NUMERICAL DESIGN OPTIMIZATION FOR THERMAL AND PRESSURE BEHAVIOUR OF MULTIPLE CURVED CHANNEL COOLING PLATES IN ELECTRIC-VEHICLE BATTERY COOLING SYSTEMS

by

Benjamin William Banks

A thesis submitted to the
Department of Mechanical & Materials Engineering
In conformity with the requirements for
the degree of Master of Applied Science

Queen’s University
Kingston, Ontario, Canada
September, 2012

Copyright © Benjamin William Banks, 2012
Abstract

The effects of climate change along with shifts in social demands have opened up commercial possibilities for new and innovative green technology. At the head of this trend is research into new technologies for Hybrid Electric Vehicles (HEVs) and Battery Electric Vehicles (BEVs). These technologies would provide for more environmentally friendly transportation; however their current performance when compared to Internal Combustion Engine (ICE) Vehicles has led to slow adoption rates. One of the key factors that could help to increase the performance of HEVs and BEVs lies in improvement of the battery systems. Through proper thermal management of the batteries the range and performance of these vehicles can be improved, helping to increase the performance of the vehicles.

This study looks at improving the thermal management of the battery system by generating more efficient cooling plates. These cooling plates are set between battery cells and contain channels that coolant is pumped through. Through optimization of these cooling channels, the efficiency of the cooling plates with regards to the average temperature and standard deviation of temperature of the battery cell can both be increased. The power required to run the cooling system can also be reduced by reducing the pressure losses associated with the cooling plate.

Numerical optimization on three models of cooling plates was performed. The models were based on multi-inlet and outlet curved channel systems, with one model constructed using arcs and the other two using 90 degree angles. Results showed that improvements of up to 80% could be made depending on the objective functions when compared to an initial design through optimization, with straight channels providing 8% more efficient designs in terms of pressure losses over curved designs, and curved designs providing 6% more efficient designs in terms of average temperature. Analysis on the effects of varying the mass flow rate, heat flux and inlet temperature was also conducted to evaluate their effects on the optimized geometries.
This study has practical applications in helping to develop new cooling plates for commercial use through implementation of the generated design features and optimization algorithms.
Acknowledgements

I would first like to thank my supervisor Dr. Il-Yong Kim for his hard work and dedication in guiding me through all this. He has been a constant source of suggestions and ideas that have helped me to see around obstacles and arrive at this point. I have learned a lot from him both in class and in study and have enjoyed it all.

I would like to also thank the staff and professors at Queen’s University for all their help through the years. They have helped me to get to this point and have provided insights and help in everything from classes to paperwork. I started here knowing very little and they have opened the doors to teach me more than I expected.

I would like to thank my family and friends from around the world for being suggestive and supportive through these long years, and for keeping my spirits up and my goals high during the good and the bad times.

I would like to thank previous and current students of the structural and multi-disciplinary systems design lab for being good sounding boards for my worst ideas and for providing entertainment and friendship.

This research could not have been completed without our industrial partner General Motors Canada and some of the great people there such as Carlton Fuerst, Derrick Chow, Justin Gammage, Brian Tossan and all the other people that helped to provide the resources needed to get this done. It was also funded by AUTO21, a member of the Network of Centers of Excellence of Canada Program.
Table of Contents

Abstract ......................................................................................................................... ii
Acknowledgements ................................................................................................. iv
List of Symbols ......................................................................................................... xii
Chapter 1 Introduction .............................................................................................. 1
  1.1 Motivation ............................................................................................................ 1
  1.2 Objective ............................................................................................................ 2
  1.3 Contributions ..................................................................................................... 4
Chapter 2 Background ............................................................................................... 5
  2.1 Global Climate Change ..................................................................................... 5
  2.2 Shift in social demand ...................................................................................... 7
  2.3 Opening for green technology ......................................................................... 8
  2.4 Rediscovery of green transportation ................................................................ 9
  2.5 Variety of electric vehicles ............................................................................ 10
  2.6 Batteries ........................................................................................................... 12
      2.6.1 Battery Development ............................................................................... 12
      2.6.2 Battery Properties .................................................................................. 13
      2.6.3 Effects of battery temperature ............................................................... 15
  2.7 Cooling System ................................................................................................ 17
      2.7.1 Air cooling systems ............................................................................... 17
      2.7.2 Liquid Cooling Systems ....................................................................... 19
      2.7.3 Cooling system characteristics ............................................................. 21
  2.8 Design Geometry ............................................................................................. 22
  2.9 Design optimization ......................................................................................... 25
  2.10 Computational Fluid Dynamics ..................................................................... 29
Chapter 3 Methods .................................................................................................... 31
  3.1 Initial Design exploration .................................................................................. 31
      3.1.1 Design Space Exploration .................................................................... 31
      3.1.2 Model Design Considerations ............................................................... 32
      3.1.3 Candidate Models for Design Space Exploration .................................. 32
      3.1.4 Evaluation of design space exploration ............................................... 33
      3.1.5 Design space exploration results ............................................................ 35
      3.1.6 Refinement of design space exploration ............................................... 39
3.1.7 Determined optimization model................................................................. 40
3.2 Problem statement......................................................................................... 41
  3.2.1 Objective functions ............................................................................... 41
  3.2.2 Mathematical Formulation ..................................................................... 42
3.3 Model construction ....................................................................................... 43
  3.3.1 Model Geometry .................................................................................... 43
  3.3.2 Identification of design variables ............................................................. 43
  3.3.3 Fixed design parameters .......................................................................... 45
  3.3.4 Modeling process .................................................................................... 46
  3.3.5 Boundary Conditions ............................................................................. 49
  3.3.6 Secondary Model .................................................................................. 51
  3.3.7 Tertiary Model ...................................................................................... 53
  3.3.8 Initial Design Analysis ........................................................................... 53
  3.3.9 Meshing .................................................................................................. 55
3.4 Evaluation method: CFD ............................................................................ 57
  3.4.1 CFD Solver Settings ............................................................................... 57
  3.4.2 Turbulence discussion ............................................................................ 58
3.5 Governing program: optimization routine .................................................... 60
3.6 Optimization validation ............................................................................... 63
  3.6.1 Mesh Convergence Test ......................................................................... 64
  3.6.2 CFD Objective function convergence .................................................... 65
Chapter 4 Results ............................................................................................... 69
  4.1 Optimization analysis.................................................................................. 69
  4.2 Reynolds number analysis .......................................................................... 70
  4.3 Three model results.................................................................................... 72
  4.4 Comparison to initial designs....................................................................... 78
4.5 Performance of optimum designs ................................................................. 79
  4.5.1 Primary Model compared to Secondary Model ........................................ 81
  4.5.2 Primary Model compared to Tertiary Model .......................................... 82
4.6 Performance of optimization ........................................................................ 83
  4.6.1 Primary Model compared to Secondary Model ........................................ 85
  4.6.2 Primary Model compared to Tertiary Model .......................................... 86
4.7 Initial design sensitivity analysis .................................................................. 86
4.8 Non-optimal performance ............................................................................ 90
Chapter 5 Summary and conclusion ................................................................. 97

5.1 Summary of results ..................................................................................... 97
  5.1.1 Examine a number of design features for liquid cooling systems .......... 97
  5.1.2 Construct models for predicting the performance of cooling plate geometries .... 98
  5.1.3 Develop an algorithmic structure for optimizing cooling plate geometries .......... 99
  5.1.4 Validate that optimization can improve design geometries ....................... 99
  5.1.5 Determine the interaction of design features with the performance of the cooling plates ................................................................. 100
  5.1.6 Examine the effects of the boundary conditions on cooling plate geometries .... 101

5.2 Limitations of work ..................................................................................... 102
  5.2.1 Constraints on geometry ........................................................................... 102
  5.2.2 Constraints on optimization ....................................................................... 103
  5.2.3 Limited boundary conditions ................................................................. 104

5.3 Future Work .............................................................................................. 105

5.4 Conclusions .............................................................................................. 106

Bibliography ................................................................................................. 108

Appendix A Reynolds Number ........................................................................ 116

Appendix B Sample Model ............................................................................ 119
List of Figures

Figure 2-1: Representation of a one dimensional optimization as if the design space could be visualized..........................................................27

Figure 3-1: Example of a number of unique models used for design space exploration. (Temperature gradients are provided to represent effects of design features and not as a measure of performance)........................................................................33

Figure 3-2: Cooling plate representative of currently used cooling plates for electric vehicles. (Temperature gradients are provided to represent effects of design features and not as a measure of performance.) ........................................................................34

Figure 3-3: Results of design space exploration analysis for Average Temperature (K) vs. Power (W) ..........................................................................................................................36

Figure 3-4: Results of Design space exploration for standard deviation of temperature (K) vs. Power (W) ..........................................................................................................................37

Figure 3-5: Plate design for curved asymmetric channel model (representation of the half model used in analysis with a symmetry plane as the top face) ........................................................................45

Figure 3-6: Modeling process for geometry creation for optimization models showing face construction......................................................................................................................47

Figure 3-7: Modeling process for geometry showing volume construction and meshing. ..............48

Figure 3-8: Diagram showing design variable assignment for curved channel model ..................49

Figure 3-9: Secondary model of a straight channel system constrained similarly to the curved channel system (representation of the half model used in analysis with a symmetry plane as the top face) ..............................................................................................52

Figure 3-10: Diagram showing design variable assignment for secondary model .......................52

Figure 3-11: Representation of tertiary model demonstrating the systems increased design freedom (representation of the half model used in analysis with a symmetry plane as the top face) .................................................................54

Figure 3-12: Diagram showing design variable assignment for tertiary design model ...............54

Figure 3-13: Four randomly generated initial designs .................................................................55

Figure 3-14: Diagram of MATLAB control program for the optimization routine ........................62

Figure 3-15: Mesh convergence test plotting pressure drop for curved channel model against mesh density ..........................................................................................................................66

Figure 3-16: Plot of residuals tracked during sample CFD analysis ..........................................67

Figure 3-17: Objective function tracking during sample CFD analysis for residuals ..................68
Figure 4-1: Results of objective function tracking during optimization for optimized models ..... 70
Figure 4-2: Normalized results of objective function tracking during optimization for optimized models ........................................................................................................ 71
Figure 4-3: Reynolds number plot for secondary model standard deviation of temperature design .......................................................................................................................... 72
Figure 4-4: Reynolds number plot for secondary model average temperature design .......... 73
Figure 4-5: Reynolds number plot for secondary model pressure drop design .................. 73
Figure 4-6: Results for optimization and CFD analysis focusing on Average Temperature Optimization ....................................................................................................................... 75
Figure 4-7: Results for optimization and CFD analysis focusing on Standard Deviation of Temperature Optimization .......................................................................................................................... 76
Figure 4-8: Results for optimization and CFD analysis focusing on Pressure Optimization .... 77
Figure 4-9: Results of initial designs for all three models (secondary and tertiary initial models are the same) .......................................................................................................................... 80
Figure 4-10: Optimum results from four randomly generated initial designs ................. 87
Figure 4-11: Average temperature profiles for initial design analysis when optimized for average temperature ........................................................................................................................................ 89
Figure 4-12: Average temperature optimized models at varied heat flux levels ............... 91
Figure 4-13: Average temperature optimized models at varied mass flow rates ............. 92
Figure 4-14: Average temperature optimized models at varied inlet temperatures .......... 92
Figure 4-15: Pressure drop optimized models at varied heat flux levels ......................... 93
Figure 4-16: Pressure drop optimized models at varied mass flow rates ....................... 93
Figure 4-17: Pressure drop optimized models at varied inlet temperatures .................. 94
Figure 4-18: Standard deviation of temperature optimized models at varied heat flux levels .... 95
Figure 4-19: Standard deviation of temperature optimized models at varied mass flow rates .... 96
Figure 4-20: Standard deviation of temperature optimized models at varied inlet temperatures .. 96
Figure A-1: Diagram showing variables used for area and wetted perimeter calculations .... 118
Figure B-2: Resultant sample model for sample design vector ........................................ 121
List of Tables

Table 2-1: Specific Energy and Energy Density for a variety of types of batteries in comparison to gasoline [39] [40] ................................................................. 15
Table 3-1: Power required to achieve a given plate average temperature ........................................... 36
Table 3-2: Power consumption as a percentage of a given reference design required to achieve a given plate average temperature ................................................................. 37
Table 3-3: Power consumption required to achieve a given plate standard deviation of temperature .................................................................................................................. 38
Table 3-4: Power consumption as a percentage of a given reference design required to achieve a given standard deviation of temperature for a plate .................................................. 38
Table 3-5: Fixed Design Parameters for curved asymmetric curved channel model ......................... 46
Table 3-6: Material Properties for cooling plate aluminum solid regions ............................................. 49
Table 3-7: Material Properties for 50-50 water-ethylene glycol coolant ................................................ 50
Table 3-8: Boundary conditions for CFD analysis ................................................................................. 50
Table 3-9: CFD Solver settings for evaluations .................................................................................... 58
Table 3-10: Reynolds numbers for potential channel widths and velocities .................................. 59
Table 3-11: Residual values taken at 150 iterations ............................................................................. 66
Table 4-1: Numerical results for all three models optimizing for average temperature .................... 74
Table 4-2: Numerical results for all three models optimizing for standard deviation of temperature .................................................................................................................. 74
Table 4-3: Numerical results for all three models optimizing for pressure drop .............................. 78
Table 4-4: Results of initial designs for all three models (secondary and tertiary initial models are the same) ............................................................................................................. 79
Table 4-5: Performance increase results for optimized plates compared to initial designs for all three models .................................................................................................................. 79
Table 4-6: Numerical results from analysis of four randomized initial designs optimized for average temperature ........................................................................................................ 88
Table 4-7: Improvement of four initial designs over initial design for curved optimization when optimizing for average temperature .................................................................................. 88
Table A-1: Results of Velocity Calculations ....................................................................................... 116
Table A-2: Results of calculations for area, wetted perimeter and hydraulic diameter .................. 117
Table A-3: Results of Reynolds Number Calculations ........................................................................ 118
Table B-4: Design variables for sample model .................................................................................. 119
Table B-5: Calculation of amplitudes for sample model .......................................................... 119
Table B-6: Summary of channel widths for sample models ....................................................... 120
List of Symbols

\( \rho = \) Density (kg/m\(^3\))

\( \dot{m} = \) Mass Flow Rate (kg/s)

\( V = \) Velocity (m/s)

\( D_h = \) Hydraulic Diameter (m)

\( \text{Re} = \) Reynolds Number

\( \nu = \) Kinematic Viscosity (m\(^2\)/s)

\( A = \) Area (m\(^2\))

\( p = \) Wetted Perimeter (m)

\( A = \) Width of the plate (m)

\( B = \) Depth of the plate (m)

\( T_{\text{avg}} = \) Average temperature on the heat generation surface (K)

\( T_o = \) Standard deviation of temperature on the heat generation surface (K)

\( P_{\text{fluid}} = \) Pressure drop across the cooling plate (Pa)

\( \text{Cineq} = \) Inequality constraints for optimization algorithm

\( \text{Ceq} = \) Equality constraints for optimization algorithm

\( X_{\text{min}} = \) Minimum value for x design variable

\( X_{\text{max}} = \) Maximum value for x design variable

\( X = \) Design variable

\( \text{Freq} = \) Wave number or number of turns of a channel

\( \text{Amp} = \) Difference in height between the top of a wave and the bottom

\( \text{Width} = \) Width of a channel segment
Chapter 1

Introduction

1.1 Motivation

With the rise of global climate change, there is a large push right now for new green technology. Social pressures at global, national, and local levels are pushing industries to develop new and efficient products that decrease consumer’s impact on the environment and provide the same if not better services. This global trend has made its way into the automobile market where internal combustion engines account for a large part of a consumer’s impact on the environment and currently dominate the market.

The automobile industry has started to respond with more efficient gasoline powered vehicles but they have also been expanding into alternative energy vehicles. Hybrid Electric Vehicles (HEVs) have been on the market for a while now providing the efficiency of electric vehicles with the range and quick refilling advantages of an internal combustion engine. While these cars are a good stepping stone, they still depend on gasoline power for extended trips. The next step in alternative energy vehicles is Battery Electric Vehicles (BEVs) which are starting to be produced by a number of industry leaders but still have a ways to go in terms of research and development.

The heart of a BEV is its battery pack, which provides the power for all of the cars components and systems as well as its drive train meaning it is responsible for the range of the car. The key part of this system is the battery cells which when in use generate significant levels of heat depending on the power demand on them. This heat can quickly lead to degradation of the battery cells through corrosive chemical reactions quickly reducing the life cycle of the
batteries. Certain batteries can also be subject to thermal runaway, where if the heat generated by the batteries is not manageable, and the temperature of the battery reaches a critical value, the chemical reactions within the battery can occur spontaneously, generating more heat and leading to catastrophic failure as the battery melts down or explodes.

In order to combat this heat generation, cooling systems must be in place to keep the battery working in an optimal temperature range. Too low a temperature will reduce the efficiency of the battery cells, and any deviation in the temperature distribution in the battery cells can lead to non-uniform discharge rates reducing the available power to the car. All this cooling costs the battery some of its energy, meaning this system also has to be as efficient as possible in order to ensure the car has as much available power as possible.

Through implementing an efficient and effective cooling system, the range and efficiency of the vehicles can be increased along with the life cycle of the batteries leading to a more environmentally friendly and safer vehicle. This can then be applied to both BEVs to make them more comparable in performance to internal combustion engine cars, and also to HEVs meaning this has a strong impact on the electric vehicle industry as a whole.

1.2 Objective

In order to decrease the volume of Electric Vehicle battery systems, they are often constructed by stacking all the battery cells together in a long thin rectangular battery pack. In order to regulate the heat, these battery cells are interspersed with cooling plates that are constructed of a conductive material with cooling channels running through the plates. Headers on either side of the plate provide and remove the coolant, circulating it to remove the heat generated by the battery cells. The most common configuration of this is to have one cooling plate cooling two battery cells with insulation on either side before the next set.
The characteristics of these cooling plates have been studied for fields such as fuel cell vehicles and high performance electronics which have shown that significant improvements on the efficiency and performance of the plates can be achieved through variations on the geometry of the cooling channels. The majority of these studies however do not perform exploration of the design space or significant rigorous optimization on these geometries leaving an opening for research.

This study will tackle the problem of numerical optimization of an electric vehicle battery cooling plate. The design for this plate will be determined from analyzing data provided by General Motors of Canada along with previous studies in the field, along with design exploration of a variety of geometry features. With the cooling plate model determined an optimization algorithm will be constructed to perform gradient-based optimization on the cooling plate optimizing for a set of objective functions. The objective functions for optimization will be the average temperature of the plate \((T_{\text{avg}})\), the standard deviation of temperature \((T_{\sigma})\) of the plate and the pressure loss across the plate \((P_{\text{fluid}})\). These objective functions will be calculated using computational fluid dynamics to model and evaluate the cooling plates using a set of realistic boundary conditions. With optimized designs attained, this study will also look at the effects of varying the boundary conditions and how they impact the design geometry of the plate and the objective functions.

The objectives of this study are therefore:

1. Examine a number of design features for liquid cooling systems
2. Construct models for predicting the performance of cooling plate geometries
3. Develop an algorithmic structure for optimizing cooling plate geometries
4. Validate that optimization can improve design geometries
5. Determine the interaction of design features with the performance of the cooling plates
6. Examine the effects of the boundary conditions on the cooling plate geometries

1.3 Contributions

Previous studies in this area have assessed fuel cell cooling plates and electronic cooling plates as well as numerical optimization of single serpentine channels for electric vehicle cooling plates; however this study represents the first numerical optimization of multiple channel cooling plates for an electric vehicle battery pack. This study shows that numerical optimization of complex multiple channel systems can be used to dramatically increase performance measures using a variety of structured models. The results from this study show that velocity of the fluid and surface area of the channel are competing factors for the average temperature of the plate, while pressure losses are reduced by increasing the size of the channels as much as possible. The results also show that standard deviation of temperature is best achieved by balancing the surface area of the channel against the increase in temperature of the coolant.

This study also demonstrates the importance of a number of boundary conditions on the geometry of the design when applied to certain objective functions. This combined with inferences about the effects of certain geometry features on the performance of cooling plates will help to guide future designs when looking to improve their performance in key areas. This study also provides groundwork for future studies to build parametric multiple channel systems for optimization of new designs.
Chapter 2

Background

2.1 Global Climate Change

Society is currently dealing with a global climate change shift. In the past one hundred years the global temperature has risen by 0.74 K, with the period from the 1910’s to current day accounting for the majority of that heating [1]. Around the world numerous scientific and national bodies recognize global climate change and attribute a significant portion of it to human sources. Other effects have come along with global climate change, most notably of which is a rise in sea levels, along with droughts and floods in various places around the world. Estimates by bodies such as the International Panel on Climate Change (IPCC) show that the global average sea level since the beginning of the 20th century has been rising by approximately 1.7mm/year [1]. Satellite observations provided by the same body have shown that since 1993 global sea levels have been rising faster at 3mm/year [1]. A large part of this rise in sea levels is attributed to an increase in meltwater from the arctic and thermal expansion. The broader implications of global climate change are not yet fully understood. Studies have noted that the global rainfall patterns have shifted producing droughts and floods in previously normal regions, along with increased numbers of category 4 and 5 hurricanes such as hurricane Katrina [2].

While climate forecasting is an extremely complex field with numerous factors to account for, models by bodies such as the IPCC and United Nations Framework Convention on Climate Change (UNFCCC) have been created to determine best and worst case scenarios given the information available. Studies such as Stott, P. et al. [3] suggest that at best over the next 100 years the earth will experience a rise of between 2 and 4 Kelvin, while other studies such as
Rowlands, D. et al. [4] suggest that the rise in the next 40 years will be between 1.4 and 3 Kelvin. Conservative estimates would therefore put the global temperature increase over the next 100 years to between 3 and 5 Kelvin which would have the potential to cause even more extreme weather problems than society has currently started to notice along with unknown effects many of which international bodies predict to be extremely damaging not only to the natural environment but to human habitation as well.

Among suggested causes of global climate change are naturally occurring and manmade products collectively known as Greenhouse Gasses (GHG). The GHG’s are comprised of elements such as chlorofluorocarbons (CFC)’s which are manmade chemicals that were used as refrigerants, solvents, and foam blowing agents until their large effect on the environment was discovered leading to the majority of the world’s countries banning or severely limiting their sale and production [5] [6] [7]. Other more natural occurring GHG’s are gasses such as carbon monoxide (CO), carbon dioxide (CO$_2$), hydrocarbons (HC), and nitrogen oxides (NO$_x$) which while found in nature, have been produced in much larger quantities by human with the dawn of the industrial age. As the earth absorbs heat from the sun it radiates some of this heat back into space, the global temperature is raised by GHG’s when they reflect the heat radiated from the earth back towards the earth, reducing the effectiveness of the earth’s cooling [8]. As more GHG’s build up in the atmosphere more of the earth’s radiative heat is reflected leading to an increasing rise in the rate of heat increase of the earth. Global measures of these GHG’s have seen large rises since the beginning of the industrial age and since with GHG’s such as CO$_2$ rising from 320 parts per million in 1965 to 390 parts per million in 2010 [9]. There are natural methods of reducing these GHG’s such as absorption of CO$_2$ through photosynthesis or mixing of atmospheric gases into the oceans [10], as well as bio-engineered solutions such as carbon
capture and storage [11]. However, there is a limit to the quantity of GHG’s that these natural methods can clear, and unless human production of these GHG’s is reduced to a manageable level, there will be an escalating problem with global climate change.

While there are many factors to the human generation of these GHG’s, the prime increase in CO₂ has been caused by the burning of oil and fossil fuels for the generation of power. According to the International Energy Agency (IEA) Oil accounts for 37% of the world’s production of CO₂ gasses from fuel combustion [12] and as of 2007 private transportation was 95% dependent on oil and accounted for over 50% of the world’s oil consumption [13]. The United States of America found transportation to account for over 36% of their CO₂ emissions and 28% of their greenhouse gas emissions overall, with transportation also accounting for 68% of their oil consumption [14]. With industrialization and expansion in countries such as China and India, as their economies increase and their expanding populations need transport this number is only expected to rise [15].

2.2 **Shift in social demand**

With the increasing problems with global climate change, there have been some measures taken by various countries to push for laws and technology to help relieve the environment and diminish humanities impact on the environment. At the global level in 1997 a number of countries such as France, Germany, Canada and Norway along with others came together in Japan to draft the Kyoto Protocol, which attempted to globalize the issue and force governments to take efforts to reduce their GHG’s [16]. It called for reductions in overall emissions of GHG’s by at least 5% below 1990 levels between 2008 and 2012, along with demonstrating progress in achieving these commitments by 2005 along with a number of other provisions related to climate rehabilitation [16].
Since then at the national level countries have started to enact other measure such as carbon credits in the European Union [17] and United States of America [18] which force companies to spend money to purchase credits that allow them to produce GHG’s which is then fed back into programs designed to offset these GHG’s and agencies who help to run and regulate the environmental protection systems.

At the local level most major cities now run recycling programs aimed at reducing waste which helps to reduce the amount of manufacturing needed and thus reduce industrial GHG’s. Cities are also starting to move to alternative energy for public transportation such as in somewhere with their move to buses run on fuel cells. Programs have started up in numerous places to get people to start using transportation such as bicycles and carpooling to also reduce their carbon footprint showing that people are starting to get more interested in managing their own carbon footprint and looking for ways and technologies with which to do this.

2.3 Opening for green technology

This social shift has opened up a key area for growth in the green technology sector. There has been a driving force to produce more efficient and eco-friendly products. Car companies are starting to produce more gas friendly cars, working to increase their miles per gallon (MPG) and advertise for economy. While other companies are working to produce more efficient technology that consume less energy and other resources such as high efficiency washing machines and energy efficient light bulbs.

Current forms of power generation still produce large quantities of GHG’s. As of 2009 according to statistics from the IEA the United States of America depends on coal power for 50% of its energy and oil and gas plants for 22% of its energy outputting 5769 million tonnes of CO$_2$ [19]. With the push for more stringent laws and policies this has expanded research into
alternative forms of energy such as wind and solar power. Numerous countries have taken large efforts to move energy production towards more renewable forms of energy. Germany as of 2011 has renewable energy accounting for 20.0% of their total gross energy production which has resulted in an approximate reduction in CO$_2$ emissions of 126 million tonnes [20], and according to the European Commission Norway as of 2009 produced 42.4% of its energy from renewable sources [21].

While switching large scale power production to alternative energy sources would help to greatly reduce humanities impact on the environment, transportation as noted earlier still accounts for a large amount of the oil usage and GHG’s produced in the industrial world. This has led to resurgence in research and production of alternative fuels and electric vehicles. As electric cars are dependent on supplied power which currently comes from GHG producing power plants, if they are combined with these renewable forms of alternative energy, they could combine to drastically reduce humanities production of GHG’s.

### 2.4 Rediscovery of green transportation

Electric vehicles have existed since the 1830’s in one form or another. During the end of the 19th century the challenges associated with running an early internal combustion engine (ICE) or steam powered car had the public using electric cars as their primary form of transport. The reliability and cleanliness of electric cars over the ICEs, that required manual starting along with much more grease and oil, allowed them to account for the majority of vehicles on the roads. However as global oil production took off and the cost of gas declined, along with ICE technological advances such as the starter motor and increase performance the electric car was soon left behind to niche roles such as golf carts [22].
With the social and global shift towards green technology taking hold, the resurgence of the electric car has begun.

In recent years demand for electric vehicles has seen a noticeable increase. Large car manufacturers such as General Motors and Nissan have started to produce electric vehicles such as the Chevy Volt released in 2010 [23] and the Nissan Leaf released in December 2010 [24]. Along with these large companies have also come smaller start-up companies such as Tesla which specialize in only electric vehicles and promoting performance that can match conventional ICE’s such as the Tesla roadster released in 2008 [25] and the Tesla Model S released in 2012 [26]. In the United States of America sales of electric vehicles have grown from 352,274 vehicles in 2007 to 12,734,356 vehicles in 2011 [27] with HEVs accounting for the majority of the sales, showing significant growth of the industry. This resurgence is driven by the social changes and consumer demand for green technology. Despite of these figures and the fact that large companies are showing interest in developing alternative transportation, the majority of cars sold are still ICEs. If these figures are to increase electric vehicle technology needs to advance to a level where their performance can consistently be compared to current ICEs.

2.5 Variety of electric vehicles

There are currently a number of technologies in development for electric transportation including fuel cell electric vehicles (FCEVs), hybrid electric vehicles (HEVs), and battery electric vehicles (BEVs). While FCEVs present a tempting option for alternative transportation potentially providing the easy filling of vehicles comparative to current ICEs with no emissions, the technology currently cannot compete with HEVs and BEVs and so this study will be limited to HEVs and BEVs.
HEVs are a compromise between the current ICE technology and future BEV technology. They operate by combining a battery electric system with a conventional small ICE system. There are many ways in which these technologies are merged. Most HEVs such as the Chevy Volt are considered to be series hybrid vehicles utilizing electric motors for the wheels run by a battery system much like BEVs and then incorporating the small ICE as a generator to recharge the batteries allowing the owner to increase the range when needed by filling up with gasoline and charging the battery pack. Other HEVs such as the Honda Insight are considered to be parallel hybrid vehicles combining an electric drive train with a conventional ICE drive train which allows the system to utilize either the ICE or the battery system to directly drive the car. Almost all of these technologies however utilize various methods to recharge the batteries including features such as regenerative breaking where the electric motors on the wheels are used as generators when the car is breaking and external plug-in sources. [28]

BEVs on the other hand run exclusively on a battery pack and have no ICE onboard. There can be electric motors on all the wheels or a motor that runs a drive train like with an ICE, however all the power is produced, stored and recharged in the onboard battery packs. These systems often incorporate technology such as regenerative breaking to help increase the range, but rely primarily on wall plugs to recharge after the batteries have been depleted. The challenge with BEVs over HEVs and ICES is most notably the range of the vehicles. Current technology allows the electric vehicles to be more efficient than ICES in utilizing stored energy. Batteries can achieve very high energy efficiencies such as lithium-ion batteries which have an energy efficiency of 95% [29]. The downside however is that these vehicles have to carry the extra weight associated with the battery pack system which can range from 53.3kg in HEVs such as the Toyota Prius [30] to 181.4kg in BEVs [31]. Electric vehicles also cannot quickly regenerate their
power. Quick recharge systems are still under development meaning that while a BEV might be able to travel up to 300 miles on a charge, it will need a number of hours to recharge. The advantage of HEVs and ICEs is that they can quickly refuel with gas and thus their range is not limited to one tank of gas.

This therefore identifies one of the primary concerns with electric vehicles and that is the battery pack system. This is a concern for HEVs as it will help, however as BEVs are entirely dependent on the battery system this area is prime for innovation and research. Improvements in the battery pack system can lead to improvements such as a decrease in weight, an increase in efficiency, an increase in the life cycle and an increase in the range of the car, all of which contribute to bringing electric vehicle technology up to the standard of ICEs and making electric vehicles more commercially viable.

2.6 Batteries

2.6.1 Battery Development

Batteries are designed to store energy in a chemical form that can then later be released as electrical energy to power a variety of devices. Batteries come in two categories: primary cells and secondary cells. Primary cells are one-time use batteries such as the majority of batteries used in objects such as flashlights or remote controls. Secondary cells are batteries that can be recharged by running current through them in reverse to turn electrical energy into stored chemical energy.

Batteries derive their power from the interaction of an electrolyte solution sitting between two terminals. As a voltage drop is applied across the terminals electrons move from one terminal to the other through the applied circuit providing a current for the system. This process is balanced by a chemical reaction between the terminals and the electrolytic solution. This
combination of electrolyte and terminal material is what defines the type of secondary or primary cell battery such as a lead-acid battery in which the terminals are lead and the electrolyte is sulfuric acid. [32]

The most widely used battery in the auto industry is the lead-acid battery which is used in almost all cars to store some energy and supply power for the starter motor and car electronics. This battery is recharged as the car runs which allows it to constantly provide energy for the electronics and lights without needing to be plugged in. Since it runs all the electronics and lights, when it is no longer being recharged and some electronics are still on, such as leaving the lights on a parked car, the battery will slowly drain and thus be unable to provide power to the starter resulting in a car that will need to be jumped by another power source.

Since the invention of the lead-acid battery there have been significant breakthroughs in battery technology with new materials and systems being developed. Some of the newer batteries known today are those such as the Lithium-Ion-Polymer (LiP) battery that is commonly found in laptops and cell phones along with electric vehicles such as the Hyundai Sonata Hybrid [33] and the Chevrolet Volt [34], as well as Nickel-Metal-Hydride (NiMH) batteries that are also used for consumer electronics as well as extensively in HEVs such as the Toyota Prius [35] and the Honda Insight [36].

2.6.2 Battery Properties

While battery packs are large and often take up a lot of space and weight they are incredibly efficient and some of the newer models can provide strong energy densities. Batteries are often measured based on their energy capacity. This can be measured in many forms but the most common are the specific energy of the battery and the energy density of the batteries. The specific energy of the battery defines the energy stored in the battery per unit mass, while the
energy density of the battery determines the energy stored in the battery per unit volume. Since BEVs want to increase passenger space and save on weight, both of these properties are important in choosing and developing battery technology.

In order to define the specific energy and the energy density the first step is to define energy storage. This is done using the parameter ‘capacity’, which is a measure of charge. In industry the capacity of a battery is often measured in the amount of hours the battery can provide a set amount of current. For example a battery that is rated for 50 amp-hours can supply 1 amp for 50 hours (this does not necessarily mean it can provide 5 amps for 10 hours as variations in discharge rate can affect the capacity of a battery).

Batteries are also defined by their voltage. This is a measure of the expected voltage across the terminals of the battery the user can expect to see when using the battery. This voltage can be affected by many factors such as its state of charge or the age of the battery.

By combining the capacity of a battery along with the voltage of a battery, the energy in a battery can be determined. This energy is normally defined as a Watt-Hour which is the voltage multiplied by the capacity. Using this value along with either the weight or volume that the battery takes up, the specific energy or energy density has now been determined. Gasoline has a typical specific energy of 12722 Wh/kg [37] and an energy density of 9100 Wh/l [38]. Current technologies in batteries will have to advance significant amounts to be able to compete with current ICE cars in terms of range, despite the increased efficiency obtained by using electric motors. Examples of energy densities and specific energies of many of the current and emerging technologies can be seen in Table 2-1 with data from [39] and [40].
Table 2-1: Specific Energy and Energy Density for a variety of types of batteries in comparison to gasoline [39] [40]

<table>
<thead>
<tr>
<th>Composition</th>
<th>Specific Energy (Wh/kg)</th>
<th>Energy Density (Wh/L)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Gasoline</td>
<td>12722</td>
<td>9100</td>
</tr>
<tr>
<td>Lead-Acid</td>
<td>30-50</td>
<td>60-100</td>
</tr>
<tr>
<td>Lithium-Ion</td>
<td>93-200</td>
<td>114-600</td>
</tr>
<tr>
<td>Nickel-Metal-Hydride</td>
<td>80</td>
<td>140-400</td>
</tr>
<tr>
<td>Nickel-Iron</td>
<td>30-55</td>
<td>60-110</td>
</tr>
<tr>
<td>Nickel-Zinc</td>
<td>60-65</td>
<td>120-130</td>
</tr>
<tr>
<td>Zinc-Air</td>
<td>230</td>
<td>269</td>
</tr>
</tbody>
</table>

2.6.3 Effects of battery temperature

The process responsible for generating energy in the battery cell also leads to production of heat. Heat is produced in the batteries from three types of losses: ohmic, concentration and activation [41]. This means that as the load on the battery increases and the rate of chemical reaction increase, such as during a drive cycle of a gar, the heat generation of the battery does too.

Temperature has very specific effects on the performance and characteristics of batteries. Batteries have an ideal temperature in which they operate most efficiently, and thus keeping them in this range is of key concern. If batteries run at too high a temperature the chemical processes involved with generating power can cause corrosion in the system reducing the efficiency and life cycle of the battery. Corrosion effects have been studied by people such as Shim et al [42] and they have shown that increase in temperatures of batteries such as Li-Ion from 25°C to 60°C can result in capacity losses of up to 60%. A number of other studies have evaluated the capacity loss of a variety of battery chemistries such as Ramadass et al [43] Ehrlich [44] and Amine et al [45]. These studies show capacity loss ranging from 51% to 70% in temperatures exceeding 45°C and
50°C. Thermal runaway is discussed by Yang et al. [46] who suggest that thermal runaway can occur around 170°C and that in this region the terminals start to break down and this combined with decomposition or mixing with the electrolyte can lead to effects such as combustion.

Low temperatures also reduce the performance of batteries however in batteries such as lithium-ion the mechanisms leading to this poor performance are still not well understood. Ehrlich [44] noted decreases in the nominal voltage and capacity as temperatures decreased from 20°C to -20°C and Nagasubramanian [47] also noted a decrease in the capacity over a range from 10°C to -40°C with a significant portion of the decline occurring between -20°C and -40°C. Zahng e al. [48] performed similar testing at a lower discharge rate between 20°C and -40°C and noted the same effects.

The distribution of temperature or heat generation in a battery cell is also important. Variations in temperature across the system can lead to either high or low temperature dependent problems in select areas of the battery cell, and as the cell is a unit as a whole, individual weak parts degrade the system as a whole resulting in decreased life cycles and efficiencies despite average temperatures being within acceptable parameters. Studies such as Pesaran et al. [49] have been conducted on lead-acid batteries and shown heat generation primarily occurs around the terminals of the battery. Kim et al [50] also showed that heat generation occurred most predominantly around the terminals in laminate lithium-ion batteries through numerical modeling and thermal imaging. These results indicate that any effort to normalize the distribution of heat will have to be tailored to the chemistry of the battery in use.

When combining these three concerns with battery temperature it is clear that systems need to be in place to manage the temperature of the battery and ensure that it is kept uniform across the battery cells and within operating conditions to get the best life span out of the battery.
and to ensure it is running as efficiently as possible. With the quantity of battery cells required to provide enough energy to match the range of ICE engines, small variations in the efficiency of each cell can combine to significantly reduce the energy capacity of the unit as a whole.

2.7 Cooling System

The cooling system is a critical component of any battery system and operates in order to keep the batteries running within their most efficient temperature range. There are many methods used to cool battery packs such as air cooling, phase change material cooling, and liquid cooling. Each of these designs has their own advantages and disadvantages along with various methods of utilizing the cooling system to maximize efficiency. Despite the type of coolant used however, most cooling systems require incorporation of cooling plates that can transfer the heat from the battery cells to surface area that the coolant can interact with. These coolant plates are thus a critical component of the cooling system which can significantly affect the effectiveness and efficiency of the cooling system no matter what coolant is used.

2.7.1 Air cooling systems

Air cooling systems have been employed in many types of technologies such as in computer chip systems along with some of the newer types of electric vehicles. The current generation of the Nissan Leaf employs air cooling to ensure the batteries are running within their efficient range. The Nissan Leaf pulls conditioned air from within the cabin and runs this air through the battery pack to manage the temperature [51]. In this way they can maximize the utilization of the air by using it for two purposes, thus reducing the amount of power drawn on the batteries for components other than the electric motor, which helps to increase the range of the car.
Air cooling has a number of advantages over other types of cooling systems that make it desirable for certain cases. The system does not need to be sealed in order to ensure nothing is leaked which means that the system is less delicate than other systems and require less strict measures on its location and integration with other systems. Air can also easily be drawn from multiple sources meaning that if the cabin air is hot, and the outside air is cold, the cooling system can quickly change the temperature of the air being used and thus maintain a very specific temperature. Since air is abundant outside the vehicle, this also means that the car does not have to carry any onboard to drive the cooling system which helps to reduce the weight of the car over those using other types of cooling systems.

The downside to air however is that it has a lower heat transfer coefficient than most liquid coolants thus requiring more air to be used to obtain the same amount of heat transfer as other types of cooling systems. Air systems often also require more surface area to compensate for this meaning more space often has to be devoted to heat sink surface area which can lead to a larger battery pack in certain situations.

Many studies have been conducted into cooling plates for air cooling systems. The majority of these studies have been for computer heat sinks however the design features of cooling plates such as these can have parallels for many different types of coolants used. With regards to computer heat sinks studies such as Bau [52] looked at the methods by which the geometry of conduits in micro heat exchangers could be modified to increase the heat transfer rate of the system. He found that tapering the width of the channels so that the gap between them grew smaller as the air passed through them allowed the air to increase in velocity which helped to counter the decrease in the heat transfer rate associated with the increase in temperature the air experienced when cooling the system. Through this method he managed to reduce the
temperature gradient across the plate by 95% using a second order polynomial as a function for the channel width. For electric vehicle cooling studies have looked at a variety of aspects such as Mahamud and Park [53] who looked at methods by which management of the increase in the temperature of the air could be achieved. They looked at a method where the inlet and outlets of the cooling system could be alternated. Through this method the temperature gradient across the system could be better managed which would allow the inlet temperature to be hotter while ensuring all cells were below a maximum temperature. Through this method they showed that they could reduce the temperature gradient across the system by about 72%.

2.7.2 Liquid Cooling Systems

Liquid cooling systems are another significantly used mechanism for maintaining the temperature of a system. Liquid cooling systems are often used when large quantities of heat need to be removed from the system due to the increased heat capacity that most liquid coolants have over air coolants. The current generation of Chevy Volts use a liquid cooling system to ensure that their battery systems are maintained at a constant temperature running a liquid coolant through cooling plates with cooling channels built in to them that transport the liquid coolant throughout the battery pack and battery cells. The coolant can be run through multiple systems such as the cabin air conditioning system or heaters to regulate the temperature of the coolant and thus provide the temperature gradient needed to maintain the batteries at the needed operating temperature.

Liquid cooling has many advantages over other types of cooling systems. The most significant advantage of liquid cooling systems comes from the increased heat capacities that liquid systems have over other systems such as air cooling systems. This increased heat capacity means that the liquid system can fit in a much smaller space since less coolant is required to
remove enough heat, and this space saving is extremely important when it comes to factors such as passenger room in electric vehicles. Since liquid cooling systems need to be self-contained it also means that liquid cooling system temperatures are easier to regulate and can be maintained at much more specific temperatures giving more precise control to the cooling system.

The self-containment of liquid cooling systems also has some disadvantages however. Liquid cooling systems require the system to carry the coolant around with it, which leads to an increase in weight dependent on how much coolant is needed for the system. The coolant also has to be separated meaning it is much harder to integrate it with other systems like air conditioning units used for other purposes which makes it harder to make use of multiple interconnected systems leading to potential reductions in the efficiency of the system. The self-containment of liquid cooling systems also means that the battery packs have to be sealed which makes them harder to integrate into systems and requires them to be much more isolated.

As far back as 1981 studies have been conducted on liquid cooling for micro channel heat sinks. Tuckerman and Pease [54] examined the effects for channel geometry for liquid cooling of high-performance heat sinks and showed that high-aspect ratio channels are desirable for reducing thermal resistance. Study has also been done into the effects of the cross sectional geometry of the cooling channels such as Fisher and Torrance [55] [56]. In their first study they look at the effects of convex cross sections and their effects on the optimal fin thickness showing that curved channel shapes can reduce the required separation between channels allowing for more channels in a cooling plate. Their second study looks at three cross sections for liquid and gas cooling systems; rounded corner rectangle, ellipse and diamond cross sections that showed similar conclusions to those of the first study.
2.7.3 Cooling system characteristics

Cooling systems of any design are one of the most critical components of an electric vehicle and interact with the car on a variety of levels. As the vehicle is an enclosed system, the cooling system has to draw energy from the car which means power used to keep the batteries operating is detracting from the power available for increasing range. This means there needs to be a balance between keeping the batteries in the most efficient temperature range for operation, while using as little power in the most efficient way to keep the batteries in this range. Since the cooling plate is one of the most critical components of a cooling system in being able to efficiently transfer this heat away from the battery cells there is a lot of potential for analysis and research into increasing the effectiveness of these plates.

Highlighting liquid cooling systems there are a number of design features that have a large impact on the characteristics and performance of the plate. These design features also place constraints and design goals on the plates which are important when dealing with the design of them. Some of the most important design features when dealing with cooling plates are the geometry of the channels, the flow rate of the coolant and the heat being generated by the battery. These boundary conditions and characteristics interact with each other in complex ways requiring an in depth study to identify the key features of any cooling plate design that can help to increase its efficiency. While all these boundary conditions and characteristics are important, some provide for more analysis then others. The design geometry of the cooling plate is a diverse and open area. There are a multitude of different shapes and structures that can vary wildly and all interact differently. Compared to this the mass flow rate of the coolant is a very scalar variable which means while it is important to consider it is best analyzed as a part of a larger study. The heat generation of the battery too exists as a scalable variable, and while it will change
significantly and at varying rates throughout a drive cycle, it too is best analyzed as part of a larger system.

2.8 Design Geometry

There are many aspects to look at when examining and optimizing the geometry of cooling plates. Aspects such as the number of channels, along with the number of inlets and outlets and the cross sectional geometries of the channels are important in determining the type of cooling system and are often based on the constraints due to the application of the system. There is no way to define the best geometry for cooling systems but by applying a specific set of conditions for a specific application the range of cooling designs can be limited.

The cross sectional shape of the channel is an important factor in the construction of cooling plates. It can be defined by the type of cooling system such as air cooled electronic heat sinks which often use open ended large triangular fins, or liquid cooling systems which need closed shapes. It can also be defined by the method in which the cooling plate is manufactured such as electric vehicle battery pack cooling plate such as the Chevrolet Volt in which two sides are stamped and thus cannot have too complex a shape. Many studies have been conducted on this such as the previously mentioned Fisher and Torrence studies [55] [56] along with other studies such as Salimpour et al [57]. They show that curved channel geometries can achieve strong results however Salimpour noted that triangular channels could allow for better stacking of channels allowing for a greater channel density which could achieve better results. Other studies look at a specific cross sectional shape and vary it through the length of the channel such as the previously mentioned Bau [52] and other studies such as Wei and Joshi [58]. Weis and Joshi examined a rectangular cross section in order to determine the best aspect ratio and fin thickness determining that there are ideal ratios dependent on boundary conditions such as pressure drop.
and flow rate. These studies however focus on long straight stacked channels and thus are not always applicable to non-linear channels that cannot always incorporate these unique cross sections.

When looking at non-linear channel geometries the main aspect that affects the temperature of the system becomes the path that the channel takes. Many previous studies have been done on this examining a broad range of methods in getting the coolant from one side of the plate to another. Studies have looked at using a single inlet and outlet with a long single serpentine channel traveling through the plate. Yu et al. [59] examined single channel systems that looped back upon themselves (known as multi-pass systems) with a variety of path lines such as channels that travelled back and forth left to right, along with more complex designs such as channels that spiraled into the center and then back out again. They found that this spiral method allowed for the most uniform temperature distribution across the plate as it accounted best for the increase in temperature of the coolant as it travelled through the system. Other studies have examined both single channel and multi-channel systems with a single inlet and outlet such as Cho et al. [60], Chen et al. [61], Choi et al. [62] and Kurnia et al. [63]. Cho et al. looked at single inlet single outlet systems with three models; two headers with straight channels across the plate, four headers with two sets of straight channels each across a different half of the plate and four headers with four sets of straight channels each across a different quadrant of the plate. They also examined the effect that the hydraulic diameter of the channels had on the performance of each of the models. They showed that below a hydraulic diameter of 20mm the single set of channels have the best average temperature, but above that the double set of channels performed best. Chen et al. also looked at single inlet single outlet systems with both single channels and multi channels. They combined single pass back systems and double pass back systems with both one
channel and two channel designs to create a variety of systems. They found that serpentine channel cooling plates worked more efficiently than parallel channel cooling plates and attributed this to the poor distribution of fluid across the parallel channels. This is a common theme with studies focusing on single inlets and single outlets as proper distribution of fluid often cannot be achieved. This problem could be resolved through the use of multiple inlets and outlets with even distribution. Choi et al. also found that single channel systems are better. The study looked at both single and multiple channel systems with a variety of pass back loops in different configurations to other studies, however they again find that parallel channels provide lower cooling efficiencies overall. They did however show that parallel channel configurations can provide better cooling than some single channel systems showing potential for optimization if a solution to fluid distribution is employed. Kumia et al. looked at multiple channel and single channel systems however they examined systems with far more channels or pass backs. These results further show the dominance of single channel spiral systems and again attribute it to the poor distribution of fluid across the channels along with the balance of heated and cooled coolant that can be achieved through spiral designs with a large number of pass backs.

A clear theme emerges from previous studies to show that multiple channels produce less effective cooling than single channels with pass back features, however very little study has been done into how multiple inlets and outlets can affect this problem. Given the relative closeness between systems and the large disadvantage that multi-channel systems have due to poor coolant distribution, this factor may be able to bring multi-channel systems even with or help them to exceed single channel system performance.
The basis for geometry optimization done for this study comes from previous work which can be found in Jarrett and Kim [64]. The work done in this study builds upon the foundations created by this study and attempts to expand to new areas of the design domain.

2.9 Design optimization

Design optimization is the process by which a design is changed in order to minimize a certain characteristic of the design. Optimizations are often represented by a mathematical formulation such as that seen in Equation 2-1.

**Equation 2-1: General form of a mathematical representation of optimization**

\[
\text{Minimize: } F(x)
\]

subject to: \[
\begin{align*}
C_{\text{ineq}}(x) &\leq 0 \\
C_{\text{eq}}(x) &= 0 \\
x_{\text{min}} &\leq x \leq x_{\text{max}}
\end{align*}
\]

where:

- \(x\) is the design
- \(F(x)\) is the objective function
- \(C_{\text{ineq}}\) are the inequality constraints
- \(C_{\text{eq}}\) are the equality constraints
- \(x_{\text{min}}\) and \(x_{\text{max}}\) are the bounds

In this case the design would be represented by design vector \(x\), where \(x\) is changed in order to improve the objective function \(F(x)\). As \(x\) is changed \(F(x)\) is measured and once \(F(x)\) has reached its lowest possible value the design is considered optimized. This process is often subject to constraints, which are represented by \(C(x)\). These constraints limit the region in which \(x\) can exist and often represent physical attributes such as the maximum length of a beam, or the smallest allowable pipe diameter. In order to perform automatic, computational optimization, an external optimization algorithm must be able to change the design features. This
means the system needs to be defined parametrically. As the optimization progresses new designs will be created starting at some initial design and progressing towards the global optimum design. As a one dimensional example, Figure 2-1 shows a representation of a system as if the design space can be visualized.

From this figure it can be seen that there are a number of local optima that can be achieved but one global optimum that contains the lowest possible value for the objective function $F(x)$.

Gradient-based optimization relies on determining the slope of the objective function and moving the design variable along this gradient towards its minimum value. The method for doing this is often a numerical approximation of the gradient at specific intervals and the finite difference method is often employed. This works by taking the value of the objective function at a set design variable, and then that design variable is perturbed slightly and a new objective function is calculated. This process is known as a sensitivity analysis and as the number of design variables is increased, it must be repeated for each one. From this analysis a gradient can be created in order to determine the best direction for the optimization to continue in. At this point the magnitude of the next step has to be calculated to determine what the next design variable will be along the gradient. In the example case shown here this magnitude is determined through a one dimensional line search where the gradient is followed to an estimated minimum point at which sensitivity analysis is conducted again for the next iteration. This is the most common way to determine the magnitude; however other methods exist for more complex systems such as non-linear systems. While this can be visually represented in cases of only a couple of design variables through lines and planes, once design variables increase beyond two the system has to be represented mathematically.
Figure 2-1: Representation of a one dimensional optimization as if the design space could be visualized

Figure 2-1 also shows some of the challenges and constraints that are associated with optimization. The first limitation to note is that there are usually many local optima, and while they can be visualized in this example, in practical applications the function of the optimization cannot be completely visualized especially when dealing with multiple design variables. This means that there will always be a level of uncertainty as to whether or not the design is at the global optimum solution. There are a number of ways to avoid getting stuck in a local optimum and they each have their own advantages and disadvantages which means they need to be properly applied to specific applications. The simplest method of combating systems with numerous local optima is to properly manage the step size or the magnitude that the design variable is changed. If a minimum value of the step size is set large enough, it may be able to
allow the optimization to skip many of these smaller local optima and follow a more global gradient of the system. This process however reduces the fidelity of the optimization meaning that a second optimization might need to be conducted afterwards to further refine the model. Determining a proper step size is obviously another difficulty.

The second limitation to note is that constraints can impose problems on the optimization which means reaching the global optimum from one initial design is impossible. In the case of Figure 2-1 it can be seen that by following the gradient from left to right the optimization process will never be able to achieve the global optimum without a step size that would be far too large to provide reliable results. This shows the importance of using multiple initial designs. The optimized designs are evaluated as a whole to ensure that the objective function has converged which can help to validate that a global optimum design has been achieved, or to identify that constraints may be isolating an optimal section and that the optimization needs to be reassessed in this region. In cases of a large number of design variables, this can present a problem however.

In order to accurately optimize every area of the design domain an infeasible amount of computational time would be required. This means that while using multiple initial designs, and employing strong optimization techniques, there will always be a measure of uncertainty as to whether the global optimum has been achieved.

Practically, there is no way of proving global optimality unless the entire design space is thoroughly examined using an exhaustive search method with a sufficiently high resolution. In real-world problems, it is usually infeasible to do this and thus the goal is therefore to determine an effective local optimum solution.
2.10 Computational Fluid Dynamics

Computational fluid dynamics (CFD) is a numerical approach to approximating the behavior of fluid systems. Numerical techniques have advanced to the point where a wide variety of physical governing equations can be solved and approximated such as heat transfer, combustion, and fluid flow. The most common numerical technique used to analyze these phenomena is the finite volume method in which a system is divided into a number of discrete cells. The physical modeling is defined and boundary conditions are applied to the system such as defining the type of fluid, or its initial velocity. The most fundamental equations used for CFD flow are the Navier-Stokes equations. Simplifications can be made based on the type of system being studied and these equations are then combined with equations for conservation of mass, momentum, energy and state along with equations such as those for incompressible fluid to generalize the system to an incompressible three-dimensional flow including heat transfer. These equations are then applied to specific regions of the system through the use of the divergence theorem and through conservative methods the calculations can be propagated through the discrete cells. The equations are discretized into a set of algebraic equations and an iterative method is used to solve the system.

These equations and systems can be quite accurately modeled in slow steady systems where there are smooth transitions between fluid and boundary layers and can in simple cases be modeled with very little approximation. These systems are often called laminar systems and are generally systems with a low Reynolds number, which is a dimensionless number representing the ratio of inertial forces to viscous forces, where kinetic energy in the system decreases due to fluid viscosity. As the Reynolds number increases past a point the behavior of the fluid starts to become unsteady at which point the systems starts to transition to a turbulent system. In turbulent
systems features such as irregularity, diffusivity and effects such as vortices and eddies start to appear. These systems require more complex equations to represent which are often specific to unique turbulent structures.

In order to accurately employ CFD, care must be taken at the start to ensure that the system is set up in a way that applies the correct equations and boundary conditions to the system to allow for accurate analysis. The first step of a CFD analysis is to construct a discretized representation of the system to be analyzed. Boundary conditions are applied such as the initial velocity of a fluid or a heated surface, and the appropriate models are selected to represent the system such as energy models for heat transfer or turbulence models. Once the system is established the iterative process takes place in order to solve the system. Upon convergence data can then be analyzed to determine how the system performed and to obtain results.

CFD is a complex system and without proper application of techniques and the use of correct equations and methods the results generated can be extremely inaccurate. With proper techniques however simple systems can be solved with almost no error, and more complex systems can be solved with a relatively high degree of accuracy. This study will be looking at an incompressible system with heat transfer which will require the use of an energy model along with some user-defined functions to properly represent the properties of the fluid.
Chapter 3

Methods

3.1 Initial Design exploration

3.1.1 Design Space Exploration

With the intent of exploring systems with multiple inlets and outlets there were a variety of design geometries that could be chosen. In order to properly identify design features that were efficient and would provide for good optimization analysis, a design space exploration was undertaken. The purpose of the design space exploration was to generate a number of unique and varied design geometries and to evaluate them over a range of boundary conditions. Through comparison of these unique designs, a strong candidate for optimization could be determined and this design could be used for a more high fidelity model and optimization algorithm.

A variety of unique designs were created for comparison. The models were generated and run using the same boundary conditions except for the mass flow rate which was varied. The mass flow rate was varied and data for the average temperature of the heat generation plane and the standard deviation of the heat generation plane was obtained. Using this data graphs were created to showing average temperature and standard deviation of temperature against the power used to run the plate. These graphs could provide a measure of the efficiency of different design features and determine a strong candidate for optimization.
3.1.2 Model Design Considerations

General Motors Canada provided technical data that helped to construct some cooling plates that have similar characteristics to current electric vehicle cooling system technologies. It was indicated that some electric vehicle battery cells are 160mm by 200mm by 1mm in size which helped to set the overall plate geometries. It was also identified that the cooling channels took up about \( \frac{3}{4} \) of this 1mm depth. Following this information a number of 3D trial models were created for design space exploration and analysis with a CFD solver Fluent. In order to run these models under realistic conditions running parameters were also provided. Given some technical data on the quantity of heat each battery cell generates and the quantity and rate that coolant is circulated in the battery system a rate of heat generation for the faces of the cooling cell in contact with the battery cell was determined to be approximately 500 W/m\(^2\). The mass flow rate for the design exploration would be varied to provide a number of data points to compare designs with. The models run were also employing a symmetry method to cut down on their size, cutting the model in two along the depth parameter to make them 0.5mm thick with a symmetry plane along the cut leaving the 3D models as 160mm by 200mm with a depth of 0.5mm.

3.1.3 Candidate Models for Design Space Exploration

A number of the design candidates can be seen in Figure 3-1. From top row, left to right the models are: Curved symmetric system, Channels with cylinders system and single cavity with cylinders system. From bottom row, left to right the models are: Curved asymmetric system, Straight channel system and single cavity system.

A baseline for performance was also generated to give a point of reference for the design space exploration. Images provided by General Motors Canada showed a type of cooling plate...
design for electric vehicles. A model was then created based on this design which could give an approximate frame of reference which can be seen in Figure 3-2.

3.1.4 Evaluation of design space exploration

These models were constructed using GAMBIT to create a 3D geometry and mesh. They were then run through a CFD solver FLUENT to simulate boundary conditions and obtain results. The boundary conditions are the same as those seen in Table 3-6 to Table 3-9. In order to properly compare the models they were run at mass flow rates ranging from 0.0001kg/s to 0.01kg/s.
Figure 3-2: Cooling plate representative of currently used cooling plates for electric vehicles. (Temperature gradients are provided to represent effects of design features and not as a measure of performance.)

This range produced a number of data points, however due to the variety in the width and number of channels, along with the variety in geometry features the mass flow rate was not an effective measure to base results on. One of the three key features of cooling plate performance is efficiency and given the data obtained from these simulations, pumping power was used as the basis for comparison. The power for each plate was obtained using Equation 3-1.

**Equation 3-1: Equation for converting output variables to Power**

\[
Power = \frac{\text{Mass Flow Rate } \left( \frac{kg}{s} \right) \times \text{Pressure Loss (Pa)}}{\text{Density } \left( \frac{kg}{m^3} \right)}
\]

The curves for power were then plotted against the average temperature of the heat generation plane and the standard deviation of the heat generation plane.
3.1.5 Design space exploration results

The results for the design exploration can be seen in Figure 3-3 and Figure 3-4. These figures show the results of the average temperature of the heat generation plane, and the standard deviation of the heat generation plane plotted against the calculated power.

Figure 3-3 has had its y-axis limited to 303K despite the results ranging up to the 320K range. Since this data is being used to determine the advantages and disadvantages of various geometry features, the results in that range are too close together to provide significant insight into the performance of the plates, and by limiting the y-axis to 303K the advantages in the higher performance regions are more defined and distinct. Average temperature values at the 302K range correspond to mass flow rates of approximately 0.001kg/s and all the data only ranges up to 0.01kg/s which is the reason why some of the data lines are shorter than others. This also applies to Figure 3-4, with the data being limited to a standard deviation of 1K.

These graphs provide a good basis for comparative analysis between the various design exploration designs, however in order to better compare the performance of the various designs power consumption values were taken at four average temperature values; 303K, 302K, 301.5K and 301.05K and four standard deviation of temperature values; 0.35K, 0.55K, 0.70K and 1.00K. These values were chosen to give the best comparison between as many of the models as possible while containing enough of a variance to establish increases or decreases in performance over a range of powers. For some of the data sets that were extremely close but did not quite reach the chosen reference points linear interpolation was used on the last data points to obtain the value, however in cases where the data stopped short the linear interpolation would have introduced too much error to be of comparative value and thus the data was excluded (as seen in Table 3-3 and Table 3-4).
Figure 3-3: Results of design space exploration analysis for Average Temperature (K) vs. Power (W)

Table 3-1: Power required to achieve a given plate average temperature.

<table>
<thead>
<tr>
<th></th>
<th>303 (K)</th>
<th>302 (K)</th>
<th>301.5 (K)</th>
<th>301.05(K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>General Motors</td>
<td>0.02649 J/s</td>
<td>0.07553 J/s</td>
<td>0.21419 J/s</td>
<td>0.92381 J/s</td>
</tr>
<tr>
<td>Representative</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Single Cavity with Cylinders</td>
<td>0.00248 J/s</td>
<td>0.00782 J/s</td>
<td>0.02592 J/s</td>
<td>0.20817 J/s</td>
</tr>
<tr>
<td>Curved Symmetric</td>
<td>0.00184 J/s</td>
<td>0.00525 J/s</td>
<td>0.01565 J/s</td>
<td>0.10971 J/s</td>
</tr>
<tr>
<td>Channels with Cylinders</td>
<td>0.00087 J/s</td>
<td>0.00305 J/s</td>
<td>0.00554 J/s</td>
<td>0.01125 J/s</td>
</tr>
<tr>
<td>Curved Asymmetric</td>
<td>0.00063 J/s</td>
<td>0.00222 J/s</td>
<td>0.00403 J/s</td>
<td>0.01091 J/s</td>
</tr>
<tr>
<td>Straight Channel</td>
<td>0.00055 J/s</td>
<td>0.00192 J/s</td>
<td>0.00280 J/s</td>
<td>0.00775 J/s</td>
</tr>
<tr>
<td>Single Cavity</td>
<td>0.00039 J/s</td>
<td>0.00131 J/s</td>
<td>0.00201 J/s</td>
<td>0.00451 J/s</td>
</tr>
</tbody>
</table>
Figure 3-4: Results of Design space exploration for standard deviation of temperature (K) vs. Power (W)

Table 3-2: Power consumption as a percentage of a given reference design required to achieve a given plate average temperature.

<table>
<thead>
<tr>
<th>Design Type</th>
<th>303 (K)</th>
<th>302 (K)</th>
<th>301.5 (K)</th>
<th>301.05 (K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>General Motors Representative</td>
<td>100.00%</td>
<td>100.00%</td>
<td>100.00%</td>
<td>100.00%</td>
</tr>
<tr>
<td>Single Cavity with Cylinders</td>
<td>9.34%</td>
<td>10.35%</td>
<td>12.10%</td>
<td>22.53%</td>
</tr>
<tr>
<td>Curved Symmetric</td>
<td>6.96%</td>
<td>6.95%</td>
<td>7.30%</td>
<td>11.88%</td>
</tr>
<tr>
<td>Channels with Cylinders</td>
<td>3.27%</td>
<td>4.04%</td>
<td>2.59%</td>
<td>1.22%</td>
</tr>
<tr>
<td>Curved Asymmetric</td>
<td>2.39%</td>
<td>2.93%</td>
<td>1.88%</td>
<td>1.18%</td>
</tr>
<tr>
<td>Straight Channel</td>
<td>2.08%</td>
<td>2.54%</td>
<td>1.31%</td>
<td>0.84%</td>
</tr>
<tr>
<td>Single Cavity</td>
<td>1.46%</td>
<td>1.73%</td>
<td>0.94%</td>
<td>0.49%</td>
</tr>
</tbody>
</table>
Table 3-3: Power consumption required to achieve a given plate standard deviation of temperature.

<table>
<thead>
<tr>
<th></th>
<th>0.35 (K)</th>
<th>0.55 (K)</th>
<th>0.70 (K)</th>
<th>1.00 (K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>General Motors Representative</td>
<td>0.10822 J/s</td>
<td>0.06096 J/s</td>
<td>0.03234 J/s</td>
<td></td>
</tr>
<tr>
<td>Single Cavity with Cylinders</td>
<td>0.05790 J/s</td>
<td>0.01119 J/s</td>
<td>0.00403 J/s</td>
<td></td>
</tr>
<tr>
<td>Curved Symmetric</td>
<td>0.03175 J/s</td>
<td>0.00912 J/s</td>
<td>0.00346 J/s</td>
<td></td>
</tr>
<tr>
<td>Curved Asymmetric</td>
<td>0.05544 J/s</td>
<td>0.00848 J/s</td>
<td>0.00443 J/s</td>
<td>0.00213 J/s</td>
</tr>
<tr>
<td>Channels with Cylinders</td>
<td>0.02019 J/s</td>
<td>0.00711 J/s</td>
<td>0.00388 J/s</td>
<td>0.00244 J/s</td>
</tr>
<tr>
<td>Straight Channel</td>
<td>0.01139 J/s</td>
<td>0.00478 J/s</td>
<td>0.00271 J/s</td>
<td>0.00174 J/s</td>
</tr>
<tr>
<td>Single Cavity</td>
<td>0.00945 J/s</td>
<td>0.00393 J/s</td>
<td>0.00216 J/s</td>
<td>0.00144 J/s</td>
</tr>
</tbody>
</table>

Table 3-4: Power consumption as a percentage of a given reference design require to achieve a given standard deviation of temperature for a plate.

<table>
<thead>
<tr>
<th></th>
<th>0.35 (K)</th>
<th>0.55 (K)</th>
<th>0.70 (K)</th>
<th>1.00 (K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>General Motors Representative</td>
<td>100.00%</td>
<td>100.00%</td>
<td>100.00%</td>
<td></td>
</tr>
<tr>
<td>Single Cavity with Cylinders</td>
<td>53.50%</td>
<td>18.36%</td>
<td>12.45%</td>
<td></td>
</tr>
<tr>
<td>Curved Symmetric</td>
<td>29.34%</td>
<td>14.96%</td>
<td>10.72%</td>
<td></td>
</tr>
<tr>
<td>Curved Asymmetric</td>
<td>100.00%</td>
<td>7.83%</td>
<td>7.26%</td>
<td>6.60%</td>
</tr>
<tr>
<td>Channels with Cylinders</td>
<td>36.41%</td>
<td>6.57%</td>
<td>6.37%</td>
<td>7.55%</td>
</tr>
<tr>
<td>Straight Channel</td>
<td>20.54%</td>
<td>4.41%</td>
<td>4.44%</td>
<td>5.38%</td>
</tr>
<tr>
<td>Single Cavity</td>
<td>17.05%</td>
<td>3.63%</td>
<td>3.54%</td>
<td>4.44%</td>
</tr>
</tbody>
</table>

The results for the average temperature analysis can be seen in Table 3-1, and the results for the standard deviation of temperature can be seen in Table 3-3. In order to provide for an easier analysis Table 3-2 and Table 3-4 show the results as compared to a reference design. The reference design for Table 3-2 is always the design based on the General Motors images, however for Table 3-4 at 0.35K the reference design is the Curved Asymmetric Channel design as some of
the less efficient designs did not manage to reduce the standard deviation of temperature to this value.

3.1.6 Refinement of design space exploration

As seen in Table 3-2 and Table 3-4 there are large improvements to be made using design space exploration. Significant improvements were made by all models over the design based on the General Motors based-line design, and there is a strong correlation between the efficiency of designs with regards to the average temperature in comparison to the standard deviation of temperature. The one exception to this is the channels with cylinders design being more efficient in the standard deviation of temperature and thus overtaking the curved asymmetric channels design.

While there are a few designs that stand out in this design space exploration, given limited time for research, results needed to be narrowed down to one model that could be expanded on through optimization. The most promising designs for optimization were thus the three best performers in terms of the average temperature of the heat generation plane.

The single cavity performed the best out of all the designs with the straight channel design following closely behind and the curved asymmetric channel design close behind that. While the results of the single cavity design are appealing other challenges exist that would make it a difficult design to employ. Due to the nature of the way in which these cooling plates are combined with battery cells and insulations to form packs, the structural integrity of these cooling cells would be of key concern; they have no structural support in the center, and are likely to crush in the center of the single cavity when pressed into a pack. Introducing structural supports for the pack would start to move the design towards that of the single cavity with cylinders.
design, and given the results of the single cavity with cylinders design, it can be seen that the single cavity design would be quickly overshadowed by other designs.

Problems with the straight channel design lie in the measure of its efficiency compared to its volumetric flow rate of coolant. While the straight channels can provide low temperatures and standard deviations of temperature, they require larger channels to achieve these then comparative designs. When using comparative coolant volumetric flow rates the straight channel systems become less efficient then comparative designs. The straight channel systems require larger channels, meaning more coolant in the thermal management system. This extra coolant will account for significant weight in the system, and in order to reduce this weight a more efficient system in volumetric flow rate is preferred.

3.1.7 Determined optimization model

With the two best performing models excluded from optimization potential, the curved asymmetric channel design is left as the potential candidate for optimization. This model has strong structural characteristics with material between channels providing support for the compression during construction of the battery packs. The potential for optimization is strong due to a large number of design variables including channel locations, frequency and amplitude of the curves and features such as channel height. Given these two factors and the performance in the design space exploration this model was chosen as the final model to be optimized for this study.
3.2 Problem statement

3.2.1 Objective functions

The three primary performance criteria of the cooling system were identified as the heat being removed from the system, the uniformity of the temperature within the system and the power loss associated with running the cooling system. These criteria however all have various ways of being quantified within a cooling plate. The heat being removed could be measured through the variation between the inlet and outlet temperature, while the uniformity of the temperature could be measured by evaluating the range of temperatures throughout the system. The performance criteria were thus selected based on their relevance to the problem at hand and to allow for the most efficient calculation during optimization. The heat being removed from the system was evaluated by measuring the average temperature ($T_{\text{avg}}$) of the plane where the heat flux was being applied too. This plane represents the face that will mate with the battery cell and thus provides the most realistic measure of the heat being drawn from the battery cell. It also provides for an accurate representation of the battery cells operating temperature which is another key component of the cooling system. The uniformity of the temperature was thus linked to this by defining it as the standard deviation of the temperature ($T_{\sigma}$) on this same plane. In this way a representative measure of the temperature of the battery cell, as well as the uniformity of that temperature within the battery cell was established which can relate the design goals to the performance of the cooling plate. For the power consumed by the cooling system, given that there was only a cooling plate to work with, the pressure loss ($P_{\text{fluid}}$) in the coolant channel across the plate was chosen. This loss, combined with the properties of the coolant and the mass flow rate can be used to calculate the power required to pump coolant through the system, and thus can be related to the power required to run the cooling system which is removed from the available
power for the vehicle. Given that one of the models has differing dimensions for the inlet and outlet channel, the total pressure was chosen instead of approximating it using the static pressure. The total pressure is the sum of the dynamic pressure (related to the kinetic energy of the fluid) and the static pressure (the pressure at a point on a body moving with a fluid).

3.2.2 Mathematical Formulation

With the three objective functions defined, a mathematical problem statement was produced to better represent the objectives of the optimization in terms that can be applied to an optimization algorithm. The mathematical problem was defined and can be seen in Equation 3-2. In this way the objective functions are accurately combined with geometry design parameters to obtain a goal that can be applied to an optimization algorithm.

**Equation 3-2: Mathematical Optimization Problem Statement**

Minimize:

\[
\begin{aligned}
T_{\text{avg}} & (F\text{req}, A\text{mp}, W\text{idth}, S\text{pace}) \\
T_\sigma & (F\text{req}, A\text{mp}, W\text{idth}, S\text{pace}) \\
P_{\text{fluid}} & (F\text{req}, A\text{mp}, W\text{idth}, S\text{pace})
\end{aligned}
\]

subject to:

\[
\begin{aligned}
F\text{req} &= f_n(x^{i,j}) \\
A\text{mp} &= f_n(y^{i,j}) \\
W\text{idth} &= f_n(y^{i,j})
\end{aligned}
\]

subject to:

\[
\begin{aligned}
\text{Constraints}_x^{i+1} - \text{Constraints}_x^i \geq \delta_x \\
\text{Constraints}_{n\text{boundary}}^j - \text{Constraints}_{n}^j \geq \delta_o \\
\text{Constraints}_n^j - \text{Constraints}_n^0 \geq \delta_o \\
\text{Constraints}_y^{j+1} - \text{Constraints}_y^j \geq \delta_y
\end{aligned}
\]

where:

\[
\begin{aligned}
T_{\text{avg}} &= \text{Average Temperature on the heat generation plane} \\
T_\sigma &= \text{Standard deviation of temperature on the heat generation plane} \\
P_{\text{fluid}} &= \text{Pressure drop across the channels} \\
F\text{req} &= \text{The number of wavelengths of the channel} \\
A\text{mp} &= \text{The difference between the y coordinates of a peak and trough} \\
W\text{idth} &= \text{The channel width defined by the y or x coordinates of two sides of the channel} \\
\text{Constraints}_n^j &= \text{number of n coordinate}
\end{aligned}
\]
The constraints here are represented simply to help express the optimization in a mathematical form. A summary of the model construction and effect of design variables and constraints can be found in the Appendix.

Frequency in this regard refers to the number of turns of the channel in the system, which can also be thought of as the wave number. The amplitude refers to the height difference between the top of the peak of the channel and the bottom of a trough of the channel. The channel width is fixed for the primary model and the secondary model, however it is variable for the tertiary model. For the constraints i refers to the edge of the channel (for example i=3 would be the bottom edge of the 2\textsuperscript{nd} channel, while i=2 would refer to the top side of the 1\textsuperscript{st} channel) as constructed during modeling outlined in Figure 3-6.. The j value in the constraints refers to the design variable along that axis (for example j=1 refers to the first x or y variable, as defined in Figure 3-10 to Figure 3-12).

3.3 Model construction

3.3.1 Model Geometry

As discussed in section 3.1.7 the model chosen for optimization is a curved asymmetric multi-channel system. A representation of the model is shown in Figure 3-5 and identifies some of the boundary conditions of the system the model will be placed in.

3.3.2 Identification of design variables

Given this outlining design, the model had to be represented by design variables that could be combined to construct a parametric model capable of optimization. The key geometry design variables for optimization were identified as:
- Frequency of the wavelengths (Freq)
- Amplitude of each wavelength (Amp)
- Width of the channels (in the y-axis) (Width)

The frequency of the wavelengths would control the x-axis locations of each of the peaks and troughs of the curves allowing for compacting of the channels in areas that might need more cooling while allowing for regions to have more spaced out curves if less cooling is required. The amplitude of each wavelength would allow for a similar effect in increasing or decreasing fluid flow across a region however in this case it would be along the y-axis. The channel width would control the velocity of the fluid for various regions at a given mass flow rate giving another control over the rate of heat transfer. The channel width is fixed for the primary and secondary model, however it is variable for the tertiary model.

Through this geometry design variable deconstruction, a large number of design variables were produced. This however posed another problem: Given such a large number of design variables, optimization iterations would take an infeasible amount of time. Each design variable compounds upon the last, quickly multiplying the length of each optimization iteration. In this regard the number of design variables had to be reduced somewhat to allow for a manageable optimization time frame. Designing the model for a variable channel width was causing the largest increase in the number of design variables. Allowing for a variable channel width at each point doubled the number of design variables and so by reducing this design feature to one design variable, by making it a uniform channel width across the plate, the computational time of each optimization was quickly brought into a feasible region while sacrificing the least amount of design freedom.
3.3.3 Fixed design parameters

Having identified the key geometry design variables and having limited the number of design variables to a manageable number the next step was to identify boundary conditions and fixed parameters relevant to the geometry of the model. The design of the model was similar to the models constructed for the design space exploration. The thickness of the cooling plate would be 1 mm, however using a symmetry boundary condition on the x-y plane at the center of the plate in the z-axis the model could be constructed with a 0.5 mm thickness. The channel width would once again be ¼ of the cooling plate thickness so in the model it is set at 0.375 mm. The models overall size and shape would also be the same as the design exploration models which
was 200 mm along the x-axis, and 160 mm along the y-axis giving a rectangular plate shape.

These values can be found summarized in Table 3-5.

**Table 3-5: Fixed Design Parameters for curved asymmetric curved channel model**

<table>
<thead>
<tr>
<th>Fixed Design Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Height (Along y-axis)</td>
<td>160 mm</td>
</tr>
<tr>
<td>Width (Along x-axis)</td>
<td>200 mm</td>
</tr>
<tr>
<td>Depth (Along z-axis)</td>
<td>0.50 mm</td>
</tr>
<tr>
<td>Channel depth (along z-axis)</td>
<td>0.375 mm</td>
</tr>
</tbody>
</table>

3.3.4 Modeling process

The preprocessor program GAMBIT was used to construct the model and mesh geometry. Journal files were used to run GAMBIT in batch mode to parametrically construct the models based on the design parameters. Figure 3-6 and Figure 3-7 illustrate the method by which the model was constructed.

Figure 3-6 outlines the method for constructing the faces that the volumes will be based upon from the parameters provided by the optimization. The design parameters provided are used to construct vertices outlining the shape of the plate, and the channels within it. This template is then duplicated for the three levels required to produce both solid and fluid regions. Straight edges are then used to connect the outer regions of the plate while curved line constructions called NURBS (Non-Uniform Rational Basis Spline) are used to connect the vertices for the curved channels. This edge creation is then duplicated for all three layers followed by creating straight edges between layers to connect everything and provide all the required structure to create faces. Faces are first created for the in plane areas, which are again duplicated for the three layers, this is followed by creating faces between the layers to finish up everything required for constructing volumes.
Figure 3-6: Modeling process for geometry creation for optimization models showing face construction.
Figure 3-7: Modeling process for geometry showing volume construction and meshing.

Figure 3-7 shows the extra faces created to construct the fluid region. Through this method the mesh could be disconnected allowing for a finer mesh within the fluid region that would not cause inflation in the solid region mesh due to mesh matching. This disconnect will be taken care of in the CFD program by creating a mesh interface between the fluid and solid faces. The solid volumes are created first by stitching together faces, followed by the fluid volumes. Groups are create for the inlet, outlet, heat generation, symmetry, fluid interface and solid interface faces in order to more readily and automatically identify these objects in the CFD program. With all the volumes created a mesh is applied to the solid and fluid in plane faces on Layer 2 and then sweep meshed through the volumes in both directions to provide a comprehensive mesh for the total system.

Using this method and with some random chosen initial design parameters a test model for the asymmetric curved channel design was constructed and can be seen in Figure 3-8 with its design variables shown.
3.3.5 Boundary Conditions

The cooling plate was constructed out of aluminum and the material properties used in the CFD can be seen in Table 3-6.

<table>
<thead>
<tr>
<th>Material Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Material</td>
<td>Aluminum</td>
</tr>
<tr>
<td>Density</td>
<td>2719 kg/m³</td>
</tr>
<tr>
<td>Thermal Conductivity</td>
<td>202.4 W/m·K</td>
</tr>
<tr>
<td>Specific Heat Capacity</td>
<td>871 J/kg·K</td>
</tr>
</tbody>
</table>

A 50-50 water ethylene-glycol mix was used as the coolant and the material properties for the CFD solver were determined from a commercial vendor of the product [65] and are the same as those used in previous work [64] which can be found summarized in Table 3-7. Some of the values are dependent on the temperature of the coolant, and given that the coolant has the
potential to heat up a noticeable amount depending on the performance of the cooling plates these values were taken into account by applying temperature dependent material properties to the CFD analysis.

**Table 3-7: Material Properties for 50-50 water-ethylene glycol coolant**

<table>
<thead>
<tr>
<th>Material Property</th>
<th>Value</th>
<th>Model Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density</td>
<td>1065 kg/m³</td>
<td>Constant</td>
</tr>
<tr>
<td>Thermal Conductivity</td>
<td>0.42 W/m·K</td>
<td>Constant</td>
</tr>
<tr>
<td>Viscosity</td>
<td>$0.0069 \times \left(\frac{T}{273.15}\right)^{-8.3}$ kg/m·s</td>
<td>Power</td>
</tr>
<tr>
<td>Specific Heat Capacity</td>
<td>$2574.7 + 3.0655 \times T$ J/kg·K</td>
<td>Polynomial</td>
</tr>
</tbody>
</table>

The boundary conditions are a defining characteristic of the optimization. The attempt for this study was to ensure that boundary conditions were realistic enough to provide optimizations that are applicable to current industry and technology, while allowing for comprehensive analysis and optimization. The boundary conditions chosen for this study can be found summarized in Table 3-8.

**Table 3-8: Boundary conditions for CFD analysis**

<table>
<thead>
<tr>
<th>Boundary Condition</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass Flow Rate at Inlet</td>
<td>0.001 kg/s</td>
</tr>
<tr>
<td>Coolant Temperature at Inlet</td>
<td>300K</td>
</tr>
<tr>
<td>Pressure at Outlet</td>
<td>0 Pa</td>
</tr>
<tr>
<td>Heat Flux on Heat Generation Plane</td>
<td>500 W/m²</td>
</tr>
</tbody>
</table>

The heat flux on the heat generation plane was the same as the heat flux used in the design exploration process and was determined from data provided by General Motors Canada.
Coolant inlet temperature was taken to be 300K while the mass flow rate was the same as that used in the design exploration as determined by technical data provided by General Motors Canada.

3.3.6 Secondary Model

As discussed in the background section, current research in cooling plate optimization has worked primarily with straight channel systems such as Jarrett and Kim [64]. Most current research deals with straight channels or channels with right angle corners. In this regard, a good way to help compare the results of this curved channel optimization would be to compare it to a similar 90 degree corner system. However since most current research also deals with single inlet and single outlet systems as opposed to multi-inlet and outlet systems there is not much data to compare these results too. As such a secondary model was created to better analyze the strengths and weaknesses of curved channel optimization. This secondary model can be seen in Figure 3-9.

The model was created with emphasis on ensuring it had as many of the same constraints as the curved channel system. The channel width is constrained to be the same throughout the system, and the 90 degree bends are designed to simulate the curves of the curved channel system as closely as possible. With these constraints there should be a good similarity to allow for close comparison between this design and the curved channel design allowing for identification of strengths and weaknesses of both designs as well as ensuring that results can be easily compared without needing to account for any advantages in geometry. A representation of the model with its design variables is seen in Figure 3-10.

In this it can be seen that many of the design variables are set up to be similar to those of the curved channel model, and these design variables are also located on the same side of the channels to ensure as much of a comparative design as possible.
Figure 3-9: Secondary model of a straight channel system constrained similarly to the curved channel system (representation of the half model used in analysis with a symmetry plane as the top face)

Figure 3-10: Diagram showing design variable assignment for secondary model
3.3.7 Tertiary Model

While constructing the secondary model and designing the constraints to match those of the curved channel model it became very apparent that the straight channel model had potential to produce a more diverse optimization due to the freedom allowed by the structure of its design. A number of extra constraints were needed to ensure that the model acted in the same way as the curved channel design, and by removing these constraints a larger variety of designs could be possible. In order to test this hypothesis and to provide more comparison between the curved channel system and straight channel systems a tertiary model was constructed in the same format as the secondary model but with far fewer design constraints. This tertiary model can be found in Figure 3-11 and Figure 3-12 shows the way in which its design variables are structured to provide for more design freedom.

This model was created using the same basic structure as the secondary model however this included more design variables. These design variables would account for a longer computation time not only in requiring more iteration steps during sensitivity analysis, but also in terms of convergence time as with more design variables the constraints required to maintain geometric integrity get more complicated and impact the optimization more.

3.3.8 Initial Design Analysis

In order to better verify the optimization process a number of different initial designs were also run to determine if the optimization is approaching the same optimum. Four randomly generated initial designs were optimized for average temperature and the initial designs can be seen in Figure 3-13. The results of optimizing these four designs for one of the objective functions will be analyzed to determine if the optimization routine finds a variety of optimum
designs suggesting the optimization is sensitive to the initial design. However if similar designs are determined it will validate the optimum designs and the optimization algorithm.

Figure 3-11: Representation of tertiary model demonstrating the systems increased design freedom (representation of the half model used in analysis with a symmetry plane as the top face)

Figure 3-12: Diagram showing design variable assignment for tertiary design model
3.3.9 Meshing

Proper meshing of the model is important to ensure accurate results are obtained by the CFD and to ensure strong optimization behavior. Improper meshing can lead to a large amount of noise in the CFD both within a simulation and between optimization iterations. If the mesh differs too much between simulations the change in results can carry through to the sensitivity analysis of the optimization and cause a change in the behavior of the optimization. In this way meshing is key part of the design process.
The mesh constructed for these models is comprised of two main parts; the mesh for the solid region and the mesh for the fluid region. Figure 3-7 showed a brief representation of the mesh construction method for both regions. The faces of the second layer are meshed first to provide a template to sweep the mesh through the other volumes. The choice to mesh an in plane face rather than a face between layers was made as it would provide a better mesh when sweeping through the volumes, and it would allow for more control of the through thickness elements. In this way a more effective and refined mapped mesh could be made for the channels ensuring the best possible mesh to help with CFD convergence, and a coarser mesh could be used for the solid elements where less elements are required for simple solid heat transfer.

The face mesh for the solid elements in layer two was swept down to layer one to create the volume mesh for the lower level solids. The solid regions of the upper level were created by selectively sweeping those face meshes up to layer three leaving a gap for the fluid volumes. The face meshes for the fluid channels were then swept up to layer three creating the volume mesh for the fluid channel and finishing the meshing of the model.

The mesh density for the fluid regions was set at 1mm\(^2\) for the x-y plane, and the mesh density for the solid regions was set at 2.5mm\(^2\) for the x-y plane. Along the z-axis the mesh was broken into 21 intervals with 7 allocated to the smaller solid region on the bottom and 14 allocated to the larger solid/fluid region ensuring the most effective mesh density for the fluid areas. This resulted in approximately 150,000 elements for a small fluid channel system and approximately 300,000 elements for a large fluid channel system. The range of elements is due to the denser mesh in the fluid region that when expanded creates more elements than when the region is solid. The method for choosing mesh density will be covered in section 3.6 when discussing the optimization validation and mesh convergence test.
3.4 Evaluation method: CFD

3.4.1 CFD Solver Settings

The models were evaluated through the use of the CFD program FLUENT. Files were created to run FLUENT in batch mode allowing for automation of the optimization system. These files were programmed to set up the boundary conditions for the system using the parameters from Table 3-8 and Table 3-6. A symmetry condition was set on the top face in line with the fluid channels (layer 3 from Figure 3-6) and a wall condition on the bottom solid faces (layer 1 from Figure 3-6). The inlet and outlets for the channels were set up as such and other faces were set to be insulated walls to reflect the insulation used around the battery cells.

User defined material models were a key part of constructing the CFD analysis. The coolant was a user defined material model with properties shown in Table 3-7. In this way the coolants properties would react to the heating that occurs in the plate and give a more accurate simulation of the changing efficiency of the coolant through the channels.

FLUENT provides a number of settings to control the methods and equations that are used to solve the models provided. Many of these settings when used incorrectly cause a large amount of error in the system and often provide entirely wrong results. These settings can also affect the speed, accuracy and assumptions used in the solver equations and thus need to be set properly to ensure quick and accurate results for use in the optimization. The CFD solver settings used for this study can be seen in Table 3-9.

In the optimization system a key part of the process is the sensitivity analysis used to build design gradients that the optimization bases its design variable progression on. During optimization small perturbations are made in the design variables to determine the optimization gradient.
Table 3-9: CFD Solver settings for evaluations

<table>
<thead>
<tr>
<th>Solver Setting</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure-Velocity Coupling Scheme</td>
<td>SIMPLE</td>
</tr>
<tr>
<td>Gradient Spatial Discretization</td>
<td>Green-Gauss Node Based</td>
</tr>
<tr>
<td>Pressure Spatial Discretization</td>
<td>Second Order</td>
</tr>
<tr>
<td>Momentum Spatial Discretization</td>
<td>Third-Order MUSCL</td>
</tr>
<tr>
<td>Energy Spatial Discretization</td>
<td>Third-Order MUSCL</td>
</tr>
<tr>
<td>Pressure Under-Relaxation Factor</td>
<td>0.2</td>
</tr>
<tr>
<td>Density Under-Relaxation Factor</td>
<td>1.0</td>
</tr>
<tr>
<td>Body force Under-Relaxation Factor</td>
<td>1.0</td>
</tr>
<tr>
<td>Momentum Under-Relaxation Factor</td>
<td>0.6</td>
</tr>
<tr>
<td>Temperature Under-Relaxation Factor</td>
<td>1.0</td>
</tr>
</tbody>
</table>

During this process lower order methods cause a larger divergence in the results during these small perturbations. This can lead to inaccurate mapping of the gradient, which can significantly affect the optimization. Through use of higher order solvers, the divergence of results from small perturbations can be reduced and allow for a more accurate representation of the gradient. This will allow for more accurate and more efficient optimization.

3.4.2 Turbulence discussion

The second key part of the CFD analysis was to determine if turbulence would need to be accounted for in the simulations. The predictor for the onset of turbulence is the Reynolds number which is a dimensionless number that represents the ratio of inertial forces to viscous forces in a system using the equation \( \left( \frac{V}{\nu} \right) \); where, \( V \) is the flow velocity, \( D_H \) is the hydraulic diameter, and \( \nu \) is the viscosity of the coolant. In this regard the important variables in this study that will have an impact on the turbulence are the mass flow rate and the channel width. Martin,
J. et al. [66] also suggested that sharp corners have an impact which will be relevant to the secondary and tertiary models more so than the curved channel system.

For the simulations done for the three models analyzed in this study Table 3-10 shows the Reynolds number for a variety of channel widths that lie within the design variable bounds. The temperature was assumed to be 300 K which results in a kinematic viscosity of $2.96 \times 10^{-6} \text{ m}^2/\text{s}$.

### Table 3-10: Reynolds numbers for potential channel widths and velocities

<table>
<thead>
<tr>
<th>Channel Width (mm)</th>
<th>Mass flow rate of 0.0005kg/s</th>
<th>Mass flow rate of 0.001kg/s</th>
<th>Mass flow rate of 0.002kg/s</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>55.2</td>
<td>110.3</td>
<td>220.7</td>
</tr>
<tr>
<td>10</td>
<td>29.5</td>
<td>59.0</td>
<td>118.0</td>
</tr>
<tr>
<td>20</td>
<td>15.3</td>
<td>30.6</td>
<td>61.2</td>
</tr>
<tr>
<td>30</td>
<td>10.3</td>
<td>20.6</td>
<td>41.3</td>
</tr>
</tbody>
</table>

Studies such as [67] showed that for the general case of a straight rectangular duct the critical turbulence transition region lies within a Reynolds number range of 1900 to 2800. The data in Table 3-10 shows that these simulations lie well within the laminar range given for a standard rectangular duct, however [66] suggested that when 90 degree corners or sharp U-Bends are taken into consideration the Reynolds number indicating the transition region for turbulent flow can fall to as low as 381. While this value is closer to the data seen in Table 3-10 it is also for different channel geometries then the ones found in the three models used in this study. However between these two studies and given the data in Table 3-10 it seems reasonable to assume that for the purposes of this analysis the simulations and models used lie within the laminar region and thus a turbulence model is not needed.
3.5 Governing program: optimization routine

The optimization routine is run through a MATLAB control unit seen in Figure 3-14. This control unit runs the system from the initial design input through to the final optimized design output. Output files are created to help with reviewing the progress of the optimization along with screen outputs for the user to keep a more direct and time dependent eye on the progress of the optimization.

The system starts with a MATLAB command to run the optimization routine with a number of design variables such as the initial design shape and other boundary conditions and design parameters. This data is used to construct an initial geometry parameters file that will then be updated with new parameters as the optimization routine runs. The parameters file is used to construct a model and mesh through a GAMBIT journal file using the modeling method seen in Figure 3-6 and Figure 3-7 which is then exported as a mesh file for the CFD program to use. The CFD program FLUENT is then started and a FLUENT journal file is used to set up the CFD analysis including boundary conditions taken from the initial optimization routine call along with the model and mesh file imported from the GAMBIT output. The CFD program runs until its convergence criteria are met at which point the outputs for the objective functions are exported to text files. These results are analyzed to calculate the objective function itself and as the loop repeats, these objective function values are monitored to check for convergence.

The fmincon function runs by conducting sensitivity analysis on the models design and determining the gradient for the optimization method based on this analysis. This requires a number of intermediate calculations between optimization iterations where design variables are perturbed slightly. These intermediate steps run on the same basis as the main optimization routine simply with the convergence check for the objective function skipped. The objective
function convergence is only checked for the main optimization iterations to ensure that design variables that no longer affect the objective function do not prematurely end the optimization by showing convergence.

Constraints were built into the MATLAB function to ensure that the design variables in the optimization routine were within a set range of boundaries. These boundaries were required to ensure that:

1. The CFD program would be able to run the model without divergence occurring.
2. The integrity of the channels and the plate was maintained with regards to the fluid channels being separate from each other and the walls of the model

If the integrity of the channels was compromised the model generation would fail and there would be nothing to pass to the CFD solver which could not provide results for optimization.

Bounds were set on the design variables as the first part of the constraints to ensure that the channels stayed within the overall shape of the cooling plate. Equations were then created for design variables as the second part of the constraints to ensure that:

1. Cooling channels cannot cross.
2. A gap of a minimum size is kept between the cooling channels and between the cooling channels and the cooling plate walls at all points.
3. The curves of the cooling channels are kept at a minimum distance for each other.

The third requirement was to ensure that the curves were orientated well enough that the CFD model would reach convergence. It also combined with the second requirement to ensure that there was enough solid area in all places to create a dense enough mesh for the heat transfer calculations to take place.
Figure 3-14: Diagram of MATLAB control program for the optimization routine.
The intermediate steps were a challenge when dealing with optimizations of this nature as during intermediate steps there is no requirement for the optimization solver to adhere to the constraints set in place to ensure geometric integrity. When these constraints are exceeded and the program accidentally generates a model that is infeasible, the optimization routine is stopped by MATLAB. This required the constraints to be stricter in certain instances to ensure that optimization did not fail. The curved channel required these stricter constraints due to the way in which the design variables for the curves in the channels interacted which was much more interdependent then in the straight channel models. The bends in the curved channel were dependent on the location of three design variables which all needed to be at least certain distances from each other for CFD convergence. This caused problems more often during intermediate steps, and required more stringent constraints.

3.6 Optimization validation

Optimization relies heavily on ensuring accurate data input in order to effectively produce accurate gradients and thus progress through optimization to the ideal final optimized design. To this end validation of the models and optimization algorithm are integral to ensuring that data provided and produced during optimization is accurate. There are two major focuses for validation in this study. The first is validation of the model and its mesh ensuring that it is rigorous enough to provide accurate results while being efficient enough in terms of computational cost to stay within the bounds of this study. The second major focus is the CFD analysis and determining at what point the solution is considered converged so that the computational time is limited to only that required to obtain accurate results.
3.6.1 Mesh Convergence Test

The mesh is a key component of the optimization validation process. Inaccurate meshing can lead to significant differences between slight design changes which can cause the sensitivity analysis to be altered and thus the optimization to progress along the wrong gradient. Too coarse a mesh for a model and this potential increases more and more, while too fine a mesh will cost a large amount of computational time making the optimization inefficient and impractical. To this end the mesh needs to be fine enough to ensure reliability between slight changes in geometry while being as coarse as possible to reduce computational costs. This mesh limit is found by conducting a mesh convergence test which measures an objective function against mesh density to correctly identify the point at which an increase in the density of a mesh has minimal impact in the accuracy of the results. For this study the most accurate measure to base the mesh convergence test from was the pressure drop across the system. This objective function converged much slower than the other objective functions and thus would be the limiting value in determining mesh convergence. The parameters of the mesh discussed in \( \theta \) were slowly increased resulting in an increase in the number of elements. The mesh convergence test was run at the same conditions the models were run under during optimization and the results can be seen in Figure 3-15.

Convergence occurred at around 39050Pa however the number of elements required to obtain this value resulted in each analysis taking a few hours. When this is coupled with the number of iterations required to run an optimization the computational cost of optimizing would be too great. The choice is therefore to find the point at which the results are within 1% of the converged value and use the number of elements required to achieve this value. This provides for a far lower computational cost with only a very small loss in accuracy. In this case Figure 3-15
shows that this was achieved at around a mesh density of around 150,000 elements. This density relates to specific mesh parameters discussed in 0 and thus varies depending on the ratio of fluid mesh region to solid mesh region on the plate, however provided very accurate results while allowing for a computational cost within the scope of this study.

3.6.2 CFD Objective function convergence

The CFD analysis continues until a user defined point of convergence is reached at which point the results are output. There are numerous variables that can be used to measure a point of convergence however the most efficient variables to use to stop the simulation are the residuals. In order to correctly determine the point at which the residuals reach a level where the simulation has come to convergence a number of designs were run through CFD in order to track their average temperature, pressure drop, standard deviation of temperature and residuals as the CFD analysis ran. Figure 3-16 shows the residual results of one of these analyses displaying an average simulation. Figure 3-17 shows the objective functions tracked during this simulation.

The objective functions for an average analysis converge to at least a 0.001% variance within 50 to 100 iterations however within this region instability occasions still exists in the z-velocity residuals. For this reason this study extended the convergence point to 150 iterations and Table 3-11 shows the values for the residuals at this point. Note that these are iterations for CFD analysis, as opposed to design iterations in optimization. This means that 100 – 150 CFD iterations will produce one CFD analysis result, which is equivalent to a single function evaluation from the view point of design optimization.
Figure 3-15: Mesh convergence test plotting pressure drop for curved channel model against mesh density

Table 3-11: Residual values taken at 150 iterations

<table>
<thead>
<tr>
<th>Residual Tracked</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Continuity</td>
<td>8.4434e-05</td>
</tr>
<tr>
<td>X-Velocity</td>
<td>1.6844e-05</td>
</tr>
<tr>
<td>Y-Velocity</td>
<td>9.8956e-06</td>
</tr>
<tr>
<td>Z-Velocity</td>
<td>1.1733e-07</td>
</tr>
<tr>
<td>Energy</td>
<td>1.4690e-11</td>
</tr>
</tbody>
</table>
Looking at the residual plot in Figure 3-16 many of the residuals are candidates for convergence criteria. The exclusion case could be made for the z-velocity which sees more instability than other values and thus may dip low too early causing premature termination. The energy value for this sample analysis is a steady measure however the magnitude of its residual occasionally varies between different geometries with varying channel sizes. In this regard the x-velocity and y-velocity residuals were chosen to act as a convergence criteria. At 150 iterations they have a value of $1.68 \times 10^{-5}$ for the x-velocity and $9.895 \times 10^{-6}$ for the y velocity. Including a factor of safety to ensure convergence given small variances in convergence between designs, the criteria for convergence was set at the x-velocity and y-velocity residuals reaching a point of $5.0 \times 10^{-6}$ which in this sample case results in a convergence at 212 iterations.
Figure 3-17: Objective function tracking during sample CFD analysis for residuals.
Chapter 4

Results

4.1 Optimization analysis

The objective functions were tracked during optimization to determine if local minimum optimal designs were occurring during some optimizations and to determine how well the optimization algorithms worked over a variety of different designs and models. The raw results from the objective function tracking can be seen in Figure 4-1.

These results show three distinct optimization paths which correspond with the three types of optimization. This is to be expected as the three objective functions have differing limits on the amount of improvement upon the initial design that is possible. In this regard it is hard to compare the results. In order to better compare the optimization between objective function types the results were therefore normalized in regard to their final performance increase which can be seen in Figure 4-2.

This graph provides a much better look at the progress of the optimization. These results show that for most optimizations the majority of the improvements occur in the first 20 iterations. The optimizations for standard deviation of temperature however seem to go against this trend improving in a more sequential step process over the course of their optimization. This suggests that there are large gradients associated with the majority of the optimizations which allow for significant initial improvements however the standard deviation optimizations start with much more gradual gradients meaning the improvements are approached in a much more gradual way.
4.2 Reynolds number analysis

As discussed in section 3.4.2 the effects of turbulence are closely tied to the Reynolds number and in the case of this study the laminar model was chosen as the Reynolds numbers looked to be well below the potential threshold for transition to turbulence. In order to validate this assumption, plots of the Reynolds number through a sample of the optimized designs were created to ensure that the simulations were within initial empirical calculations. The Reynolds number for the standard deviation of temperature optimized design using the secondary model can be seen in Figure 4-3, while the average temperature optimized design can be seen in Figure 4-4 and the pressure drop optimized design in Figure 4-5.
The highest seen Reynolds number in these sample designs is 44.2, which is almost an order of magnitude lower than the potential transitional region for u-bends suggested by [66]. This is also in the smallest potential channel size, with the larger channel sizes showing maximums of 19 and 16.5 which further helps to support the use of a laminar model over a turbulent one.

Using the optimization algorithm the three models; primary curved channel design, secondary straight channel with curved constraints design and tertiary straight channel with lax constraints, were run and optimized from a randomly selected initial design to see how the
optimization system worked and to try to identify some key geometry design features that contribute to the performance of various objective functions.

![Figure 4-3: Reynolds number plot for secondary model standard deviation of temperature design](image)

4.3 Three model results

A number of identifying geometry features can be seen between the various designs that show that key geometry features are prevalent to each design with some being unique to that type of optimization while others are somewhat shared between two designs identifying a level of commonality between these objective functions. Aluminum was used as the material for these plates in this simulation; however other materials with different thermal conductivities could affect the optimizations. These geometric features therefore could change given alternative materials.
Figure 4.4: Reynolds number plot for secondary model average temperature design

Figure 4.5: Reynolds number plot for secondary model pressure drop design
While the qualitative results of Figure 4-6, Figure 4-7 and Figure 4-8 are helpful at identifying geometry features, without numerical results it is hard to process how the features are affecting the performance of the cooling plates and to what extent they help or hinder the objective function. The numerical data for each of the optimizations is therefore given in the following tables. Table 4-1 shows the numerical results for all three models when optimized using the average temperature of the heat-generation plane as the objective function. Table 4-2 shows the numerical results for all three models when optimized using the standard deviation of temperature for the heat-generation plane as the objective function. Table 4-3 shows the numerical results for all three models when optimized using the pressure drop from the inlet to the outlet as the objective function.

**Table 4-1: Numerical results for all three models optimizing for average temperature**

<table>
<thead>
<tr>
<th>Model</th>
<th>$P_{\text{fluid}}$ (Pa)</th>
<th>$T_{\text{avg}}$ (K)</th>
<th>$T_\sigma$ (K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Primary Model</td>
<td>916.80</td>
<td>304.07</td>
<td>1.9788</td>
</tr>
<tr>
<td>Secondary Model</td>
<td>1098.99</td>
<td>303.50</td>
<td>1.5983</td>
</tr>
<tr>
<td>Tertiary Model</td>
<td>1328.10</td>
<td>303.74</td>
<td>1.6378</td>
</tr>
</tbody>
</table>

**Table 4-2: Numerical results for all three models optimizing for standard deviation of temperature**

<table>
<thead>
<tr>
<th>Model</th>
<th>$P_{\text{fluid}}$ (Pa)</th>
<th>$T_{\text{avg}}$ (K)</th>
<th>$T_\sigma$ (K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Primary Model</td>
<td>1429.63</td>
<td>304.93</td>
<td>1.9501</td>
</tr>
<tr>
<td>Secondary Model</td>
<td>4706.49</td>
<td>305.46</td>
<td>1.2215</td>
</tr>
<tr>
<td>Tertiary Model</td>
<td>3162.65</td>
<td>304.79</td>
<td>1.1711</td>
</tr>
</tbody>
</table>
Figure 4-6: Results for optimization and CFD analysis focusing on Average Temperature Optimization
Figure 4-7: Results for optimization and CFD analysis focusing on Standard Deviation of Temperature Optimization
Figure 4-8: Results for optimization and CFD analysis focusing on Pressure Optimization
Table 4-3: Numerical results for all three models optimizing for pressure drop

<table>
<thead>
<tr>
<th>Model</th>
<th>$P_{\text{fluid}}$ (Pa)</th>
<th>$T_{\text{avg}}$ (K)</th>
<th>$T_{\sigma}$ (K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Primary Model</td>
<td>864.01</td>
<td>304.53</td>
<td>2.0897</td>
</tr>
<tr>
<td>Secondary Model</td>
<td>587.27</td>
<td>303.46</td>
<td>1.9597</td>
</tr>
<tr>
<td>Tertiary Model</td>
<td>589.25</td>
<td>303.46</td>
<td>1.8032</td>
</tr>
</tbody>
</table>

This data is helpful at identifying the results at a qualitative level, but in this form it is hard to properly compare between designs, as well as between the curved channel model and the straight channel models as they started with different initial designs. In this regard an analysis of the initial designs is needed to provide a baseline with which to compare the relative improvements the optimizations had on the models.

4.4 Comparison to initial designs

The initial models used for the secondary and tertiary optimizations were the same to better help compare between the two models and their relative performances; this model was an attempt to duplicate the random initial model used for the primary model so that the curved channel could also be more accurately compared to the straight channel models. Figure 4-9 shows the initial models used for the optimizations and also the results of these initial models run through the CFD solver. The data for these models is presented in numerical form in Table 4-4. The performance increase between the optimized designs and the initial designs can be taken as a numerical difference however a simpler method to present the data in would be a measure of its performance increase. In this regard Table 4-5 shows the improvement of the optimized designs from the initial designs as a percent of performance increase. The baseline values were set at zero for the pressure and standard deviation of temperature, but since the coolant is at 300 K the
average temperature of the plates are normalized to this value. The values are calculated using Equation 4-1.

**Equation 4-1: Calculation of optimization improvement values**

\[
\%\text{ Improvement} = \left(1 - \frac{\text{Optimized Result} - \text{Baseline Value}}{\text{Initial Design Result} - \text{Baseline Value}}\right) \times 100
\]

**Table 4-4: Results of initial designs for all three models (secondary and tertiary initial models are the same)**

<table>
<thead>
<tr>
<th>Initial Design</th>
<th>( P_{\text{fluid}} ) (Pa)</th>
<th>( T_{\text{avg}} ) (K)</th>
<th>( T_{\sigma} ) (K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Primary Model</td>
<td>4597.27</td>
<td>307.111</td>
<td>2.591</td>
</tr>
<tr>
<td>Secondary Model</td>
<td>5864.41</td>
<td>305.9275</td>
<td>1.6494</td>
</tr>
<tr>
<td>Tertiary Model</td>
<td>5864.41</td>
<td>305.9275</td>
<td>1.6494</td>
</tr>
</tbody>
</table>

**Table 4-5: Performance increase results for optimized plates compared to initial designs for all three models**

<table>
<thead>
<tr>
<th>Model</th>
<th>Optimized for ( P_{\text{fluid}} )</th>
<th>Optimized for ( T_{\text{avg}} )</th>
<th>Optimized for ( T_{\sigma} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Primary Model</td>
<td>81.21%</td>
<td>42.78%</td>
<td>24.76%</td>
</tr>
<tr>
<td>Secondary Model</td>
<td>89.98%</td>
<td>40.92%</td>
<td>25.94%</td>
</tr>
<tr>
<td>Tertiary Model</td>
<td>89.95%</td>
<td>36.97%</td>
<td>28.99%</td>
</tr>
</tbody>
</table>

**4.5 Performance of optimum designs**

A basic comparison of the performance of the designs can be achieved from a simple analysis of the results shown in Table 4-1 to Table 4-3. The primary model performed the weakest in all optimizations. This seems to be due to the complexities of the constraints required to ensure the primary design provides a valid design, it from achieving its potential.
Figure 4-9: Results of initial designs for all three models (secondary and tertiary initial models are the same)
The complexity of these constraints leads the optimization to get stuck within local minima areas and thus stops.

This limitation of constraints leads to the primary model not achieving its largest possible channel width within the design domain. Of note however is the fact that the primary design performs significantly better in terms of the pressure loss when optimizing for other objective functions meaning that this design may be more desirable when accounting for multiple factors in a design.

4.5.1 Primary Model compared to Secondary Model

The primary model is most closely related to the secondary model. As the secondary model is designed from the primary model basis, they share many common features. Both designs have set channel widths through the entire plate, as well as having curves located in the same positions along the x-axis. The constraints placed upon these designs for optimization are also similar in order to simulate similar design changes during optimization. These designs however differ in the primary area of corner shape. The secondary model has sharp corners, and sharp bends and in this way distinguishes itself. These similarities and differences mean that comparison of these two designs will give insight into geometric features of similar designs between straight and curved channels.

The secondary model optimized designs all performed better when compared to the primary model. The objective functions for each design were lower and in some cases by a large margin. At face value this would suggest that straight channels are more desirable then curved channels of similar structure but there are some important limitations to note. Of key importance is the fact that while the secondary model performs better in terms of the objective function for average temperature and standard deviation of temperature, the pressure drop in both these cases
is higher than the primary model. This shows that given a single goal the straight channel system can provide more efficient designs compared to the curved channel system, however if multiple aspects of the design need to be controlled there is a noticeable loss in secondary objectives for the secondary model.

### 4.5.2 Primary Model compared to Tertiary Model

The primary model is also fundamentally linked to the tertiary model. The secondary model is based off the primary model, and the tertiary model is simply a less constrained form of this. In this way the basic layout of the plate is the same, however there are a number of features that differentiate these two designs. The tertiary model again is straight instead of curved, and the relaxing of the constraints allow the tertiary model to change channel widths at any section, allowing for more drastic changes in amplitude and wavelength spacing. This increase in complexity however comes at a cost. The design space with this increased freedom is much more complex and can lead the optimization to get stuck in local minima’s and design constraints when attempting to navigate such a complex space.

The tertiary model also performed more efficiently in all areas when only looking at one design variable however it too shares the problem that the secondary model has. Neither of the straight channel systems perform well in the pressure loss category when optimized for average temperature or standard deviation of temperature. The tertiary model has less of a significant performance loss when looking at the standard deviation optimized design, but more of a performance loss when looking at the average temperature optimized design. This is most likely because in the standard deviation of temperature optimized design the reduction in the constant channel width allows for larger channel widths as the channels expand for the tertiary design. The average temperature design however seems to have encountered the suggested problem with the
complexity of the design space. While optimizing it seems that the complexity of the design space would not allow for larger channels in certain sections leading to a less efficient design than the secondary model.

4.6 Performance of optimization

For the average temperature optimizations seen in Figure 4-6, with numerical values in Table 4-1 and performance increases in the second column of Table 4-5, there are some key characteristics to note. The average temperature optimized designs seem to be trying to balance out the goal of increasing the fluid surface area as much as possible against the goal of trying to increase the velocity of the fluid using narrow channels. There is evidence of this in the tertiary model where there is a combination of narrower and thicker channels throughout the system. Increasing the channel widths would increase the surface area contact between the coolant and the plate, increasing the heat transfer rate, while reducing the channel width increase the fluid velocity thus increasing the heat transfer rate. These objectives however are conflicting meaning there must be a balance between the two that provides the optimal amount of heat transfer out of the system. The average temperature models however all do share a commonality in the amplitude of the channels. All three models attempt to reach the largest amplitude they can, which can be explained by an optimization goal of increasing the path length of the channel as much as possible. This would make sense as it would be a way to increase the surface area contact between the coolant and the plate such as the design aspect of increasing the channel width was attempting, while not having to compete with a reduction in channel size for velocity. This supports the suggestion that velocity is a competing design aspect.

For the standard deviation of temperature optimizations seen in Figure 4-7, with numerical results in Table 4-2 and performance increases in the third column of Table 4-5, there
are some very different characteristics. There is a noticeable increase in the amplitude of the channels as they progress from the inlet to the outlet. This is more pronounced in the secondary and tertiary models as compared to the curved channel model which may be due to the way in which the fluid flows through the systems, and the methods needed to construct the curved channel system. This would suggest that the optimization is trying to balance a variable which changes as the channel progresses which leads to the conclusion that it is trying to account for the fluid temperature. At the inlet the fluid temperature is 300K however as it progresses through the system it absorbs heat increasing its temperature along the path length. This decreases the thermal gradient between the coolant and the wall resulting in a lower heat transfer rate. The optimization seems to be attempting to compensate for this by increasing the path length of the fluid resulting in more surface area for heat transfer which balances the lower thermal gradient. The tertiary model also has a significant difference from the other models in this optimization, which identifies an objective function under which it has a key design advantage. The channel width for the tertiary model slowly increases across the length of the plate, resulting in an inlet that is smaller than the outlet. The advantage of this is seen both in its performance in Table 4-5 and the temperature plot in Figure 4-7. The variability in channel width allows for a more consistent control of the heat transfer rate as the coolant heats up, resulting in a more uniform region around the channel as evidenced by the larger green zone in the tertiary model compared to the primary and secondary models. Through this it can be seen that the optimization is trying to balance the deviation in the heat profile between the cold zone that is created by the coolant at the inlet and the hot zone created by heated coolant at the outlet. This suggests that there is a balance and connection between the inlet channel width and outlet channel width defined by the ratio of channel width increase across the plate.
The pressure optimizations seen in Figure 4-8, with numerical results in Table 4-3 and performance increases in the first column of Table 4-5, again show a very different set of geometry features. The primary feature seen here is the size of the channels in the secondary and tertiary models. The optimizations for these two models seem to be attempting to increase the channel size to its largest possible. This makes sense as the pressure would be much smaller in larger channels and with large open bends the losses would be smaller than those of small tight corners. In the tertiary model the optimization also seems to be attempting to reduce the sharpness of the corner by increasing the wall size in the center rise. This does not appear on the first and last walls; however this may be due to the overriding goal of increasing channel width in these smaller sections, as compared to the longer region in the center. This reduction in sharpness correlates very well with the primary model which has a slightly different structure than the two straight channel models. There seems to be a balance in this model between channel size and the increase in the depth of the curves that an increase in channel size would make. Since this model does not have sharp corners it is attempting to minimize the amplitude while increasing the period between curves, and balancing this against a competing design goal of increasing the channel width. The ideal channel for pressure loss would of course be a straight channel, however due to the constraints this is not possible. The structure of the primary model however suggests that the optimization is trying to get as close to this straight large channel as possible, since it does not have to account for the sharp corners of the straight channel models.

4.6.1 Primary Model compared to Secondary Model

In terms of optimization the primary design optimization does perform very well as compared to its initial design. The primary model optimization is slightly less effective than the secondary model optimization when optimizing for pressure loss, however it is almost the same
when optimizing for standard deviation of temperature and slightly better when optimizing for average temperature. This suggests that while the numerical results show the primary model is not achieving its true potential, the optimization results show that there are large gains to be made with optimization over the original design that could show the primary design coming out as more efficient given more effective and lax constraints.

4.6.2 Primary Model compared to Tertiary Model

In terms of optimization the primary design optimization is noticeably less efficient then the tertiary design optimization. The cause for this is obviously the more lax constraints that the tertiary design has over the primary design. While the primary design may show potential for improvement over the secondary design, in order to compete with the tertiary design it most likely would need both lax constraints to allow for more optimization, as well as more design freedom to allow for varying channel widths which would be very computationally expensive.

4.7 Initial design sensitivity analysis

In order to better determine if the optimization algorithm is improving the design towards a global optimum or if instead the optimization algorithm is finding local minima close to the initial design, an examination of a variety of initial designs was done to see the effect these designs would have on the optimized value. Four initial designs were constructed that varied as many design variables as possible and randomized them within their constraints. The initial designs can be seen in Figure 3-13. In this way the initial designs would be as random as possible while being within the feasible domain for optimization. These initial designs were then run through the optimization algorithm optimizing for average temperature of the heat generation plate to produce the results seen in Figure 4-10
These geometry results were then run through the CFD solver FLUENT in order to obtain the average temperature profiles for the heat generation plane seen in Figure 4-11. These results show similar temperature profiles to those of the optimized design in Figure 4-6 with a slowly rising temperature gradient across the plate.

The numerical results for these optimizations can be found in Table 4-6 and show values similar to the optimized design found in Table 4-1. These results were then compared to the same
initial design as the optimized design was and their performance improvements of this initial
design can be seen in Table 4-7.

Table 4-6: Numerical results from analysis of four randomized initial designs optimized for
average temperature

<table>
<thead>
<tr>
<th>Model</th>
<th>$T_{\text{avg}}$ (K)</th>
<th>$P_{\text{fluid}}$ (Pa)</th>
<th>$T_o$ (K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Initial Design Analysis 1</td>
<td>304.07</td>
<td>992.35</td>
<td>1.6786</td>
</tr>
<tr>
<td>Initial Design Analysis 2</td>
<td>304.09</td>
<td>980.46</td>
<td>1.8202</td>
</tr>
<tr>
<td>Initial Design Analysis 3</td>
<td>304.09</td>
<td>978.09</td>
<td>1.7608</td>
</tr>
<tr>
<td>Initial Design Analysis 4</td>
<td>304.08</td>
<td>993.35</td>
<td>1.8199</td>
</tr>
</tbody>
</table>

Table 4-7: Improvement of four initial designs over initial design for curved optimization
when optimizing for average temperature

<table>
<thead>
<tr>
<th>Initial Design</th>
<th>Improvement over standard initial design</th>
</tr>
</thead>
<tbody>
<tr>
<td>Initial Design Analysis 1</td>
<td>42.07%</td>
</tr>
<tr>
<td>Initial Design Analysis 2</td>
<td>42.53%</td>
</tr>
<tr>
<td>Initial Design Analysis 3</td>
<td>42.46%</td>
</tr>
<tr>
<td>Initial Design Analysis 4</td>
<td>42.69%</td>
</tr>
</tbody>
</table>

When looking at these randomized initial designs and the optimizations they have
produced it can be seen that there are similarities to the final optimized design such as how the
channel width has been increased to be as large as allowed by the constraints for the curved
channel designs. There are however a number of differences that distinguish these designs from
each other and the optimized design. This would suggest initial design dependent optimization if
not for the results that can be seen in Table 4-7. These optimizations achieved almost identical
performance increases to each other and to the final optimized design over the initial design. This
suggests that while the designs are different from each other geometrically, they are very similar in their performance.

![Figure 4-11: Average temperature profiles for initial design analysis when optimized for average temperature](image)

The conclusion from this is that within the design domain bounded by the constraints of the geometry, there are a number of local minima. This suggests that key geometry features such as channel width are of primary importance, and that once the design has optimized to within a certain geometry range the results become insensitive to some of the less impactful design variables. In order to test this, algorithms would have to be developed to allow the optimization to proceed with more lax constraints and this is outside the scope of this study.
4.8 Non-optimal performance

The optimized plates obtained in this study were for a determined set of boundary conditions, however when in commercial use the plates will undergo a variety of conditions which the plates may not be designed to handle. It is important to evaluate the models at these conditions to see some of the effects that a variety of boundary conditions have in order to properly identify the advantages or limitations of designs. The following figures show results of the models run at a range of heat flux levels, inlet temperatures and mass flow rates to evaluate the effect these boundary conditions have on the optimized plate’s performance.

4.8.1 Average Temperature Optimized Models

The boundary condition with the most impact on the average temperature of the plate is the inlet temperature of the coolant seen in Figure 4-14 which is to be expected; variations in the temperature gradient from the coolant with the quantity of coolant running through the system far outweighs the temperature gradient from the heat generation plane. In these results there also seems to be a linear relationship between the inlet temperature and the average temperature of the plate, a characteristic that the heat flux also seems to share seen in Figure 4-12. The heat flux seems to have the same amount of impact on the average temperature as the mass flow rate does, however the mass flow rate seems to be the only boundary condition with a non-linear relationship seen in Figure 4-13. Mass flow rates below 0.001kg/s seem to have a larger impact on the average temperature of the plate than mass flow rates over 0.001kg/s. Since the plates are optimized for this value, the data supports the theory that the velocity of the fluid is a key consideration during plate optimization and that mass flow rate is one of the most important boundary conditions when optimizing the geometry of a plate for the average temperature objective function. From this data coolant temperatures required to maintain the average
temperature of the plate at a specific value can be easily predicted given the linear relationships of heat flux and inlet temperature. This data also shows that the most reliable way to control the average temperature of the plate would be to keep the mass flow rate constant and vary the coolant temperature.

Figure 4-12: Average temperature optimized models at varied heat flux levels

### 4.8.2 Pressure Drop Optimized Models

Heat flux seemed to have a very small impact on the pressure drop in terms of the secondary and tertiary models while having an almost negligible impact on the pressure drop for the primary model as seen in Figure 4-15. These optimized models seem to have a linear relationship between mass flow rate and pressure drop as seen in Figure 4-16 while having a nonlinear relationship between coolant temperature and pressure drop as seen in Figure 4-17. This data would therefore suggest that in regards to controlling the pressure drop, the coolant temperature should be kept constant while the mass flow rate is varied.
Figure 4-13: Average temperature optimized models at varied mass flow rates

Figure 4-14: Average temperature optimized models at varied inlet temperatures
Figure 4-15: Pressure drop optimized models at varied heat flux levels

Figure 4-16: Pressure drop optimized models at varied mass flow rates
Figure 4-17: Pressure drop optimized models at varied inlet temperatures

This data conflicts with the average temperature optimized designs, showing that while the two objective functions may show many similarities in geometry features, the relationships between the boundary conditions and these features is quite different.

4.8.3 Standard Deviation of Temperature Optimized Models

With regards to the standard deviation of temperature, all the boundary conditions seemed to have a noticeable effect on the objective function. An interesting thing to note in this data is the results between the secondary and tertiary models. Throughout the data in figures Figure 4-18, Figure 4-19 and Figure 4-20 there is a noticeable variation between which model is more efficient. This data may show how the geometry features in these two different designs are very dependent on the boundary conditions, highlighting the sensitivity of the standard deviation of temperature optimizations. The heat flux again seems to have a linear relationship, this time...
with the standard deviation of the plate seen in Figure 4-18. The nonlinear relationship between mass flow rate and average temperature seen in Figure 4-13 also seems to apply to the standard deviation of temperature seen in Figure 4-19 further highlighting the suggestion that constant mass flow rate should be a key design choice when constructing cooling systems using these optimized plates. Inlet temperature seems to have the least impact on the standard deviation of temperature as seen in Figure 4-20. This is most likely due to the balance between inlet size and channel path length that has already been optimized in the designs, thus allowing for a more consistent and lower scaling in temperature with variations in coolant temperatures.

Figure 4-18: Standard deviation of temperature optimized models at varied heat flux levels
Figure 4-19: Standard deviation of temperature optimized models at varied mass flow rates

Figure 4-20: Standard deviation of temperature optimized models at varied inlet temperatures
Chapter 5

Summary and conclusion

5.1 Summary of results

The analysis done in this study was to optimize three models of liquid cooling plates for electric vehicle battery packs to provide improvements on the objective functions of average temperature of the heat generation face, standard deviation of temperature on the heat generation face and pressure loss through the cooling plates. The results are broken down into the following 6 stages of analysis used to study these three models.

5.1.1 Examine a number of design features for liquid cooling systems

Design space exploration was undertaken to examine the design domain of a liquid cooling plate and identify geometry features that could lead to increases in efficiency and performance. Six design geometries were analyzed using a variety of different types of geometry features in order to capture as much of the design space as possible. These designs were joined by a model designed to roughly represent a currently used cooling plate in electric vehicles. The cooling plates were constructed in GAMBIT and a mesh file was produced and passed to the CFD solver used for this study; FLUENT. This solver determined the average temperature of the heat generation plate, and the standard deviation of the heat generation plate when undergoing various mass flow rates of the coolant. The pressure drop across the plate was used to determine the power required to run the plate at that mass flow rate and the results of average temperature and standard deviation of temperature were analyzed against the power to generate graphs of performance. Three design geometries were determined to be the best however there were constraints to two of them. The first model had structural problems that would make it useless for
commercial applications and both the first and second models were too simplistic to allow for proper optimization of the design features. In this regard a model of multiple curved channels along a plate oriented in an asymmetrical pattern was determined to provide the most efficiency and best performance while allowing for further optimization.

5.1.2 Construct models for predicting the performance of cooling plate geometries

Three models based on the multiple curved channel design were to be created for optimization. The preprocessor GAMBIT was used to construct each model using a journal file reading from a separate parameters file to generate the models parametrically. This parametric nature allowed for a multitude of various designs based on the three models. The first model consisted of curved channels created using NURBS across multiple vertices while the second and third model had straight angular channels which used simple straight edges.

The models were meshed using variable mesh allowing for finer mesh in the fluid region with coarser mesh in the solid regions where only heat transfer was occurring. The meshed models were then passed to the CFD solver FLUENT. This solver used a variety of boundary conditions including a fixed heat flux on the heat generation wall representing a battery cell and a set temperature mass flow inlet to represent cooled coolant entering the system. FLUENT was used to simulate the cooling plate geometries and record the average temperature of the heat generation plate, the standard deviation of temperature of the heat generation plane and the pressure loss across the channels for each of the models.

The density and accuracy of the model mesh was verified using a mesh convergence test while the accuracy of the CFD results was verified using analysis of the objective function and residual convergence.
5.1.3 Develop an algorithmic structure for optimizing cooling plate geometries

With a program capable of constructing and evaluating a variety of design geometries based on three multiple curved channel designs using the preprocessor GAMBIT and the CFD solver FLUENT, there was now a foundation for constructing an optimization algorithm system. GAMBIT and FLUENT would be used to construct and evaluate each individual intermediate and iteration step of the optimization system and these functions would reside as part of a larger program developed in MATLAB using the fmincon function to perform the optimization itself. This program would take an initial design and through iterative gradient based solving would optimize the design and output a final design.

The accuracy of this program was verified through optimization of multiple initial designs in order to determine if they converged to the same values or if the final design was dependent on the initial design indicating that local optimized designs were being achieved and not globally optimum designs.

5.1.4 Validate that optimization can improve design geometries

With the construction and evaluation system for models set up, and an algorithmic optimization process developed the three types of models were optimized for three objective functions; average temperature of the heat generation plane, standard deviation of temperature of the heat generation plate and pressure drop across the plate. Random initial designs were generated for each of the three models and optimized using the MATLAB system to achieve final designs. Optimizing for each of these objective functions resulted in notable performance increases over the initial models.

The pressure drop across the models was reduced to between 10 and 20% of its value in the initial designs when optimized for pressure drop. The average temperature of the heat
generation plate was reduced to between 60 and 75% of its value in the initial designs when optimized for average temperature. The standard deviation of temperature was reduced to between 25 and 30% of its value in the initial designs when optimized for standard deviation of temperature.

5.1.5 Determine the interaction of design features with the performance of the cooling plates

A variety of design features were noted for each of the three objective functions that seem to have specific effects on the performance of the plates. For the average temperature it was noted that there was a balance between the velocity of the coolant obtained by decreasing the channel size, and the heat transfer rate obtained by increasing the channel size. Analyzing the differences between average temperature and pressure drop it seems as if there are two potential global optimums separated by a region of decreased performance. The first of these optimums balances the velocity of the fluid with the heat transfer rate, while the second of these optimums increases the size of the channel to the point where it overtakes the reduction in performance from a decrease in velocity.

For the pressure loss more simple design features were noted that had expected results on the models performance. For the straight sided models the optimization attempted to increase the size of the channels as much as possible to decrease the pressure losses associated with coolant velocity and corners and to increase the area in which the fluid is traveling through as much as possible. For the curved channel system this was slightly different, in this model the optimization attempted to reduce the amplitude of the wavelengths as much as possible and to increase the gap between wavelengths to better streamline the flow. This was most likely due to constraints placed on the model that restricted the width of the channel which therefore didn’t allow the full
benefits of channel size to be within the feasible region and thus the global optimum design considerations were reducing pressure losses from turns.

For the standard deviation of temperature the models were very different. The models in this case were defined by a balance of the heat transfer effects of fluid heating up as it traveled across the plate, with the heat transfer effects of increasing the surface area it travelled across. As the coolant travelled across the plates it heated up and thus reduced the heat transfer rate, this was countered by taking longer path lengths as the coolant travelled thereby allowing time for equal amounts of heat to be absorbed in different areas. The tertiary model also helped in this by increasing the channel width as the coolant travelled to increase the surface area it interacted with and thus increase the heat transfer rate that way.

Design features were compared between the primary and secondary design, and primary and tertiary design to compare performance and determine the benefits and challenges associated with each design.

5.1.6 Examine the effects of the boundary conditions on cooling plate geometries

The effects of variation on the inlet temperature of the coolant, the mass flow rate of the coolant and the heat flux of the heat generation plane were analyzed by varying each boundary condition individually and analyzing the data for each of the three models and their optimized designs. The three models optimized for average temperature showed that the mass flow rate had a nonlinear effect on the average temperature of the plate while the other two boundary conditions seemed to be linear. This suggested that the geometry of the plates was dependent on the mass flow rate of the coolant, but was independent of the heat flux of the heat generation plane and the temperature of the coolant.
The three models optimized for pressure drop showed a nonlinear relationship between the inlet coolant temperature and the pressure drop across the system, while the mass flow rate and the heat flux had a linear or minimal impact on the pressure drop. This suggests that in order to accurately control the pressure drop when optimized for the pressure drop, it is best to hold the inlet coolant temperature constant and vary the mass flow rate.

The three models optimized for standard deviation of temperature also showed a nonlinear relationship between the mass flow rate and the standard deviation of temperature of the heat generation plane. This is similar to the average temperature plate and shows that the optimization geometry for temperature is primarily dependent on the mass flow rate.

5.2 Limitations of work

5.2.1 Constraints on geometry

The optimization performed here is limited to a small section of the design space. As discussed in 3.1.1 there are a number of different design models that have the potential be the global optimum solution for the conditions used in this study. The three models studied defined by their design parameters explore only a small amount of this design space. The models were limited to two wavelengths and two channels, which while necessary to allow for optimization, limits the amount of the design space explored.

Optimization is in itself limited to analyzing a small part of the design space, and thus cannot of itself obtain a global optimum to the problem. Given more resources new studies would have to be conducted that could explore more design features and work their way towards the global optimum design. Previous work suggested features such as branching channels along with variable thickness channels have potential to increase cooling effects so further studies could examine and optimize for these. Features such as these however would require more
computational time or redesigns of the modeling process to evaluate. Studies such as these however could be combined to accurately evaluate the impact of design features and thus help to search for the global optimum design.

5.2.2 Constraints on optimization

The complexities of the geometries used in this study are fairly simple by technical standards but considering the number of design variables, for optimization they comprise a complex system. Many of the design variables in these systems are highly dependent on numerous other design variables leading to a very complex interaction between multiple elements of the systems. Due to these complexities the constraints required during optimization in order to ensure that the channel integrity was maintained for all models, and to ensure that analysis of the geometries came to convergence were complex.

Restrictions did not allow for significant channel sizes in the curved channel model, and required rather large angles in the bends without which the CFD would not converge. In the peaks and troughs of the wavelengths there were numerous design variables that compounded on each other often causing mesh or geometry failure in these areas. This required strong constraints in these areas which limited the shape of the bends. These constraints combined to limit the design space and freedom for the curved model. The straight channel models had fewer limitations due to the simpler interaction around curves however they too had limits between variables that caused problems in objective function convergence occasionally.

The gradient based optimization solver often ignores the constraints placed on the design variables during intermediate iterations where it performs sensitivity analysis to calculate the gradient. This causes problems when it generates a design that cannot be constructed or meshed. Since this design cannot be generated and run through the CFD solver, there is no way to provide
data for the gradient, and the solver provides no way to interpret this. This problem leads to either failure of the solver, or requires dummy data to be inserted which can affect the calculated gradients and the sensitivity analysis. In order to get around this problem the constraints had to be further restricted and the interval step reduced to ensure that the solver could reach the boundary conditions, but that steps during sensitivity analysis would small enough and within the added constraints to not cause failure of the model.

5.2.3 Limited boundary conditions

The boundary conditions used in this study are representative of only a snapshot of a relatively intensive battery state. During the drive cycle there would be significant variation of the heat generated by the battery and thus the heat flux to the coolant. The drive cycle would also most likely have variability in the temperature of the coolant and its mass flow rate depending on the requirements needed to keep the batteries at their ideal operating temperature. Since this study only looked at a snapshot it cannot account for the variability in these boundary conditions. Optimization of all these conditions would be an extremely intensive computational cost and thus are outside the scope of this study.

The modeling of the interaction between the cooling plate and the battery cell was also idealized. The battery cells and cooling plates are combined and then pressed into a pack however there is no indication that they are adhered together in this pack. This means that the idealization that the battery cell is in contact at all points with the cooling plate may be inaccurate, especially depending on the method in which the cooling plate is constructed. More industrial data would need to be provided to accurately represent this interaction and provide more realistic simulations.
5.3 Future Work

The most effective aspect to expand upon this work would be to create an optimization solver that would allow gradient based optimization to be completed without leaving the constraints of the design during the sensitivity analysis. With this method available the constraints of models could be greatly expanded for complex systems such as the curved channel system allowing for a more in depth analysis of the local design space. This would also allow the optimization to focus on other areas of the design such as more channels, and more wavelengths that could further enhance this exploration.

Outside of this, the best method would be to continue with 90 degree angle models and examine the effect of more wavelengths and channels through optimization of a few key systems within the local design space. In this way although the local design space would not be fully explored, some inferences about the way in which these features affect the cooling plate could be constructed.

The material used in this analysis was aluminum. Other materials with different conductive properties could lead to changes in the efficiency and speed of distribution of heat through the plate. This means that some materials might need different channel geometries to account for this change. This could be another area to look at and determine the effect of thermal conductivity on optimization.

The objective functions analyzed here can be combined using a weighting system to optimize multiple objectives at the same time. This aspect was not performed in this study, but analysis of multiple objectives and the effect of different weighting factors could be used to better determine the effects on the geometry and the tradeoffs between objective functions.
Finally, in order to determine if this design is close to the global optimum, more of the global design space will need to be explored and optimized to generate data on where local optima lie and if any designs can increase their performance over the designs studied here.

5.4 Conclusions

In order for electric vehicles to compete with internal combustion engine vehicles for share of the transportation sector their performance must be increased to a comparable level. Increases in the efficiency and performance of the battery system can provide large gains in improving multiple aspects of the electric vehicle performance and thus is a key area for research and advancement. The most promising area of battery technology available for optimization is the cooling system in which cooling plates can be optimized for efficiency and design.

This study worked to optimize a cooling plate geometry based on a set of boundary conditions to improve the average temperature, standard deviation of temperature and pressure loss associated with the cooling plate. This was achieved through the use of numerical optimization using a MATLAB optimization algorithm and CFD solver FLUENT to evaluate the performance of designs.

This study used design space exploration to examine a variety of geometry features that could contribute to a more efficient cooling plate. From this design exploration one design feature was chosen and model for three cooling plates based on that feature were created. The primary cooling plate was a system with two channels constructed of two wavelengths of curved bends. The secondary and tertiary cooling plates were constructed using sharp 90 degree angles in the same approximate shape as the curved channel system.

While this study examined a few sections of the design space it is not possible to determine if the global optimum was achieved. Further exploration of models from other areas of
the design space would be needed to more accurately show if these models are close to the global optimum. The algorithms used to construct these models and optimization structures however, along with observations of the impact of certain boundary conditions and geometry features on the objective functions can be applied to future optimization and design as a basis for effective performance enhancing and to create optimization systems for other areas of the design space.

This study provides data on geometry features of multiple curved channels that can reduce the power consumption of plates which can increase the efficiency of a battery cooling system. It also provides data on features that can reduce the standard deviation of temperature allowing for a more uniform battery cell which increases its life cycle and features that reduce its average temperature so it can operate in its most efficient range. This data can be combined with other studies to improve each aspect of an electric vehicle and through research and dedication can be one step to creating a competitive alternative energy vehicle for the demanding market.
Bibliography


Appendix A
Reynolds Number

The first step to calculating the Reynolds number is to calculate the velocity of the fluid based on the mass flow rate and the channel width of the duct. This is achieved by using Equation A-3 incorporating values from Equation A-2 and Equation A-1. This gives us velocity results found in Table A-1.

**Equation A-1:** Density used for Reynolds number calculations

\[ \rho = 1065 \left( \frac{kg}{m^3} \right) \]

**Equation A-2:** Mass flow rate used for Reynolds number calculations

\[ \dot{m} = \text{mass flow rate} \left( \frac{kg}{s} \right) \]

**Equation A-3:** Equation used to calculate velocity for Reynolds number calculations

\[ V = \frac{\dot{m}}{\rho A} \left( \frac{m}{s} \right) \]

**Table A-1: Results of Velocity Calculations**

<table>
<thead>
<tr>
<th>Channel Width (m)</th>
<th>Mass Flow Rate (kg/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0.0005</td>
</tr>
<tr>
<td></td>
<td>0.001</td>
</tr>
<tr>
<td></td>
<td>0.002</td>
</tr>
<tr>
<td>0.005</td>
<td>0.1252 m/s</td>
</tr>
<tr>
<td></td>
<td>0.2504 m/s</td>
</tr>
<tr>
<td></td>
<td>0.5008 m/s</td>
</tr>
<tr>
<td>0.01</td>
<td>0.0626 m/s</td>
</tr>
<tr>
<td></td>
<td>0.1252 m/s</td>
</tr>
<tr>
<td></td>
<td>0.2504 m/s</td>
</tr>
<tr>
<td>0.02</td>
<td>0.0313 m/s</td>
</tr>
<tr>
<td></td>
<td>0.0626 m/s</td>
</tr>
<tr>
<td></td>
<td>0.1252 m/s</td>
</tr>
<tr>
<td>0.03</td>
<td>0.0209 m/s</td>
</tr>
<tr>
<td></td>
<td>0.0417 m/s</td>
</tr>
<tr>
<td></td>
<td>0.0835 m/s</td>
</tr>
</tbody>
</table>
To calculate the hydraulic diameter a few variables and equations need to be defined. These variables are outlined with Equation A-4, Equation A-5, Equation A-6 and Equation A-7. The variables a and b for these equations are shown represented in Figure A-1.

**Equation A-4: Channel width used for Reynolds number calculations**

\[ a = \text{Channel Width} \ (m) \]

**Equation A-5: Channel depth used for Reynolds number calculations**

\[ b = \text{Channel Depth} = 0.00075 \ (m) \]

**Equation A-6: Area used for Reynolds number calculations**

\[ A = a \times b \ (m^2) \]

**Equation A-7: Wetted perimeter used for Reynolds number calculations**

\[ \rho = 2a + 2b \ (m) \]

Using these variables and equations and substituting into Equation A-8 the hydraulic diameter can be calculated. Results for the hydraulic diameter along with the equations used to find it can be shown summarized in Table A-2.

**Equation A-8: Equation used for calculating hydraulic diameter for Reynolds number calculations**

\[ D_h = \frac{4A}{\rho} \ (m) \]

**Table A-2: Results of calculations for area, wetted perimeter and hydraulic diameter**

<table>
<thead>
<tr>
<th>Channel Width (m)</th>
<th>A (m)</th>
<th>( \rho ) (m)</th>
<th>( D_h ) (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.005</td>
<td>0.00000375</td>
<td>0.0115</td>
<td>0.001304348</td>
</tr>
<tr>
<td>0.01</td>
<td>0.00000750</td>
<td>0.0215</td>
<td>0.001395349</td>
</tr>
<tr>
<td>0.02</td>
<td>0.00001500</td>
<td>0.0415</td>
<td>0.001445783</td>
</tr>
<tr>
<td>0.03</td>
<td>0.00002250</td>
<td>0.0615</td>
<td>0.001463415</td>
</tr>
</tbody>
</table>
To calculate the Reynolds number the previous values are used in conjunction with the
kinematic viscosity shown in Equation A-9. These are substituted into Equation A-10 to calculate
the Reynolds number and values for this can be found in Table A-3.

**Equation A-9: Kinematic viscosity used for Reynolds number calculations**

\[
\nu = 2.96 \times 10^{-6} \left( \frac{m^2}{s} \right)
\]

**Equation A-10: Equation used to calculate Reynolds number**

\[
Re = \frac{VL}{\nu}
\]

**Table A-3: Results of Reynolds Number Calculations**

<table>
<thead>
<tr>
<th>Channel Width (m)</th>
<th>0.0005</th>
<th>0.001</th>
<th>0.002</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.005</td>
<td>55.17</td>
<td>110.34</td>
<td>220.67</td>
</tr>
<tr>
<td>0.01</td>
<td>29.51</td>
<td>59.02</td>
<td>118.03</td>
</tr>
<tr>
<td>0.02</td>
<td>15.29</td>
<td>30.58</td>
<td>61.15</td>
</tr>
<tr>
<td>0.03</td>
<td>10.32</td>
<td>20.63</td>
<td>41.26</td>
</tr>
</tbody>
</table>
Appendix B
Sample Model

An example of a design vector represented by a model is presented to better represent how geometries are designs and how constraints interact between the design variables. The following design vector is used for this example:

\[
[10 15 50 60 80 95 150 170 10 20 15 30 35 40 45 50 60 70]
\]

This gives us the following design variables seen in Table B-4.

**Table B-4: Design variables for sample model**

<table>
<thead>
<tr>
<th>Variable #</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
<th>10</th>
</tr>
</thead>
<tbody>
<tr>
<td>X Variable</td>
<td>10</td>
<td>15</td>
<td>50</td>
<td>60</td>
<td>80</td>
<td>95</td>
<td>150</td>
<td>170</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Y Variable</td>
<td>10</td>
<td>20</td>
<td>15</td>
<td>30</td>
<td>35</td>
<td>40</td>
<td>45</td>
<td>50</td>
<td>60</td>
<td>70</td>
</tr>
</tbody>
</table>

This design vector gives us the model found in Figure B-2. Some examples of the constraints and design variables can be now seen. The frequency for this model would be 2 as there are two sets of curves seen defined by x-variables 1-4 and x-variables 5-8. There are four amplitudes for this model. The amplitudes are summarized in Table B-5.

**Table B-5: Calculation of amplitudes for sample model**

<table>
<thead>
<tr>
<th>Amplitude</th>
<th>Variables</th>
<th>Calculation</th>
<th>Resulting Amplitude</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Y8-Y1</td>
<td>50-10</td>
<td>40</td>
</tr>
<tr>
<td>2</td>
<td>Y9-Y1</td>
<td>60-10</td>
<td>50</td>
</tr>
<tr>
<td>3</td>
<td>Y9-Y2</td>
<td>60-20</td>
<td>40</td>
</tr>
<tr>
<td>4</td>
<td>Y10-Y2</td>
<td>70-20</td>
<td>50</td>
</tr>
</tbody>
</table>
There are nine channel widths for this model. The primary and secondary models will only ever have one channel width; however this will be calculated with the same calculation as the first channel width in this example (substituting the design variables that represent these lines in the respective models) so this example will demonstrate calculations for all three models. The channel widths are summarized in Table B-6.

**Table B-6: Summary of channel widths for sample models.**

<table>
<thead>
<tr>
<th>Channel Width</th>
<th>Variables</th>
<th>Calculation</th>
<th>Resulting Channel Width</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Y8-Y5</td>
<td>50-35</td>
<td>15</td>
</tr>
<tr>
<td>2</td>
<td>X2-X1</td>
<td>15-10</td>
<td>5</td>
</tr>
<tr>
<td>3</td>
<td>Y3-Y1</td>
<td>15-10</td>
<td>5</td>
</tr>
<tr>
<td>4</td>
<td>X4-X3</td>
<td>60-50</td>
<td>10</td>
</tr>
<tr>
<td>5</td>
<td>Y9-Y6</td>
<td>60-40</td>
<td>20</td>
</tr>
<tr>
<td>6</td>
<td>X6-X5</td>
<td>95-80</td>
<td>15</td>
</tr>
<tr>
<td>7</td>
<td>Y4-Y2</td>
<td>30-20</td>
<td>10</td>
</tr>
<tr>
<td>8</td>
<td>X8-X7</td>
<td>170-150</td>
<td>20</td>
</tr>
<tr>
<td>9</td>
<td>Y10-Y7</td>
<td>70-45</td>
<td>25</td>
</tr>
</tbody>
</table>

There are a large number of constraints, so a couple will be given for each set to demonstrate how they are used. The first set of constraints are used to limit the channels from crossing themselves along the x-axis, and to ensure a minimum distance from each other. One example of this is to restrict the first channel from crossing itself.

\[ X(j + 1) - X(j) \geq X_{min} \]

The first channel is defined by X1 and X2 so this constraint would be created when j=1.

\[ X(1 + 1) - X(1) \geq X_{min} \]

The minimum gap in this study is 5mm to ensure the mesh is stable enough for CFD convergence.
\[ X(2) - X(1) \geq 5 \]

We can see that this constraint is satisfied with the given design variable. This constraint ensures that during optimization X2 can never be within 5mm of X1 and always has to be larger ensuring that this channel does not invert.

Figure B-2: Resultant sample model for sample design vector

The second use of this constraint is to ensure a minimum distance between vertical channels. To demonstrate this we will look at the constraint to ensure the first and second vertical
channels are separated. The gap between them is defined by \( X_2 \) and \( X_3 \). This means that this constraint will be applied when \( j=2 \).

\[
X(2 + 1) - X(2) \geq X_{\text{min}} \\
X(3) - X(2) \geq 5
\]

We can again see this constraint is satisfied and how it ensures that there is a minimum amount of space between the vertical channels to ensure the mesh is stable enough for analysis.

The next two constraints are used to ensure that the outer bounds of the channel do not exceed a minimum distance from the edge of the boundaries. Since there are two channels and the height of the plate is 160mm, the boundary of \( y \) is 80. The boundary for the \( x \) axis is the plate width which is 200. An example of these constraints for the \( y \)-axis is to ensure the top of the first horizontal length of the channel does not reach past the middle of the plate (potentially intersecting the bottom of the second horizontal length of the second channel). This will employ the first of the two constraints. This is a \( y \)-axis constraint so \( n=y \). This involves the first length of the channel so this constraint will be applied when \( j=10 \).

\[
\text{Boundary} - n(j) \geq n_{\text{min}} \\
80 - Y(10) \geq 5
\]

An example of these constraints for the \( x \)-axis is to ensure the start of the first vertical section of the channel does not occur before the beginning of the plate. This will employ the second of the two constraints. Since this is for the first vertical length the constraint will occur when \( j=1 \) and since it is an \( x \)-axis constraint \( n=x \).

\[
n(j) - 0 \geq n_{\text{min}} \\
X(1) - 0 \geq 5
\]

We can see that both forms of these constraints are satisfied for this model.
The final constraint is employed in the same manner as the first constraint however for the y-axis. This constraint varies depending on the model. For the primary model it follows the form of the first constraint exampled above and shown in Equation 3-2. For the secondary model the constrains takes on a different form to ensure that the various horizontal channel segments are separated by a minimum gap. This is also the method used by the tertiary model, however with more design variables there are a larger number of constraints. The tertiary model also employs this same constraint to ensure there is a minimum channel width throughout the system.

An example of this constraint ensuring a minimum gap between channels could be ensuring a minimum gap between the first and second horizontal channel segments. The constraint would follow the form:

\[ Y(5) - Y(3) \geq 5 \]

An example of this constraint ensuring a minimum channel width for the tertiary model would be the constraint following the form:

\[ Y(3) - Y(1) \geq 5 \]

These constraints follow a number of geometric sequences for the primary and secondary model in order to constrain all boundaries and variables. There is thus no simple representative equation to represent them for the secondary and tertiary model, however the explanation above demonstrates their use.