correlations for offset fin heat exchangers, which are based on designs of strip fins made of metal materials that naturally have straight edges. Without accomplishing physical experiments on heat exchanger designs with rounded edges some measures could potentially be taken to fit the existing correlations to the rounded fin shapes. However, the validity of the introduced corrections can be estimated numerically (if no physical experiments are planned in the future). The three-dimensional numerical simulations were performed initially with the constant physical properties model due to the length of computational time expected. For the helium side with the baseline hydraulic diameter and velocity the flow was around the transition region \((Re = 2430)\), and was simulated using both laminar and turbulence models. When the turbulent model was used the turbulence model chosen is the standard \(k-\omega\) model with the transitional flow option.

The inlet Reynolds numbers for the helium and liquid salt were calculated as 2282 and 75, respectively while the Reynolds numbers at the outlet were calculated as 2873 and 281 for helium and liquid salt respectively. The Reynolds numbers at the outlet was calculated by judging the maximum possible outlet temperatures for the helium and molten salts as 885 K and 1200 K, respectively. It can be said that the outlet temperatures are judged not to go below the above specified values in worst case scenarios. The above test calculations were performed to decide whether the flow regimes are going to oscillate, but it is expected that the helium side is going to be in the transitional region while the liquid salt is going to stay laminar.

The FLUENT software package that numerically solves fluid flow and heat transfer equations in the complex geometry of the considered heat exchanger design requires a
mesh file to operate. This file is generated by a companion software package Gambit. The journal file for GAMBIT was generated using an in-house code JFGEN3D. The code is capable of generating an input file for GAMBIT for any desired fin/channel dimensions and any number of periodic modules. Also the fin edges can be chosen to have either rectangular or curved geometry.

If the graphical performance of the computer does not allow the generated graphics to be visualized in GAMBIT, then the results could be visualized using Tecplot, a graphical post-processing software package. The Figures 6.1 and 6.2 shows the mesh generated for the helium and liquid salt sides for the CFD modeling using GAMBIT.

Figure 6.1 Helium channel mesh
Figure 6.2 Liquid salt channel mesh

The inlet temperature, the exit pressure, and inlet velocity, based on the mass flow rate were used as boundary conditions, as explained in the previous chapter. Symmetric boundary conditions were used in the span wise direction of the flow channels and along the channel height, due to the symmetrical nature of the heat exchanger geometry. At the symmetric planes the heat flux is assumed to be zero, and the normal velocity component at the symmetry plane is also zero; therefore, no convective flux across the symmetry plane occurs. Thus, the temperature gradients and tangential components of the velocity gradients in the normal direction are set to zero. Conjugate heat transfer, which includes conduction through the material and convection through the fluids, was used in order to solve the energy equation. No other thermal boundary conditions were required for the problem since the solver will calculate heat transfer directly from the solution in the adjacent cells.
6.1.1 Numerical simulations with laminar model

The three-dimensional numerical simulations for the baseline heat exchanger designs were first performed with the incompressible and laminar flow model. The fluid and thermal boundary conditions were used as discussed in the previous chapters. Tables 6.1 and 6.2 provide the summary of the comparison for the CFD results between the curved and rectangular fin edged channels for both helium and liquid salt sides, respectively.

Table 6.1. Numerical solution results for the helium channel for the two baseline geometries with incompressible laminar model

<table>
<thead>
<tr>
<th>Variable</th>
<th>Curved Fin edge Channel</th>
<th>Rectangular fin edge Channel</th>
</tr>
</thead>
<tbody>
<tr>
<td>Friction Factor (f)</td>
<td>0.024</td>
<td>0.027</td>
</tr>
<tr>
<td>Thermal Capacity (MW)</td>
<td>51.2</td>
<td>51.2</td>
</tr>
</tbody>
</table>

Table 6.2. Numerical solution results for the liquid salt channel for the two baseline geometries

<table>
<thead>
<tr>
<th>Variable</th>
<th>Curved Fin edge Channel</th>
<th>Rectangular fin edge Channel</th>
</tr>
</thead>
<tbody>
<tr>
<td>Friction Factor (f)</td>
<td>0.105</td>
<td>0.115</td>
</tr>
<tr>
<td>Thermal Capacity (MW)</td>
<td>51.9</td>
<td>52</td>
</tr>
</tbody>
</table>

It can be observed from the above results that the curved and the rectangular fin channels provide the same thermal capacity. But the curved fin channels provide the
thermal capacity at lower pressure drop, as seen from the friction factor values of both fluid channels. The difference predicted by the CFD calculations with the laminar model is about 12% for the helium channel and 9% for the liquid salt channel between the curved and the rectangular fin channels. Thus, the preliminary CFD investigations show that the curved fin channels provide a better overall performance, due to the channel blockages being smoother than the rectangular fin edge channels.

Since the physical properties may vary with respect to temperature, due to the large temperature difference in the channel, the incorporation of the temperature physical property model was done to the numerical model discussed above. This study was done to ensure the correct choice between the constant and temperature dependent physical property model during the three-dimensional turbulent numerical simulations. The temperature dependent model discussed in the Chapter 3 was used to perform the numerical simulations and Tables 6.3 and 6.4 summarize the results obtained.

Table 6.3. Numerical solution results for the helium channel for the two baseline geometries with temperature dependent physical properties

<table>
<thead>
<tr>
<th>Variable</th>
<th>Curved Fin edge Channel</th>
<th>Rectangular fin edge Channel</th>
</tr>
</thead>
<tbody>
<tr>
<td>Friction Factor (f)</td>
<td>0.024</td>
<td>0.027</td>
</tr>
<tr>
<td>Thermal Capacity (MW)</td>
<td>51.5</td>
<td>51.5</td>
</tr>
</tbody>
</table>
Table 6.4. Numerical solution results for the liquid salt channel for the two baseline geometries with temperature dependent physical properties

<table>
<thead>
<tr>
<th>Variable</th>
<th>Curved Fin edge Channel</th>
<th>Rectangular Fin edge Channel</th>
</tr>
</thead>
<tbody>
<tr>
<td>Friction Factor (f)</td>
<td>0.115</td>
<td>0.125</td>
</tr>
<tr>
<td>Thermal Capacity (MW)</td>
<td>50.3</td>
<td>50.3</td>
</tr>
</tbody>
</table>

The results of the numerical simulations with variable physical properties showed the comparison between the curved and rectangular fin edge channels for both the fluids similar to the previous numerical model. The thermal powers of both fluids were similar for both geometries, and the frictions factor for the helium and liquid salts varied by 12% and 8% between the curved and the rectangular fin edges, respectively. The percent variations in the numbers were similar to the ones that were calculated for the constant physical property model. The numerical solution results that were obtained with the two physical property models are compared in the Tables 6.5 and 6.6.
Table 6.5. Heat Exchanger Overall Performance from numerical calculations for rectangular fin edge case

<table>
<thead>
<tr>
<th>Property</th>
<th>Constant material properties</th>
<th>Variable material properties</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Helium side pressure drop (kPa)</td>
<td>18.7</td>
<td>18.6</td>
<td>0.5%</td>
</tr>
<tr>
<td>Liquid Salt side pressure drop (kPa)</td>
<td>8.5</td>
<td>9.3</td>
<td>9%</td>
</tr>
<tr>
<td>Overall Thermal Power (MW)</td>
<td>50.8</td>
<td>50.9</td>
<td>0.2%</td>
</tr>
</tbody>
</table>

Table 6.6. Heat Exchanger Overall Performance from numerical Calculations for Curved Fin Edge Case

<table>
<thead>
<tr>
<th>Property</th>
<th>Constant material properties</th>
<th>Variable material properties</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Helium side pressure drop (kPa)</td>
<td>16.7</td>
<td>16.7</td>
<td>0%</td>
</tr>
<tr>
<td>Molten Salt side pressure drop (kPa)</td>
<td>7.8</td>
<td>8.6</td>
<td>10%</td>
</tr>
<tr>
<td>Overall Thermal Power (MW)</td>
<td>50.8</td>
<td>50.9</td>
<td>0.2%</td>
</tr>
</tbody>
</table>
The results of the numerical simulations with temperature dependent physical properties were compared (Tables 6.5 and 6.6) to similar numerical simulations in the laminar flow with constant material properties. It was found that the influence of temperature dependent physical properties did not affect the flow and heat transfer for the helium side for both rectangular and curved fin edge heat exchanger channels. But the temperature dependent physical properties did affect the pressure drop for the molten salt side by about 9% for both rectangular and curved fin edge heat exchanger channels. Since the influence of temperature dependent physical properties is less than 10%, it can be neglected. Due to the limitations in the computer resources, it was decided to use constant material properties during the numerical simulations with turbulence model. The temperature dependent physical properties had some effect on the molten-salt channel flow, but its affect on the overall performance was found to be insignificant. The effect of temperature dependent physical properties can be neglected taking into consideration the amount of computational time it would take for a numerical model comprised of temperature dependent physical properties and turbulence to converge.

6.1.2 Numerical simulations with turbulence modeling

After some literature review and some initial study about the capabilities of the different turbulence models for the numerical simulations for both cases of heat exchanger geometry it was decided to use the K-omega turbulence model. It has the capability to capture eddies, vortices, and other flow physics in the transition region, which are expected in the heat exchanger flow channels. The helium side was simulated as turbulent flow, while the liquid salt side was specified as a laminar zone. For the present case since the helium channel flow is expected to be in the lower transition
region; thus, the turbulence intensity was specified as 1%. The other boundary conditions, heat transfer model were the same as the ones that were used for the previous numerical simulations of the baseline heat exchanger designs. Theoretically, only the results which are based on turbulence modeling are the ones which are near to the actual scenario, while the laminar ones are published only for preliminary studies and comparison purposes.

The Tables 6.7 and 6.8 provide a summary of the comparison for the CFD results between the curved and rectangular fin edged channels for helium and liquid salt channels, respectively.

Table 6.7. Numerical solution results for the helium channel for the two baseline geometries

<table>
<thead>
<tr>
<th>Variable</th>
<th>Curved Fin edge Channel</th>
<th>Rectangular fin edge Channel</th>
</tr>
</thead>
<tbody>
<tr>
<td>Friction Factor (f)</td>
<td>0.024</td>
<td>0.033</td>
</tr>
<tr>
<td>Thermal Capacity (MW)</td>
<td>51.2</td>
<td>51.8</td>
</tr>
</tbody>
</table>

Table 6.8 Numerical solution results for the liquid salt channel for the two baseline geometries

<table>
<thead>
<tr>
<th>Variable</th>
<th>Curved Fin edge Channel</th>
<th>Rectangular fin edge Channel</th>
</tr>
</thead>
<tbody>
<tr>
<td>Friction Factor (f)</td>
<td>0.105</td>
<td>0.115</td>
</tr>
<tr>
<td>Thermal Capacity (MW)</td>
<td>50.5</td>
<td>50.9</td>
</tr>
</tbody>
</table>
Similar to the results of the laminar models the model here also predicted that the shape of the fins to have more effect on the fluid flow than the heat transfer. The only change in the percentage variations in the pressure drop results of the two fluids was for the 28% difference between the helium side pressure drops for the two channel configurations, while the laminar model projected a difference of about 12%, despite the friction factor values of the curved fin edges remained the same.

6.1.3 Comparison between the laminar and turbulence modeling results

The results obtained from the two different numerical models are compared to see the effects of the turbulence on the baseline design calculations. The comparisons are summarized in Tables 6.9 and 6.10.

Table 6.9. Heat Exchanger Overall Performance comparison between the laminar and turbulence numerical models Calculations for Curved Fin Edge Case

<table>
<thead>
<tr>
<th>Property</th>
<th>Laminar Model</th>
<th>Turbulence Model</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Helium side pressure drop (kPa)</td>
<td>16.7</td>
<td>16.9</td>
<td>1.2%</td>
</tr>
<tr>
<td>Liquid Salt side pressure drop (kPa)</td>
<td>7.8</td>
<td>7.8</td>
<td>N/A</td>
</tr>
<tr>
<td>Thermal Power (MW)</td>
<td>50.8</td>
<td>50.9</td>
<td>0.2%</td>
</tr>
</tbody>
</table>
Table 6.10. Heat Exchanger Overall Performance comparison between the laminar and turbulent numerical models Calculations for Rectangular Fin Edge Case

<table>
<thead>
<tr>
<th>Property</th>
<th>Laminar Model</th>
<th>Turbulence Model</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Helium side pressure drop (kPa)</td>
<td>18.7</td>
<td>23</td>
<td>23%</td>
</tr>
<tr>
<td>Liquid Salt side pressure drop (kPa)</td>
<td>8.5</td>
<td>8.5</td>
<td>N/A</td>
</tr>
<tr>
<td>Thermal Power (MW)</td>
<td>50.8</td>
<td>51.4</td>
<td>1.2%</td>
</tr>
</tbody>
</table>

It can be observed that the effects of turbulence were more pronounced on the helium channel with rectangular fin edges. There was a 23% difference in the pressure drop results for the rectangular fin channels, while there was only a 1% difference in the pressure drop results for the helium channel with curved fin edges. There was not much of an effect on the overall thermal performance of the either type of heat exchanger channels; and there was a 1% difference in the thermal performance between the straight and curved fin edge cases. This again reiterated that the curved fin edge channels are the better of the two, considering the overall performance and pumping power requirements. In general it can be concluded that the effect of turbulence was more felt on the helium channel with rectangular fin edges, which can be attributed to the fact that the flow constrictions in the rectangular fin edge channels induce more disturbances and increase the complexity of the flow. For the curved fin edge the flow despite being complex, the
flow around the fins in the trailing edges and the leading edges keep it very smooth, due to the smoother profile of the fin edges in the channel. So in all future numerical simulations it was decided to use the turbulent model for the helium channel when the Reynolds number of the flow is greater than 2000.

6.2 Design including manufacturing geometrical effects (MGE)

The main purpose of this investigation is to update the design to include manufacturing geometrical effects on the heat exchanger baseline design with curved fin edge channels. As its clear that by the method of fabrication of these type of heat exchanger channels the ceramic materials would result in fillets present in the flow area as shown in Figure 6.3. The studies were conducted only for both laminar and turbulent flow cases for the helium side with constant material properties.

The other physical properties of material remained the same, except periodic boundary conditions were used instead of symmetry boundary conditions on the top and bottom surfaces, since the geometry is no longer symmetric. The investigations in Chapter 4 proved the validity of the use of periodic boundary conditions for upper and lower walls.
As a preliminary investigation the numerical simulations were performed for the helium flow channel with the laminar model. Tables 6.11 and 6.12 summarize the results of the numerical simulations, for the helium and liquid salt channels, respectively using the laminar model.

Table 6.11. Comparison of numerical solution results for the He side for laminar model

<table>
<thead>
<tr>
<th>Property</th>
<th>Without fillets</th>
<th>With fillets</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure drop (kPa)</td>
<td>16.74</td>
<td>17.32</td>
<td>+3.5 %</td>
</tr>
<tr>
<td>Thermal power (MW)</td>
<td>51.2</td>
<td>51.8</td>
<td>+1.2 %</td>
</tr>
</tbody>
</table>
Table 6.12. Comparison of the numerical solution results for the LS side

<table>
<thead>
<tr>
<th>Property</th>
<th>Without fillets</th>
<th>With fillets</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure drop (kPa)</td>
<td>7.84</td>
<td>6.47</td>
<td>-17.47 %</td>
</tr>
<tr>
<td>Thermal power (MW)</td>
<td>50.4</td>
<td>50.8</td>
<td>+0.94 %</td>
</tr>
</tbody>
</table>

The influence of the fillets is very strong for pressure drop on liquid salt side, showing about a 17% effect. The fillets are almost negligible for other parameters, such as: the helium side pressure drops and the thermal capacity of the channels. The reason for the pressure drop difference on the LS side is that for geometry with fillets the surface is smoother, as compared with geometry without fillets.

The investigation of the effect of the fillets was performed further with the turbulence model for the helium channel flow. The summary of the investigations are shown in Tables 6.13 and 6.14.

Table 6.13. Numerical solution results comparison for He side (turbulent case)

<table>
<thead>
<tr>
<th>Property</th>
<th>Without fillets</th>
<th>With fillets</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure drop (kPa)</td>
<td>16.87</td>
<td>21.21</td>
<td>+25.73 %</td>
</tr>
<tr>
<td>Thermal power (MW)</td>
<td>50.2</td>
<td>51.5</td>
<td>+2.6 %</td>
</tr>
</tbody>
</table>
Table 6.14. Numerical solution results comparison for LS side (turbulent case)

<table>
<thead>
<tr>
<th>Property</th>
<th>Without fillets</th>
<th>With fillets</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure drop (kPa)</td>
<td>7.84</td>
<td>6.47</td>
<td>-17.47 %</td>
</tr>
<tr>
<td>Thermal power (MW)</td>
<td>50.3</td>
<td>51.4</td>
<td>+2.07 %</td>
</tr>
</tbody>
</table>

For the turbulent case the influence of the fillets is very strong for the pressure drop on the LS side (17%) and He side (26%). But the influence is almost negligible (less than 3%) for the thermal performance. Since the fillets have a large influence on the pressure drop and are negligible for temperature distribution in the turbulence model, it could be interesting to include the geometrical effects to the calculated model.

6.3 Analytical calculations

The analytical calculations were based upon the Manglik and Bergles (1995) correlations for both laminar, transition, and turbulent regimes. Some limitations in using their correlations were that their offset strip-fin geometry had rectangular fin edges and no pitch in the flow direction. Whereas in the presented case the fin edges have a curvature at the fin tips and there is some pitch given in the flow direction, in order to keep the flow area constant. In addition to the above stated differences the flow area for the design used to derive the empirical correlations also had some interruptions, like steps due to the method of manufacture. Whereas in our case there are no steps on the floor of the flow channel as shown in Figure 6.4. The values of the pressure drop and heat transfer coefficient for the helium side were calculated analytically. Also another constraint in using these empirical equations were that the prandtl number of the fluid needs to be in
the range of 0.6-0.9, in the present case both working fluid fall within that range. The use
of such empirical correlations needs to be validated in order to predict the accurate values
of heat transfer and pressure drop for this type of heat exchanger geometry; thus, it was
decided to perform two-dimensional CFD analysis for analyzing the effect of the
different channel geometries. The initial sensibility study based on the analytical
calculations showed that the geometrical parameters had a large effect on the fanning
friction factor $f$ but less of a effect on the colburn factor $j$. Thus, it was decided that
performing the simulations for flow dynamics alone would be enough in analyzing the
effects of the fin-channel geometry on the overall performance of the heat exchanger.

![Figure 6.4 Cross section of the channel geometries considered in the analytical
correlations](image)

\[ f_h = \left( \frac{P_{xh} - t_h}{h} \right) \]

\[ j_h = \left( \frac{P_{xh} - t_h}{h} \right) \]

(6.1)

(6.2)
Where the non-dimensional parameters such as $\alpha, \beta, \delta$ are defined as follows with $h$ in the suffix denoting the helium side.

$$\alpha_h = \frac{P_{sh} - t_h}{height_h} \quad (6.3)$$

$$\delta_h = \frac{t_h}{P_{sh}} \quad (6.4)$$

$$\gamma_h = \frac{t_h}{P_{sh} - t_h} \quad (6.5)$$

$$D_h = 2 \cdot \frac{height_h}{1 + height_h \cdot \frac{l_h + t_h}{P_{sh} \cdot P_{sh} - l_h \cdot l_h}} \quad (6.6)$$

$$Re_h = \frac{\mu_h \cdot D_h}{\nu_{He}} \quad (6.7)$$

The analytical calculations were performed based on the above empirical correlations and the results were as follows for the friction factor was $f = 0.035$. Theoretically, the empirical correlations are for a heat exchanger channel with rectangular fin geometry; hence, the results of the friction factor will be compared to those of the rectangular fin edge channels only.

6.4 Numerical simulation of the offset strip fin heat exchanger with no-gap in flow direction ($P_x = l$)

The simulations were performed for the numerical simulations for the rectangular and curved fin heat exchanger channel with no gap in the flow direction. This is a much closer approximation of the traditional type of offset strip-fin heat exchanger for which the present empirical correlations were derived. This would enable a check of the validity
of the empirical correlations, which have been derived for an offset strip fin channel design much closer to this design. The numerical modeling studies were performed for both rectangular and curved fin edge heat exchanger channels and the results were compared to the analytical calculations for the helium and the liquid salt channels.

Table 6.15. Summary of the comparison between three-dimensional numerical and analytical calculation for rectangular fin edge helium channel

<table>
<thead>
<tr>
<th>Properties</th>
<th>Numerical</th>
<th>Analytical</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure Drop (kpa)</td>
<td>45.6</td>
<td>32.3</td>
</tr>
<tr>
<td>Friction factor $f$</td>
<td>0.058</td>
<td>0.041</td>
</tr>
<tr>
<td>Thermal Power (MW)</td>
<td>52</td>
<td>50</td>
</tr>
</tbody>
</table>

Table 6.16. Summary of the comparison between three-dimensional numerical and analytical calculation for curved fin edge helium channel

<table>
<thead>
<tr>
<th>Properties</th>
<th>Numerical</th>
<th>Analytical</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure Drop (kpa)</td>
<td>26.8</td>
<td>32.3</td>
</tr>
<tr>
<td>Friction factor $f$</td>
<td>0.034</td>
<td>0.041</td>
</tr>
<tr>
<td>Thermal Power (MW)</td>
<td>51.4</td>
<td>50</td>
</tr>
</tbody>
</table>

Table 6.17. Summary of the comparison between three-dimensional numerical and analytical calculation for rectangular fin edge liquid salt channel

<table>
<thead>
<tr>
<th>Properties</th>
<th>Numerical</th>
<th>Analytical</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure Drop (kpa)</td>
<td>43.4</td>
<td>8.8</td>
</tr>
<tr>
<td>Friction factor $f$</td>
<td>0.527</td>
<td>0.107</td>
</tr>
<tr>
<td>Thermal Power (MW)</td>
<td>51.1</td>
<td>50</td>
</tr>
</tbody>
</table>
Table 6.18. Summary of the comparison between three-dimensional numerical and analytical calculation for curved fin edge liquid salt channel

<table>
<thead>
<tr>
<th>Properties</th>
<th>Numerical</th>
<th>Analytical</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure Drop (kpa)</td>
<td>12.6</td>
<td>8.8</td>
</tr>
<tr>
<td>Friction factor $f$</td>
<td>0.154</td>
<td>0.107</td>
</tr>
<tr>
<td>Thermal Power (MW)</td>
<td>50.7</td>
<td>50</td>
</tr>
</tbody>
</table>

Tables 6.15 – 6.18 summarize the results for the comparison of the CFD and analytical calculations. It was observed that the helium side ($Pr = 0.66$) and liquid salt side ($Pr = 0.89$) are in the valid range of Prandtl numbers for which the analytical correlations can be used. Hence, an attempt has been made to check the validity of the analytical correlations over this slightly close offset strip fin channel heat exchanger design. In general it could be said that there was a large discrepancy for both the fluids when the results of the analytical calculations were compared to the CFD results for both rectangular and curved fin edge cases. The reasons of the variations in the results for both cases can be attributed to several reasons. From Tables 6.15 and 6.16 it can be observed that the friction factor for the helium channel with rectangular and curved fin edge varying by 30% and 17% from the analytical calculations. While the former had its value predicted higher than the analytical results and the latter had its value lower than the analytical results. The 17% difference between the analytical calculations and the CFD results of curved fin edge helium channel can be explained due to the difference in the profile of the fin edges. But the 30% difference between the analytical calculations and the CFD results of the rectangular fin edge helium channel seems a bit too large. It can only be attributed to the fact that for the heat exchanger experiments from which the correlations have been derived had some steps on the flow area (the top and bottom of the
flow channels). Also, the fin thickness-length, thickness-pitch, and pitch-height ratios that were used for the test apparatuses in the derivation of the analytical correlations did not match this heat exchanger channel geometry, and may have attributed to some of the affect on the results. But these effects compounded with the differences in the shape of the fins attribute for the 17% difference. When numerical calculations predicted the friction factor values lower than the analytical calculations for the curved fin edges.

When the comparisons between the CFD and analytical calculations for the liquid salt were made, shown in Tables 6.17 and 6.18, it can be seen that the friction factor values predicted by the numerical calculations were unlike the helium channel, and they were consistently larger than the analytical calculations. There was an 80% and 30% difference between the numerical and analytical calculations for the liquid salt channels with rectangular and curved fin edges, respectively. But similar to the helium channel the difference between the curved fin edge channel friction factors results were much closer to the analytical calculation results than the rectangular fin edge channel values. Inspite of an evident difference in the profile of the fin edges for both helium and liquid salt sides in the curved fin edge channels predicted values of friction factor were much closer to that of the analytical calculations, and leads to the question of the validity of the analytical correlations to be used for this kind of heat exchanger geometry. Even being closer to the cores used in derivations than the baseline designs considered (with gap in flow direction).

Hence, only experiments or numerical simulations for the heat exchanger design matching the dimensions of the offset strip fin cores used in the derivation of the empirical correlations will provide the best way forward.
After the three dimensional numerical model validated and established the parametric studies were performed to make some design improvements. The optimization studies were performed to take into account the effect that every geometrical parameter involved in the heat exchanger design has on the overall performance of the heat exchanger. In other words, to arrive at the optimal geometrical parameters that will give the best trade-off between the required pumping power and the obtainable thermal power obtained. The baseline design results obtained for the curved and rectangular fin edge heat exchanger channels have already proved that the curved fin channel provides a better overall performance. However, it was still decided to perform this study for both channel geometries. The main strategy that has been followed is that if a particular parameter is changed the values of other parameters remain unaltered; therefore, the variation in the results can be attributed to the change in that parameter. The baseline design parameters are kept as the reference values, and the values of the changed parameters are more than or less than the baseline parameter values. The studies were also performed in such a way that for all designs the overall dimensions such as the length, width, and the height are almost the same. The suitable numerical model (laminar or turbulence) was chosen after a calculation of the Reynolds number for the helium side was performed.
7.1 Effect of the Overall Length on computational time

However, these studies involve a significant amount of computational time. Therefore, an investigation into reducing the computational time was made while retaining results that can be compared on an equal scale. The results of this investigation revealed that the time step for each iteration could be reduced by 50%, if Fluent is run through a parallel processor. Also, the time step per iteration could be reduced further by solving the flow and turbulence equations first and then solving the energy equation. By solving the flow and turbulence equations separately the time to reach a converged solution could be reduced by approximately three days. However, there is a numerical difference associated with the energy equation when solving the flow and turbulence equations apart from the energy equation. This means that the results from the energy equation are 1.2% larger than the benchmark case.

Studies on making the HTHX shorter, for optimization reasons, was also investigated. The two different sizes chosen were 18 modules and 9 modules long. This translates into two heat exchangers that are 1/2 and 1/4 the length of the benchmark case, respectively. Both cases converged in 80,000 iterations, or in half the number of iterations it took for a 37 module heat exchanger. Also, it took both cases only 4 days to converge, or approximately 1/4 of the time it takes to run the current 37 module heat exchanger. The results from this study are shown in the table below, but it should be noted that only the pressure drop values can be scaled.
Table 7.1. Numerical solution results for the two baseline geometries

<table>
<thead>
<tr>
<th></th>
<th>ΔP (kPa)</th>
<th>Scaled ΔP (kPa)</th>
<th>LMTD (K)</th>
<th>Thermal Power (MW)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Helium</td>
<td>Flinak</td>
<td>Helium</td>
<td>Flinak</td>
</tr>
<tr>
<td>Baseline</td>
<td>23</td>
<td>8.5</td>
<td>23</td>
<td>8.5</td>
</tr>
<tr>
<td>18 Module</td>
<td>11</td>
<td>4.2</td>
<td>22</td>
<td>8.4</td>
</tr>
<tr>
<td>9 Module</td>
<td>5.6</td>
<td>2.1</td>
<td>22</td>
<td>8.4</td>
</tr>
</tbody>
</table>

From the above results it is quite clear that the pressure drops can be scaled up or scaled down from the baseline design depending on the number of modules. The thermal power results from Figure 7.1 show that they become asymptotic after about the 18th module of the heat exchanger. In other words the last few modules of the baseline design play very a small role on the thermal performance. So for the parametric studies it was decided to maintain the overall length of the heat exchanger at 0.44 m, and the change in the geometrical parameters will be accommodated in such a way that the results can be compared without any anomaly.
7.2 Effect of fin thickness parameter (t)

The fin thickness studies involved increasing and decreasing the fin thicknesses (Figure 7.2) on the helium and liquid salt channels. On the helium side the fin thickness was increased from 0.75 mm to 1.05 mm and decreased to 0.45 mm. The fin thickness on the helium side could not be studied for any value smaller than 0.45 mm, due to manufacturing restrictions. The fin thickness on the liquid salt side was increased from 1.25mm to 1.40mm and decreased to 0.95mm. None of the changes in fin thickness had a significant effect on the thermal power in the helium and liquid salt channels shown in, Figures 7.3 and 7.4 respectively. The variations in the thermal power values were only about 3% for the helium side and almost negligible for the liquid salt sides.
Figure 7.2. Heat exchanger module showing the thickness parameter under study

Figure 7.3. Fin thickness vs Thermal power for helium channel
However, this was not the case for the pressure drop values. For the helium and liquid salt channels an increase in fin thickness caused an increased pressure drop, and a decrease in fin thickness caused the pressure drop to decrease shown in, Figures 7.5 and 7.6 respectively. It should also be noted that the helium channel was the most sensitive to change, especially in terms of pressure drop where there was a 30% effect on the pressure drop. The effect was about 25% for the liquid salt side in most cases.
Figure 7.5. Fin thickness vs Pressure drop for helium channel

Figure 7.6. Fin thickness vs Pressure drop for liquid salt channel
7.3 Effect of channel height \((h)\)

The optimization studies for the channel height parameter shown in Figure 7.7 were performed by decreasing the channel height on the helium side from 2.0 mm to 1.0 mm and 0.5 mm, and then increasing the channel height up to 4.0 mm, in steps of 1.0 mm. On the liquid salt side the channel height was decreased from 1.0 mm to 0.5 mm, and then increased up to 4.0 mm, in steps of 1.0 mm.

![Figure 7.7 Heat exchanger module showing the fin height parameter under study](image)

These parametric changes showed considerable effect on the helium side. Decreasing the channel height on the helium side increased the thermal power, and increasing the channel height on the helium side decreased the thermal power shown in Figure 7.8. When the channel heights were varied from 1.0 mm to 0.5 mm the thermal power increased by about 6% for both helium channel geometries. But when the channel heights were increased from 1.0 mm to 4.0 mm, in steps of 1.0 mm, the thermal power degraded by about 7% for each 1.0 mm increase. However, it can be seen that the thermal power on the liquid salt side was not affected by either an increase or decrease in channel height.
Changes in the channel heights did have a significant impact on pressure drop values for both channels. A decrease in channel height on the liquid salt side, to 0.5 mm, increased the pressure drop by a large margin, and an increase in channel height significantly lowered the pressure drop shown in, Figure 7.9. Also, the pressure drop behavior caused by increasing and decreasing the channels heights on the helium side was the same as the liquid salt side, but the effects were more or less showing the same trend and producing a considerable difference.
7.4 Effect of pitch in x-direction ($P_x$)

The studies for the pitch in the x-direction shown in, Figure 7.10 were performed by decreasing the helium and liquid salt channels x-direction pitch from 3.0 mm to 2.75 mm, the greatest possible decrease due to limitations in the mesh generating capabilities. Also, the pitch in the x-direction for the helium and liquid salt channels was increased up to 6.0 mm, in steps of 1.0 mm.
The thermal power for both channels increased by only about 1% with the decrease in $x$-direction pitch as in Figure 7.11; however, the thermal power for both channels started to degrade by about 3-4% with an increase in pitch in the $x$-direction.

Figure 7.11. $X$-direction pitch vs Thermal power for both channels
Results from these parametric studies shown in Figure 7.12, showed that a decrease in the x-direction pitch raised the pressure drop in the helium and liquid salt channels by about 15%, and an increase in pitch in the x-direction decreased the pressure drop by about 30% in the helium and liquid salt channels.

Figure 7.12. X-direction pitch vs Pressure drop for both the channels

7.5 Effect of gap-length ($P_y$-l)

Optimizing the channel gap length shown in Figure 7.13 was done by decreasing the gap length in the helium and liquid salt channels from 2.0 mm to 0 mm, in steps of 1.0 mm. The gap length for the helium and liquid salt channels were also increased up to 4.0 mm, in steps of 1.0 mm.
These changes had no significant effect on the thermal power for the liquid salt and helium channels as shown in Figure 7.14. The maximum variation between the two extreme values of gap-lengths (0 mm and 4.0 mm) showed only a 3% difference.
The results of the study, Figure 7.15, show that the pressure drop decreases as the gap in the flow direction is increased. But the most significant degradation in the pressure drop values for all four cases were found when the gap-length was zero. The most significant change was on the liquid salt side with rectangular fin edges where there was an 80% difference between the gap length values of 0 mm and 1.0 mm. This can be attributed to the flow channel being very small, due to the combination of the fin thickness and the absence of a gap in the flow direction. However, the other three cases still had a significant difference, of about 40%, between the 0 mm and the 1.0 mm gap-length cases. There was about a 10% difference in the pressure drop values for all four cases: 1.0 mm to 2.0 mm and 2.0 mm to 3.0 mm. When the gap-lengths for all the cases were increased further, from 3.0 mm to 4.0 mm, the pressure drop values varied only by about 5%, which was not a significant effect. In general the effect of the gap-length on the pressure drop was more significant for the rectangular fin edge channels than compared to that of the curved and rectangular fin edge channel.
Figure 7.15. Gap length vs Pressure drop for both channels

7.6 Effect of fin length (l)

Optimizing the fin length of the channel shown in Figure 7.16 was done by increasing the fin lengths on the helium and liquid salt channels from 10.0 mm to 11.0 mm and 15.0 mm. The fin lengths were also decreased from 10.0 mm to 9.0 mm and 5.0 mm.

Figure 7.16. Heat exchanger module showing the fin length parameter under study
The changes in fin length values had almost no effect on the thermal power for the liquid salt and helium channels, as shown in Figure 7.17. The maximum variation between the two extreme values of fin lengths, 5.0 mm and 15.0 mm, showed only a 2% difference.

![Figure 7.17. Fin length vs Thermal power for both channels](image)

The results of the study, Figure 7.18, show that the pressure drop decreases as the fin length is increased. The most significant was the 20% and 17% difference between the values of 5.0 mm and 9.0 mm for the helium side with rectangular and curved fin edges, respectively. This can be attributed to the boundary layers in the flow channel being restarted frequently for the case of lower fin lengths. However, the other three cases, 9.0 mm, 10.0 mm, and 11.0 mm, had only small differences. There was about a 10% difference in the pressure drop values for the 11.0 mm and 15.0 mm fin lengths on the helium side. For the liquid salt side the overall effect was less pronounced. The pressure...
drop values varied only by about 10% for both rectangular and curved fin channels when
the fin length was changed from 9.0 mm to 5.0 mm. But the pressure drop decreased only
by about 5%, for both liquid salt channel configurations, when the value was changed
from 11.0 mm to 15.0 mm. But the 9.0 mm, 10.0 mm, and 11.0 mm cases had almost no
change in the pressure drop values on the liquid salt side.

Figure 7.18. Fin length vs Pressure drop for both channels
CHAPTER 8

DISCUSSION OF RESULTS

Overall, the numerical models are able to predict the overall performance of the heat exchanger design. These models can be used to further support the experimental program for these kinds of heat exchanger channel configurations. Since the past research were based upon the flow channels as discussed in Chapter 7 and the lack of research in these kinds of channels, it is imperative to perform experiments on these kinds of channels. The complex nature of the flow was observed from the two-dimensional numerical simulations and as expected there were no recirculation regions at the leading edges of the fins for the Reynolds numbers under this study. As the presence of recirculation regions at the leading edges tend to reduce the heat transfer coefficient quite considerably.

When the numerical simulation results between the two different physical property models were compared, using the laminar flow approach, the differences were more pronounced on the hydrodynamics than the heat transfer. The difference was insignificant for the helium side while the liquid salt side had a difference of 10% in the pressure drop results. This can be due to the difference between the nature of the liquids (FLINAK) and gases (helium). The difference was less for the helium gas side because the variations of properties with respect to temperature were less compared to the liquid salt side.

112
The insignificant effect on heat transfer due to the different physical property models can be due to the data availability for the thermal conductivity of FLINAK. However, the comparison tests may even show different results when performed using turbulence modeling.

8.1 Comparison between the curved and rectangular fin edge channels

It was found that the shape of the fins had significant effect on the pressure drop while the difference in thermal performance was less. This can be due to the sharp constrictions that occur in the flow channel due to the rectangular fin edges while for a curved fin edge channel the flow blockage is smoother with no sudden changes in the flow area along the flow direction. In other words better aerodynamic shapes reduce the form drag quite considerably. The change in thermal performance was less due to the fact that the overall mass flux and flow area was constant for both the cases.

8.2 Study of the effect of fillets on the geometry

The effect of fillets was more pronounced on the fluid flow rather than the heat transfer. The effect on pressure drop was about 25% and 17% for the helium and liquid salt channels, respectively. The effect on hydrodynamics was due to the reduction and change in the cross-section of the flow area. The fillets increased the pressure drop for the helium channel while it reduced the pressure drop for the liquid salt channel. This can be attributed to the difference in the flow physics between gases and liquids.
8.3 Comparison of numerical results with analytical calculations

There were huge differences in friction factor between the numerical results for the heat exchanger channel with no gap in the flow direction and the analytical calculations for the friction factor. Some of this can be attributed to the differences in the flow channels as explained in Chapter 6, and the ratio of the geometrical parameters that were used during the experiments. The differences were more pronounced for the liquid salt side due to the fact that the behavior of a liquid could be significantly different compared to that of air, which was used in the analytical experiments.

8.4 Effect of the geometrical parameters

The effects of the changes in the geometrical parameters, as discussed in Chapter 7 is felt more on pressure drop than the heat transfer. This can be due to the surface area of the channels being so small (in the range of 2.25 mm).

When the fin thicknesses were varied it was observed that the increase in thickness value increased the pressure drop quite significantly. This occurs since a fin normal to the flow will experience a form drag due to the difference in pressure in front of and behind the fin. Also, the wake width depends on the fin thickness, and the momentum boundary layer thickness. The increase in thickness of the fins increases the wake width and the recirculation regions are wider. Thus, the pressure gradients are increased in the flow channels.

The channel heights when increased tend to decrease the pressure drops quite considerably, but a further decrease in the channel heights from the baseline helium and liquid salt channel dimensions increased the pressure drop by a huge margin. The reason
for the difference may be the top and bottom walls having a greater influence on the friction factor.

The effect of the pitch in the x-direction did have a significant effect on the pressure drop, but it only had a small effect on the thermal performance for the helium and liquid salt sides. This is due the reduction in spanwise pitch tends to reduce the transverse spacing between the adjacent fins of the same row, which in turn leads to a increased interaction between the shear layers of the fluid flowing between the fins. Thus, the reduction in pitch increased the pressure drop, while an increase in the pitch reduced the pressure drops.

When the effect of the gap lengths on the heat exchanger performance were studied it was found that the changes in the gap lengths between the fins in the flow direction affected the pressure drops quite significantly, while it did not have much of an effect on the thermal performance. The reduction in the gap lengths reduced the length of the wakes and the momentum boundary layers quite significantly. Hence, the mixing of the layers increased the pressure drops quite considerably, while the increase in gap lengths reduced the pressure drops.

The effect of fin length, despite having a significant effect on the pressure drop, the thermal performance was not significant. The changes in the fin length values directly influence the wakes between the fins and the velocity profiles at the trailing edges. The decrease in the fin length values for a constant overall heat exchanger length basically increased the number of modules per meter; thus, the breaking of the momentum boundary layers is more frequent. Hence, due to these reasons the change in fin length varied the pressure drop value quite significantly.
CONCLUSIONS AND SUGGESTIONS

In this project the flow and heat transfer across a counterflow offset strip fin heat exchanger has been studied. Three main cases are analyzed in this, namely:

1. a heat exchanger channel with rectangular fin edges
2. a heat exchanger channel with curved fin edges
3. fluid flow with constant and temperature dependent physical properties
4. a heat exchanger channel geometry which includes effects due to manufacturing methods
5. heat exchanger channel with no gap between the fins in the flow direction

The proper boundary conditions were applied and the governing equations were solved using the 2-D and 3-D numerical models developed for this project. The results obtained for these cases are summarized below:

- Both the 2-D and 3-D numerical solution results show that the heat exchanger channels with curved fin edges provide a better overall performance.
- The pressure drop is periodic with respect to the length of the channel.
- The reattachment length of the vortices formed behind the trailing edge of the fins increased with an increase in Reynolds numbers.
• The variation of physical properties with respect to temperature had no effect on the helium side.

• The temperature dependent physical properties had about a 10% influence on the hydrodynamic performance of the liquid salt channel.

• The inclusion of the fillets in the geometry showed some significant effect on the friction factor of both the fluid channels.

• The effects due to the manufacturing methods did not have an effect on the thermal performance of the either fluids.

• The investigation on the design with no gap between the fins in flow direction is a much better approximation of the heat exchanger channel configurations used to derive the empirical correlations. The design predicted a very high increase in the pressure drops, but the thermal performance was almost the same to the baseline heat exchanger design.

• The results from the numerical simulations for flow channels with no gap between the fins varied significantly from the analytical calculations.

The optimization studies were performed and the results can be summarized as follows:

• The thermal power remains constant after a certain overall length of the heat exchanger.

• Increases in fin thickness increased the pressure drops quite significantly while the decreases in the fin thicknesses reduced the pressure drops significantly.

• Changes in fin thickness value did not affect the thermal performance.
• The increase in the channel height values for the helium side degraded the thermal performance while a decrease in the channel heights increased the thermal performance.

• The effect of channel height was insignificant on the thermal performance of the liquid salt side.

• In terms of pressure drop the increase in channel heights reduced the pressure drops significantly, for both helium and liquid salt.

• The decrease in channel heights increased the pressure drops considerably for both fluids.

• The increase in the x-direction pitch degraded the thermal performance and a decrease in the x-direction pitch increased the thermal performance marginally for both sides.

• Decrease in the x-direction pitch increased the pressure drops while increasing the x-direction pitch dropped the pressure drops quite significantly for both channels by a considerable margin.

• The effect of the gap length between the fins was insignificant on thermal performance.

• Pressure drops increased significantly as the gap lengths in the flow direction were decreased. Whereas, as increase in the gap lengths reduced the pressure drops marginally.

• The effect of fin length was insignificant on the thermal performance.

• The smaller fin lengths had higher pressure drops for a fixed heat exchanger length.
• The increase in the fin length values reduced the pressure drops in a considerable manner.

Suggestions

• The results of the analytical calculations were based on the existing empirical correlations for a different type of offset strip fin heat exchanger channels and varied quite significantly from the results of the numerical solution. Therefore, there is a need to perform experiments on this type of heat exchanger channel.

• From the results, it is clear that the micro-channel heat exchangers will be a further step in keeping the design compact and providing a good performance.

• Performing a thermo-mechanical stress analysis is quite important, taking into consideration about the huge difference in the operating pressures of the two fluids and the large temperature differences between the surfaces.

• There have been articles and findings in the literature that predict the flow to become unsteady in these kinds of channels, even at low Reynolds number values. This would prove to be an important phenomenon to be considered for this heat exchanger design.

• There is a need to study the leak tightness of the solid material, due to the extreme operating conditions.
REFERENCES


123

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
VITA

Graduate College
University of Nevada, Las Vegas

Sundaresan Subramanian

Local Address:
4248, Cottage Circle, Apt 1,
Las Vegas, Nevada 89119

Degrees:
Bachelor of Engineering, Mechanical Engineering
University of Madras, Chennai, India

Publications:
Subramanian, S., Ponyavin, V., DeLosier, C.R., Chen, Y.T., Hechanova, A.E.,

Ponyavin, V., Subramanian, S., DeLosier, C.R., Chen, Y.T., Hechanova, A.E.,

Subramanian, S., Ponyavin, V., DeLosier, C.R., Chen, Y.T., and Hechanova,

Subramanian, S., Ponyavin, V., DeLosier, C.R., Chen, Y.T., Hechanova, A.E.,

124

Subramanian, S., and DeLosier, C.R., CFD Simulation of an Offset Strip-Fin Heat exchanger, ANS Students Conference 2005, Columbus, Ohio, USA, April 2005


Thesis title:
CFD Modeling of Compact Offset Strip-Fin High Temperature Heat Exchanger

Thesis Examination Committee:
Chairperson, Dr. Yitung Chen, Ph. D.
Committee Member, Dr. Anthony E. Hechanova, Ph. D.
Committee Member, Dr. Robert Boehm, Ph. D.
Graduate Faculty Representative, Dr. Jichun Li, Ph. D.