Performance Analysis and Modeling of a Printed Circuit Heat Exchanger with Air and Carbon Dioxide as Working Fluids

by

Amr Daouk

A thesis submitted to
the Faculty of Graduate and Postdoctoral Affairs
in partial fulfillment of the requirements for the degree of
Master of Applied Science
in
Mechanical Engineering

Department of Mechanical and Aerospace Engineering
Carleton University
Ottawa, Ontario, Canada
September 2016

Copyright © 2016 by Amr Daouk

Abstract

A Printed Circuit Heat Exchanger (PCHE) was tested using air and carbon dioxide as working fluids to determine the temperature behavior of the fluid in the PCHE. These tests are conducted and analyzed to pave way for testing the heat exchanger with supercritical carbon dioxide ($S - CO_2$), to obtain data on its performance for use in $S - CO_2$ Brayton cycles.

The tests have been conducted at thermal steady state where a total of 18 data sets have been tested. Air inlet temperature has been varied from $70^\circ C$ to $100^\circ C$ to $140^\circ C$ where both air and CO_2 were both kept at a pressure of either 5 bars or 10 bars while varying the flow from 5 LPM to 10 LPM. Results pertaining to the heat rate and pressure drop were analyzed and discussed.

A 3D COMSOL model was created to simulate the PCHE's performance and the results obtained from the simulations have been analyzed and compared to the results obtained experimentally. The results show an average percentage error of 5.65% and 5.73% when comparing the outlet temperatures of air and CO_2 respectively.

Further improvements to the test loop is required to remove limitations constricting the range of operation of the loop allowing us to obtain more data in wider ranges of temperatures, pressures and flow rates.

Acknowledgments

I would like to express my deepest gratitude and regards to my supervisors, Dr. Henry Saari and Dr. Oren Petel, for their excellent guidance, patience, and willingness to provide support whenever it was needed during the course of this project. I am also grateful to Natural Resources Canada for providing this research opportunity with special thanks to Nema Najafali for providing a great and welcoming environment at Canmet Energy.

I would like to thank my Father for constantly pushing me to become better and better and my Mother who provided me with comfort with every step I took. My parents have sacrificed so much for me to be here and I owe them everything.

I would also like to thank my friends who made this an incredible journey and helped me become a better person everyday with special thanks to Rim for her massive support.

Table of Contents

Abstract	ii
Acknowledgments	iii
Table of Contents	iv
List of Tables	vii
List of Figures	viii
1 Introduction	1
1.1 Power Conversion Cycles	1
1.2 Motivation	2
1.2.1 Carleton University's $S - CO_2$ Closed Loop Brayton Cycle . .	2
1.2.2 Heat Exchangers	3
1.3 CanmetENERGY Printed Circuit Heat Exchanger	4
1.4 Objective	4
2 Literature Review	6
2.1 SCO_2 Brayton Cycles	6

2.2	Printed Circuit Heat Exchangers	16
2.3	Supercritical Carbon Dioxide Correlations	27
2.4	Summary	32
3	Experimental Setup	34
3.1	CO_2 /Air Test Loop	34
3.2	Printed Circuit Heat Exchanger (PCHE)	38
3.3	Gas Booster	41
3.4	Heater	42
3.5	Instrumentation	43
3.5.1	Pressure	43
3.5.2	Temperature	45
3.5.3	Mass Flow	47
3.5.4	Data Acquisition (DAQ)	48
4	Numerical Model	49
4.1	Introduction	49
4.2	Theoretical Background	49
4.2.1	Conservation of Mass	49
4.2.2	Conservation of Momentum	50
4.2.3	Conservation of Energy	50
4.3	PCHE Dimensions	52
4.4	COMSOL Model	56
4.4.1	Model Assumptions	57
4.4.2	Boundary Conditions	59

4.5	Grid Analysis	61
4.5.1	Mesh Creation	61
4.5.2	Mesh Sensitivity Analysis	63
4.6	Solver	68
5	Results and Discussion	70
5.1	Introduction	70
5.2	Data Analysis	73
5.3	Pressure Drop	75
5.4	Heat Rate	79
5.5	Temperature	85
5.5.1	Experimental and Numerical Results	87
5.5.2	Hot Inlet Temperature Gradient	92
5.5.3	Effect of Flow on Temperature Behavior	98
5.5.4	Overall Heat Transfer Coefficient	101
5.5.5	Design Point Simulation	105
6	Conclusions	108
6.1	Overview	108
6.2	Future Work	110
	References	112
	Appendix A Error Analysis	116
	Appendix B 3D COMSOL Results	120

List of Tables

2.1	FE, NE, EERE Application Space [9]	15
3.1	PCHE Design Point	39
4.1	PCHE details provided by Heatric	53
4.2	CO_2 Outlet Temperature with Varying Number of Elements	64
5.1	Test Specifications	72
5.2	Reynold's Number and Pressure Drop values for CO_2	76
5.3	Reynold's Number and Pressure Drop values for air	76
5.4	Calculated Heat Rate	80
5.5	Test Condition with Outlet Temperatures	86
5.6	Comparison of Experimental and Simulation Results	88
5.7	PCHE Design Point	102
5.8	Overall Heat Transfer Coefficients	104
A.1	Test 1 Air Conditions	118
A.2	Enthalpy for Air	118

List of Figures

1.1	Carleton University Brayton Cycle Loop [2]	2
2.1	Cycle Efficiency vs Source Temperature for Steam, CO_2 and He En- ergy Cycles [3]	7
2.2	Phase Diagrams of CO_2 in Pressure-Temperature Plane (left) and Density-Pressure Plane (right)[4]	7
2.3	Turbine size comparison amongst different power cycles [1]	9
2.4	Closed Loop Brayton Cycle [2]	10
2.5	Corresponding H-S Diagram [2]	11
2.6	Turbo-alternator-compressor design of Sandia S- CO_2 test loop [5] . .	12
2.7	Simple diagram of Sandia S- CO_2 test loop [5]	12
2.8	EPS100 Configuration [7]	14
2.9	Echogen current and future gas turbines[8]	14
2.10	DOE S- CO_2 Brayton cycle designs for indirect and direct heating [9]	16
2.11	Overview of compact heat transfer surfaces [11]	17
2.12	Size comparison of Shell and Tube heat exchangers and PCHEs of the same heat load. [10]	18
2.13	Etched Plate [12]	19

2.14	Micrograph of section through diffusion bonded PCHE core with channels ranging from 0.5 - 5 mm in diameter [13]	19
2.15	Section of stacked plates showing microchannels [10]	20
2.16	Argonne National Laboratory PCHE setup [15]	21
2.17	Zigzag channel length and angles. [15]	22
2.18	Friction factor for PCHE channels [15]	23
2.19	Example of temperature gradient (top) and velocity gradient (bottom) from Kar simulations [16]	24
2.20	Figelys simplified model (left) and complex model(right). [17]	25
2.21	Van Meters 2D model [20]	26
2.22	ANSYS CFX model for Kim et al. [22]	27
3.1	CO_2 /Air PCHE Test Loop Front	35
3.2	CO_2 /Air PCHE Test Loop Back	36
3.3	Piping and Instrumentation Diagram for PCHE Test Loop	37
3.4	PCHE Used in Test Loop	39
3.5	PCHE Data Sheet	40
3.6	Gas Booster	41
3.7	Watlow Immersion Heater	42
3.8	Swagelok Pressure Gauge.	43
3.9	ABB 266MSH Differential Pressure Transmitter	44
3.10	Swagelok Pressure Regulator for Air and CO_2	45
3.11	Swagelok Temperature Gauge	46
3.12	6 in J-Type Thermocouple	47
3.13	Alicat Scientific MC Mass Flow Meter	47

3.14	Data Acquisition System	48
4.1	PCHE Data Sheet	52
4.2	Simplified shape of PCHE channel [19]	55
4.3	Uninsulated PCHE showing the inlets and outlets	56
4.4	COMSOL Model of the whole PCHE	58
4.5	COMSOL Model of one column of PCHE (left) and an enlargement of model channels (right)	58
4.6	Quadrilateral surface mesh with boundary layers	62
4.7	Quadrilateral surface mesh with boundary layers	63
4.8	Number of Mesh Elements vs Temperature	64
4.9	Mesh quality for Meshram et al. Model [34]	66
4.10	Mesh quality for Li et al. Model [33]	66
4.11	Mesh quality for Figley Model [17]	67
4.12	Mesh quality for Kim et al. Model [22]	68
5.1	Reynold's Number vs Pressure Drop for Air	77
5.2	Reynold's Number vs Pressure Drop for CO_2	78
5.3	Air Inlet Temperature vs. Heat Rate for Test Set A	81
5.4	Air Inlet Temperature vs. Heat Rate for Test Set B	82
5.5	Air Inlet Temperature vs. Heat Rate for Test Set C	83
5.6	Test 4 Temperature Distribution	90
5.7	Isometric View of Temperature Distribution in Central Channels . . .	91
5.8	Test 1 Temperature Distribution	93
5.9	Effect of Flow Rate on Temperature Drop Gradient	95

5.10	Enlargement of the Inlet of "Effect of Flow Rate on Temperature Drop Gradient"	95
5.11	Temperature Profiles for Vanmeter with CO_2 at (a)100 kg/h (top) and (b) 200 kg/h (bottom) [20]	96
5.12	Temperature Profiles for Vanmeter with CO_2 at (a)300 kg/h (top) and (b) 400 kg/h (bottom) [20]	97
5.13	Test 2 Temperature Distribution	99
5.14	Test 8 Temperature Distribution	99
5.15	Test 14 Temperature Distribution	100
5.16	PCHE Partial Specifications	102
5.17	Design Point Simulated Temperature Distribution	105
5.18	Temperature Difference Trend with Variation of Air Inlet Temperature	106
B.1	Test 1 Temperature Distribution	121
B.2	Test 2 Temperature Distribution	122
B.3	Test 3 Temperature Distribution	123
B.4	Test 4 Temperature Distribution	124
B.5	Test 5 Temperature Distribution	125
B.6	Test 6 Temperature Distribution	126
B.7	Test 7 Temperature Distribution	127
B.8	Test 8 Temperature Distribution	128
B.9	Test 9 Temperature Distribution	129
B.10	Test 10 Temperature Distribution	130
B.11	Test 11 Temperature Distribution	131
B.12	Test 12 Temperature Distribution	132

B.13 Test 14 Temperature Distribution	133
B.14 Test 15 Temperature Distribution	134
B.15 Test 16 Temperature Distribution	135
B.16 Test 17 Temperature Distribution	136
B.17 Test 18 Temperature Distribution	137

Chapter 1

Introduction

1.1 Power Conversion Cycles

With the increasing power demand in the world due to the increasing population, more efficient power conversion cycles are required. The conventional steam Rankine cycles require turbines to operate at high temperatures, have high capital costs, are large in size and have relatively low efficiencies reaching 40% [1]. Alternative power conversion cycles are being developed to replace the current Rankine steam cycles by using supercritical carbon dioxide ($S - CO_2$) as the working fluid of these cycles. The most popular of cycles for the use $S - CO_2$ is the closed loop $S - CO_2$ Brayton cycle which is due to the manipulation of the non-ideal gas above the critical point. $S - CO_2$ Brayton cycles provide smaller infrastructure, lower costs and higher efficiencies when compared to the conventional Rankine steam cycles.

1.2 Motivation

1.2.1 Carleton University's $S - CO_2$ Closed Loop Brayton Cycle

One of the fourth year engineering projects in the Department of Mechanical and Aerospace Engineering at Carleton University is the Carleton University Brayton Cycle Loop (CU-BCL). It is a pilot scale indirectly fired closed loop power cycle which is funded by CanmetENERGY, a division of Natural Resources Canada. A schematic of the plant is shown in Figure 1.1.

Figure 1.1: Carleton University Brayton Cycle Loop [2]

The aim is to construct a 250 kW thermal pilot scale $S - CO_2$ Brayton cycle loop for use in high efficiency fossil fuel based electricity generation systems.

1.2.2 Heat Exchangers

With a large amount of energy being transferred in the $S - CO_2$ cycles, the use of heat exchangers is common in order to manipulate the temperature of the cycle to reach the desired values. For example, as seen from Figure 1.1, the CU-BCL uses three heat exchangers. A main heat exchanger coupled with the heat source to transfer the energy from burning of fossil fuels to the fluid, a precooler before entering the compressor to drop the temperature and finally a recuperator which transfers most of the energy within the cycle and helps increase the turbine inlet temperature while lowering the thermal energy provided from the heat source. For these kinds of cycles to be efficient, highly effective heat exchangers are required.

Compact heat exchangers were developed to meet the demand of high effectiveness, high integrity and small size heat exchangers to be coupled with the power conversion cycles and that is what makes them appealing for use. Heatric, a company specializing in the design of compact heat exchangers specializes in the design of Printed Circuit Heat Exchangers (PCHEs) which are currently widely being studied for $S - CO_2$ applications. PCHEs are heat exchangers made through chemically etching semicircular channels onto several steel plates. When the etching is complete, the plates are stacked and bonded together where nozzles are then welded into the inlets and outlets.

Given that PCHEs are currently being widely studied for $S - CO_2$ applications, there remains a major lack of data in the literature. Two major areas where this

lack of data exists are in off design conditions and low flow conditions.

1.3 CanmetENERGY Printed Circuit Heat Exchanger

Modelling the PCHEs for performance of the cycle has been proven to be difficult due to the lack of geometrical data of the interior of the PCHE due to this information being proprietary. Therefore, a heat exchanger was purchased by CanmetENERGY in order to conduct some performance testing. The PCHE purchased by CanmetENERGY is designed for testing air and CO_2 as the working fluids. This PCHE is much smaller than the ones used by CU-BCL and was not designed for $S - CO_2$ however testing and modelling it would provide invaluable data to be used when modelling the CU-BCL PCHEs especially for low flow and off design conditions.

1.4 Objective

The motivation for the work conducted was to explore the performance of a PCHE and obtain performance data to fill the gap currently present for low flow and off design conditions and also to be used in modelling the performance of the PCHEs used in CU-BCL. Ultimately these performance models would be used as a tool to predict the heat exchanger behaviors while modeling the Brayton cycle loop. In a collaboration project between Carleton University and CanmetENERGY, a PCHE test loop has been constructed to test the performance of the PCHE when using air and CO_2 below the critical region as working fluids.

Chapter 2 will discuss a literature review where similar work has been done around the world. Chapter 3 outlines the design of the test loop whereas Chapter 4 outlines the details of the CFD model created to model the PCHE. Chapter 5 discusses the results obtained from the experiments conducted and finally, Chapter 6 concludes this thesis with a summary followed by some recommendations.

Chapter 2

Literature Review

2.1 SCO_2 Brayton Cycles

S- CO_2 Brayton Cycle is a high efficiency closed loop power conversion cycle that uses carbon dioxide in its supercritical state as a working fluid. S- CO_2 power cycles can be coupled with almost any heat source including nuclear, solar or fossil fuel and when compared to a conventional Rankine cycle, has smaller infrastructure and lower capital costs [3]. Figure 2.1 shows cycles efficiencies of steam, CO_2 and Helium cycles where CO_2 shows the highest efficiencies with relatively low source temperatures. The high efficiency of this cycle arises from the low compressibility of CO_2 (31.1 °C and 7.39 MPa) which causes a sudden increase in density with minimal pressure change.

Figure 2.1: Cycle Efficiency vs Source Temperature for Steam, CO_2 and He Energy Cycles [3]

Figure 2.2: Phase Diagrams of CO_2 in Pressure-Temperature Plane (left) and Density-Pressure Plane (right)[4]

Figure 2.2 (a) and 2.2 (b) show the phase diagrams of Carbon Dioxide for pressure-temperature and density-pressure planes. As seen from Figure 2.2(a), liquid and gaseous phases become indistinguishable above the critical point as represented by the dashed lines. Around the critical point, vast changes in density, specific heat capacity and viscosity occur with small changes in temperature. As seen from Figure 2.2(b), the changes in density are very high in the supercritical region and start to decrease as we move beyond the critical point. $S\text{-}CO_2$ is hence used in these power cycles due to its movement in gaseous form while maintaining the energy transfer properties of a liquid due to its high density. Because of these properties, $S\text{-}CO_2$ Brayton cycles use relatively small turbomachinery compared to steam Rankine cycles or Helium operated Brayton cycles. Turbines in $S\text{-}CO_2$ Brayton cycles usually provide 30% of its work towards cycle compression and that compares well to steam Rankine cycles and Helium cycles which need 45% of the turbines work for compression [1]. Figure 2.3 shows the Turbomachinery size comparison between the previously mentioned three cycles.

Figure 2.3: Turbine size comparison amongst different power cycles [1]

Other than the turbomachinery, heat exchangers play a vital role in closed loop Brayton cycles. The use of heat exchangers in these loops are needed for several reasons. A heat exchanger can be used to connect the Brayton cycle with the heat source used where one stream in the heat exchanger contains the working fluid of the cycle while the other contains the flue gas from the heat source. Another use of a heat exchanger in Brayton cycles is the use of recuperators to recirculate the heat in the cycle and increase the overall efficiency. After the stream exits the turbine, heat needs to be removed from the working fluid to enter the compressor at the desired temperature; A recuperator is then fitted in the cycle to transfer the thermal energy from the stream exiting the turbine to the stream exiting the compressor where both streams in the heat exchanger will contain the cycles working

fluid. If the temperature of the working fluid is not brought down enough from the compressor inlet, the cycle is fitted with another heat exchanger to act as a cooler where one stream is the working fluid while the other is a coolant. Figure 2.4 illustrates an example of how the heat exchanger/ turbomachinery combination can be used. Figure 2.5 shows how that configuration is translated into its corresponding H-S diagram showing the major role the heat exchangers play in heating and cooling the fluid in the cycle. As seen, most of the heat is transferred in the recuperator which recirculates the heat allowing us to achieve a high turbine inlet temperature to achieve a higher cycle efficiency while reducing the need for external heat sources to be used to continuously heat the fluid to achieve that high temperature.

Figure 2.4: Closed Loop Brayton Cycle [2]

Figure 2.5: Corresponding H-S Diagram [2]

In 2010, Sandia National Laboratories [5] investigated advanced Brayton cycles using $S\text{-CO}_2$ as the working fluid. Sandia built a small scale, 260 kW thermal, Brayton cycle loop to study some of the issues that arise when compressing CO_2 near the critical point [5]. Given the small amount of thermal energy, small single stage radial turbomachinery were chosen. A 1.2 in diameter turbine was used alongside a 1.47 in diameter compressor in the turbo-alternator-compressor setup as seen in Figure 2.6. Figure 2.7 shows a simple diagram of the $S\text{-CO}_2$ test loop with the corresponding temperatures and pressures.

Figure 2.6: Turbo-alternator-compressor design of Sandia S-CO₂ test loop [5]

Figure 2.7: Simple diagram of Sandia S-CO₂ test loop [5]

In general, the results from their tests agreed fairly well with their models showing that the heated but unrecuperated Brayton loop tests showed that the turbo-compressor is near break-even conditions for a turbine inlet temperature of 60°C and as expected, the higher the turbine inlet temperature, the higher the efficiency of the cycle. Since then, this Sandia loop has been a benchmark for small scale S-CO_2 Brayton cycle loops done by Sandia which include their 1MW thermal loop.

Echogen Power Systems LLC started looking into using S-CO_2 as their working fluid to replace steam in their heat recovery systems. In 2011, they started with a pilot scale 250kWe demonstration thermal engine which completed its initial testing at the American Electric Powers research center [6]. Their models and measurements seemed to fairly agree which gave them the confidence to continue in creating larger cycles using S-CO_2 . Echogen then released the EPS100 CO_2 heat engine for commercial use which is seen in Figure 2.8 [7]. The EPS100 is designed as a 7 to 8 MW class heat recovery engine where the combustion products are in a 500°C to 550°C range with a flow rate of approximately 65 to 70 kg/s. Two separate turbines are used, where one drive turbine is connected directly to the compressor while the other power turbine is coupled to a four-pole synchronous generator through a gearbox for power generation. All the recuperators used are Printed Circuit Heat Exchangers which are introduced in section 2.3. Figure 2.9 shows the future of Echogens gas turbines, where they aim to release the EPS200, a 16MW class heat recovery engine [8].

Figure 2.8: EPS100 Configuration [7]

Figure 2.9: Echogen current and future gas turbines[8]

The United States Department of Energy (DOE) has contributed to the research on $S\text{-}CO_2$ Brayton Cycles to use in replacement of the incumbent steam Rankine cycles. The DOE started a collaboration between Nuclear Energy (NE), Fossil Energy (FE) and Energy Efficiency and Renewable Energy (EERE) as energy sources to power the Brayton Cycles. Table 2.1 shows their application space.

Table 2.1: FE, NE, EERE Application Space [9]

<i>Application</i>	<i>Size(MWe)</i>	<i>Temperature($^{\circ}C$)</i>	<i>Pressure(MPa)</i>
Nuclear (NE)	10 - 300	350 - 700	20 - 35
Fossil Fuel (Indirect Heating) (FE)	300 - 600	550 - 900	15 - 35
Fossil Fuel (Direct Heating) (FE)	300 - 600	1100 - 1500	35
Concentrating Solar Power (EERE)	10 - 100	500 - 1000	35
Shipboard Propulsion	<10 - 10	200 - 300	15 - 25
Waste Heat Recovery (FE)	1 - 10	<230 - 650	15 - 35
Geothermal (EERE)	1 - 50	100 - 300	15

The DOE had focused on two main designs for their Fossil Energy cycles. One was for Indirect heating, where a heater would be coupled to the primary heat exchanger which gave a turbine inlet temperature ranging from $650^{\circ}C$ to $750^{\circ}C$ and direct heating which included a pressurized oxy-combustion system coupled with a turbine, generator and the primary heat exchanger, which in turn provided a turbine Inlet Temperature of $1150^{\circ}C$. Figure 2.10 shows the design of both cycles [9].

Figure 2.10: DOE S- CO_2 Brayton cycle designs for indirect and direct heating [9]

The DOE started a design for a 550 MWe indirect heating S- CO_2 Brayton cycle power plant. As seen from Figure 2.8, the indirect S- CO_2 cycle is a recompression cycle (i.e. contained two compressors to compress the fluid twice). The DOE's models and analyses concluded that the cycle's overall efficiency was sensitive to both pressure drop across the cycle and heat transfer efficiency across the heat exchangers. Therefore, their challenge was to design recuperators to maximize heat transfer efficiency, minimize pressure drop and minimize cost. To meet their design criteria, they contracted with Thar energy that provided them with recuperators that are 99% effective and with a maximum pressure drop of 20 psi which gave an approximate cycle of efficiency of 54% [9].

2.2 Printed Circuit Heat Exchangers

The need for more compact and higher efficiency heat exchangers to be used in power cycles has increased over the years. A regular shell and tube heat exchanger occupies a large footprint and has effectiveness values ranging from 60% to 85% depending

on the conditions used [10]. Therefore, the use of compact heat exchangers has increased over time due to their smaller size and higher efficiencies. Figure 2.11 shows an overview of compact heat transfer surfaces. As seen in the figure, compact heat exchangers contains a much higher heat transfer surface area density as the size of the channels decrease which contributes to the higher effectiveness values and the smaller footprint

Figure 2.11: Overview of compact heat transfer surfaces [11]

Printed Circuit Heat Exchangers (PCHEs) are a type of compact heat exchangers manufactured by Heatric, a subsidiary of Meggitt (UK) Ltd, which for the same thermal duty, are up to 85% smaller than an equivalent shell and tube heat exchanger. Figure 2.12 shows the comparison of a PCHE to a shell and tube heat exchanger of the same heat load.

Figure 2.12: Size comparison of Shell and Tube heat exchangers and PCHEx of the same heat load. [10]

PCHEx are made through chemically etching semicircular channels onto a steel plate for fluid passage. Tens of plates are then stacked on top of each other. The plates go through a diffusion bonding process which encourages grain growth across the initial plate boundaries. The plates are then bonded into a monolithic structure, and enveloped in a casing [10, 12, 13]. Figures showing the etched plates, a micro-graph of a bonded PCHEx core and a section of the stacked plates can be found in Figures 2.13, 2.14 and 2.15.

Figure 2.13: Etched Plate [12]

Figure 2.14: Micrograph of section through diffusion bonded PCHE core with channels ranging from 0.5 - 5 mm in diameter [13]

Figure 2.15: Section of stacked plates showing microchannels [10]

Due to confidentiality policies, information related to the internal geometry and channel arrangements are not always disclosed. For this reason, modelling of these PCHEs has been proven difficult. Companies such as Thar Energy, CompRex LLC and Vacuum Process Engineering, who are known manufacturers of compact heat exchangers are more open to sharing their designs with their customers, therefore modeling of their heat exchangers has been relatively easier than modeling heat exchangers manufactured by Heatric. Several researchers have estimated internal geometries of the PCHEs in order to model their heat exchange characteristics, for example, Pieve [14], described several methods that can be used to size the channels of the heat exchanger. These techniques simplify efforts to properly model the thermal

behavior of the fluid in the heat exchanger.

Argonne National Laboratories purchased a small PCHE from Heatric, which was rated at 200°C and hot and cold side pressures of 82 and 216 bar respectively. Moissyetsev et al. [15] tested the heat exchanger having both streams as CO_2 to provide an evaluation of heat transfer modeling of the PCHE.

Figure 2.16: Argonne National Laboratory PCHE setup [15]

Flow areas, hydraulic diameters, total number of channels were supplied by manufacturer however important data such as channel angles, active heat transfer area length and plate arrangement were not provided. Assumptions were made in order to model the PCHE, the channels were assumed semicircular in cross section, the header was assumed to consist of a 2 cm metal pressure boundary, 5 cm nozzles and

a 4 cm flow distribution region and it was assumed that the frictional pressure drop in the heat transfer area constitutes 80% of the total pressure drop.

The heat transfer length was calculated as the heat exchanger length minus twice the header length. The channel length is calculated based on the channel perimeter, total surface area and number of channels on each side. They've used the ratio of the channel length to the heat transfer length to calculate the channel angle as seen in Figure 2.17.

Figure 2.17: Zigzag channel length and angles. [15]

The data they gathered along with the channel geometry calculations helped them develop friction factor correlations for the PCHE which when compared to Heatric's friction factor correlation curves, ranged fairly well as seen in Figure 2.18. From there pressure drop correlations were developed and can be found in section 2.3.

Figure 2.18: Friction factor for PCHE channels [15]

Several researchers have developed CFD models of Heatric PCHEs. Kar [16], developed a 3D model for a Heatric PCHE with air as the working fluid. He modelled a single channel once as a semicircle and once as an ellipsis giving that channel a certain amount of heat loss as a boundary condition and observed its temperature and velocity behaviors using ANSYS Fluent as seen from Figure 2.19.

Figure 2.19: Example of temperature gradient (top) and velocity gradient (bottom) from Kar simulations [16]

Figley [17] developed a 3D model for a PCHE using Helium as the working fluid using ANSYS Fluent to create and simulate his models. Figley's model was later redeveloped and modified by Daouk et. al using COMSOL Multiphysics for both modelling and simulation, where results were compared and discussed [18]. Figley created two models for comparison, a complex model which modelled the entire heat exchanger with a total of 240 channels and a simplified model which modelled only one column of the heat exchanger containing 20 channels as seen in Figure 2.20.

Figure 2.20: Figelys simplified model (left) and complex model(right).
[17]

His results showed a difference of up to 0.52% when comparing outlet temperatures between both models and a pressure drop of up to 0.68% when comparing the pressure drop across the channels. These results give confidence to model only a section of the heat exchanger in order to save computational expense.

Argonne National Laboratories have purchased and commissioned a 17.5 kW PCHE from Heatric with $S\text{-}CO_2$ and water as the working fluids. Song [19] and Van Meter [20] have modelled the PCHE and discussed their results. Their models have shown a 15% relative error for the outlet temperatures of the PCHE when compared to the data obtained and were successfully able to model the zig zag arrangement of the channels to accurately calculate the pressure drop as seen in Figure 2.21.

Figure 2.21: Van Meters 2D model [20]

In 2006, Ishizuka et al. [21] from Tokyo Institute of Technology (TiTech) have constructed a PCHE test loop to test a 3 kW PCHE purchased from Heatric. They were able to successfully model the PCHE numerically using ANSYS Fluent and a quasi-two-dimensional calculation code. From their results, they were successfully able to calculate the heat transfer coefficient of the fluid and calculate the heat exchanger effectiveness to be 99%. Van Meter recreated their models and his error values when comparing his results with those of TiTech ranged from 0.10% to 7.94% when calculating the outlet temperature values.

S. Kim et al. [22] using ANSYS CFX was able to create and simulate a 3D model of 3 channels of a PCHE used in an $S\text{-}CO_2$ Brayton cycle while including periodic boundary conditions on all sides of the model to simulate the entire PCHE.

The results were then compared to three existing correlations for calculating Nusselt number, including the correlations developed by Ishizuka et al. [21], and the results varied from 5.6% difference to 44% difference.

Figure 2.22: ANSYS CFX model for Kim et al. [22]

2.3 Supercritical Carbon Dioxide Correlations

Two of the most common and traditional correlations used today to calculate the Nusselt number are the Gnielinski and the Dittus-Boelter correlation. The Gnielinski correlation [23] is given by

$$Nu_G = \frac{\frac{f}{8}(Re - 1000)Pr}{1 + 12.7(\frac{f}{8})^{0.5}(Pr^{\frac{2}{3}} - 1)} \quad (2.1)$$

where Pr is the Prandtl number, f is the friction factor given by

$$f = (1.82 \log(Re) - 1.64)^{-2} \quad (2.2)$$

and Re is the Reynolds number which is >2300 .

The Dittus-Boelter correlation is given by

$$Nu_{DB} = 0.023 Re^{\frac{4}{5}} Pr^{\frac{1}{3}} \quad (2.3)$$

where the correlation is valid for $10,000 < Re < 120,000$ and $0.7 < Pr < 120$.

For calculating the friction factor, the most conventional correlation used is the Blasius equation which, for turbulent flow, is given by

$$f = \frac{0.316}{Re^{0.25}} \quad (2.4)$$

Where $4000 \leq Re \leq 105$. For laminar flow, the equation is given by

$$f = \frac{C_f}{Re} \quad (2.5)$$

Where C_f is a constant with the value of 64 for circular channels and is a function of the height and width ratio for rectangular channels.

The Gnielinski and Dittus-Boelter correlations are both meant for conventional sized channels. The definition of the term conventional has been debated by researchers where some have quoted diameter values ranging from 2 cm [24] to 3 mm

[25] as the lower limit. A similar problem arises for the friction factor correlations where researchers have observed that for micro channels, the transition away from the laminar regime has been noted at Reynolds numbers as low as 200 and observed a fully turbulent regime at Reynolds numbers ranging from 400 - 1500 [26, 27]. Given that the flow behaves differently in micro channels than it does in conventional sized channels, research has been conducted on several mini/micro channels to develop their corresponding Nusselt number and friction factor correlations.

Peng et al. [28] conducted experiments to investigate the flow characteristics for water flowing through rectangular microchannels with hydraulic diameter values ranging from 0.133 - 0.367 mm. The data obtained corresponded to the following correlation for friction factor

$$f = \frac{C_{f,l}}{Re^{1.98}} \quad (2.6)$$

for laminar flow and

$$f = \frac{C_{f,t}}{Re^{1.72}} \quad (2.7)$$

for turbulent flow.

The following correlations were developed for the Nusselt number calculation

$$Nu = C_{H,l} Re^{0.62} Pr^{\frac{1}{3}} \quad (2.8)$$

for laminar flow where $C_{H,l}$ is a coefficient dependent on channel geometry and

$$Nu = C_{H,t} Re^{\frac{4}{5}} Pr^{\frac{1}{3}} \quad (2.9)$$

for turbulent flow where $C_{H,t}$ is a coefficient dependent on channel geometry and Moisseytsev et al. [15] conducted experiments to test a PCHE that uses CO₂ as both the cold and hot stream. Given that the channels in a PCHE are micro and zigzagged, a friction factor correlation based on their data was developed. For straight channels, theyve calculated the friction factor through

$$f_0 = \frac{16}{Re} \quad (2.10)$$

for $Re < 1700$, and

$$f_0 = \frac{0.0791}{Re^{0.25}} \quad (2.11)$$

for $Re > 2300$. The zigzag channel enhancement is calculated through

$$\frac{f}{f_o} = 1 + a_f(Re + 50) \quad (2.12)$$

for $Re < 1300$, and

$$\frac{f}{f_o} = kRe^c \quad (2.13)$$

for $Re \geq 1300$, where

$$c = \frac{\ln\left(\frac{1+223283a_f^2}{1+a_f^2(1300+50)}\right)}{\ln\left(\frac{100,000}{1300}\right)} \quad (2.14)$$

$$k = \frac{1 + a_f(1300 + 50)}{1300^c} \quad (2.15)$$

$$a_f = 4.5 \times 10^{-3} \tan\left(\frac{\alpha}{2}\right) \quad (2.16)$$

As mentioned in section 2.1, fluids in the supercritical region behave differently due to the drastic change in density around the critical point. Therefore, correlations for Nusselt number calculations concerning supercritical CO_2 were developed by researchers to take into consideration the effects of the proximity to the critical point.

S. Liao and T. Zhao [29, 30] conducted experiments to measure heat transfer coefficients of S- CO_2 flowing through horizontal mini/micro channels and in miniature tubes. Theyve developed correlations to calculate the Nusselt number for the bulk fluid through

$$Nu_b = 0.124 Re_b^{0.8} Pr_b^{0.3} \left(\frac{Gr}{Re_b^2}\right)^{0.203} \left(\frac{\rho_w}{\rho_b}\right)^{0.842} \left(\frac{c_p}{c_{pb}}\right)^{0.384} \quad (2.17)$$

and Nusselt number for the fluid at the wall through

$$Nu_b = 0.124 Re_w^{0.8} Pr_w^{0.3} \left(\frac{Gr}{Re_b^2}\right)^{0.203} \left(\frac{\rho_w}{\rho_b}\right)^{0.437} \left(\frac{c_p}{c_{pw}}\right)^{0.384} \quad (2.18)$$

where Gr is the Grashof number, ρ is the density and Cp is the specific heat capacity. The subscripts b and w represent the bulk fluid and the fluid at the wall respectively.

S. Mokry and I. Pioro [31] tested S- CO_2 flowing in a vertical bare tube and have developed Nusselt number correlations based on their data. After obtaining their data, a preliminary correlation was developed

$$Nu_b = 0.0345 Re_b^{0.77} Pr_b^{0.17} \left(\frac{\rho_w}{\rho_b} \right)^{0.47} \quad (2.19)$$

After adding some primary data to their whole set of data, the correlation was updated to

$$Nu_b = 0.0345 Re_b^{0.86} Pr_b^{0.23} \left(\frac{\rho_w}{\rho_b} \right)^{0.59} \quad (2.20)$$

As observed from both sets of correlations, the cross sectional averaged Prandtl number and the ratio of density of the fluid at the wall temperature to the density of the fluid at the bulk temperature were used in order to account for the large temperature gradients in the cross section.

2.4 Summary

A lot of research has been conducted on the performance and modelling of PCHEs. Argonne National Laboratories [15] tested a small PCHE rated at $200^\circ C$ and hot and cold side pressures of 82 and 216 bar respectively. The work they have conducted in running tests and creating a model for their PCHE allowed them to develop friction factor correlations for the PCHE which compared well to Heatric's data. CFD models have been created by researchers such as Kar [16], Figley [17] and Kim et al. [22]. Kar [16] investigated the effects the geometry of the channels have of on the temperature and velocity behaviors whereas Figley [17] investigated the results of modelling one column of the PCHE instead of the whole PCHE. Kim et al. [22] compared CFD results with known analytical correlations. Song [19], Van Meter [20] and Ishizuka

et al. [21] tested their respective PCHEs and compared their experimental data with their CFD simulation results.

Even though the research conducted has a wide range of operational data, the main focus of most of them was high flow and on design conditions leaving a gap for low flow and off design conditions. This research addresses this gap by testing and simulating the PCHE at low flow and off design conditions.

Chapter 3

Experimental Setup

3.1 CO_2 /Air Test Loop

An experimental apparatus was assembled at CanmetENERGY, a division of Natural Resources Canada (NRCAN), to examine the heat transfer and fluid flow characteristics of a Heatric PCHE. The primary use of the PCHE is to use heated air in order to heat cold CO_2 . The loop is currently not equipped to allow us to achieve supercritical conditions however, a redesign of the loop is scheduled in the future which would allow us to test CO_2 in its supercritical phase. The loop was designed by CanmetENERGY, where they purchased and setup all the equipment. Commissioning of the loop was a combined effort where CanmetENERGY helped towards operating the loop to reach stability. The bulk of the time taken was used to commission the loop where certain instruments needed to be replaced due to some issues that did not allow them to operate them normally.

In this test loop, the air and CO_2 are both held at equal pressures. Figure 3.1 and Figure 3.2 show pictures of the test loop front and back respectively whereas Figure 3.3 shows the piping and instrumentations diagram for the test loop. The top stream supplies CO_2 to the heat exchanger whereas the bottom stream supplies air. Both fluids enter the inlets of the PCHE in a horizontal direction and change to a vertical direction in the bulk of the heat exchanger providing 100% counter flow heat exchange.

Figure 3.1: CO_2 /Air PCHE Test Loop Front

Figure 3.2: CO_2 /Air PCHE Test Loop Back

Figure 3.3: Piping and Instrumentation Diagram for PCHE Test Loop

Both fluid streams flow through 1/2" 316 stainless steel pipes with the exception of the pipe going from the temperature gauge to the regulator in the CO_2 stream which is composed of 1/4" 316 stainless steel. The CO_2 stream starts from the CO_2 supply tank with a fixed pressure which then passes through a valve and temperature and pressure gauges before reaching a regulator which drops the pressure to a desired value. The mass flow rate of the flow is controlled and the temperature and pressure of the flow are registered before entering the heat exchanger. The air stream starts from the air supply, and passes by an air booster pump. After pressurizing the air, the air is circulated with a controlled mass flow rate through the loop where it passes temperature and pressure gauges as well as an electric heater before entering the heat exchanger. Two differential pressure transmitters are located at the inlet and outlet of the heat exchanger to measure the pressure drop across the PCHE.

3.2 Printed Circuit Heat Exchanger (PCHE)

The PCHE configured in this test loop was purchased from Heatric in 2008 for the purpose of testing and examining for further use in other cycles which use CO_2 as the operating fluid. This PCHE has outer dimensions of 76 x 996 x 55 mm with a nozzle size for all inlets and outlets of 20 mm NB. The nozzles and core of the PCHE are composed of 304H Stainless Steel which weighs 30 kg in total. The design heat load is 0.8 kW and the heat exchanger is designed to withstand a temperature of up to 800°C with a heat transfer area of 0.83 m^2 . Figure 3.4 shows a picture of the PCHE used in the test loop and Table 3.1 shows the design point values of this heat exchanger. The data sheet provided by Heatric can be found in Figure 3.5

Figure 3.4: PCHE Used in Test Loop

Table 3.1: PCHE Design Point

	<i>Air</i>	<i>CO₂</i>
Inlet Temperature (°C)	650	2
Outlet Temperature (°C)	5	585
Pressure (barg)	10	10
Mass Flow Rate (kg/h)	4.14	4.69

SERVICE PER ITEM		SIDE A				SIDE B				Rev
		In		Out		In		Out		
1	FLUID CONDITIONS	Air				CO2				
2	Fluid	Air				CO2				
3	Flow rate, total kg/hr	4.14				4.69				
4	Flow rate, gas/vapour kg/hr	4.14		4.14		4.69		4.69		
5	Flow rate, liquid kg/hr	0.00		0.00		0.00		0.00		
6	Temperature °C	650.0		5.0		2.0		585.0		
7	Design temp: max/min °C	800 / 0				800 / 0				
8	Pressure: inlet/design barg	0.0 / 10.0				8.1 / 10.0				
9	Pressure drop: calc/allwd bar	0.05 / 0.05				0.00 / 0.05				
10	FLUID PROPERTIES	Liq	Vap	Liq	Vap	Liq	Vap	Liq	Vap	
11	Density kg/m³	-	0.4	-	1.3	-	18.9	-	5.6	
12	Specific heat J/kg K	-	1133	-	1007	-	918	-	1178	
13	Viscosity cP	-	0.0450	-	0.0179	-	0.0137	-	0.0409	
14	Thermal conductivity W/mK	-	0.0630	-	0.0245	-	0.0159	-	0.0592	
15	Dew point °C									
16	CONNECTIONS									
17	No. of nozzles	1		1		1		1		
18	Nozzle size mm NB	20 NB		20 NB		20 NB		20 NB		
19	Flange class	-		-		-		-		
20	Max strainer aperture microns	300				300				
21	MATERIALS									
22	Nozzle material	SS 304H		SS 304H		SS 304H		SS 304H		
23	Flange material	-		-		-		-		
24	Header material	-		-		-		-		
25	Header belt material	-		-		-		-		
26	Core material	SS 304H								
27	THERMAL DESIGN									
28	Design heat load kW	0.8		LMTD °C		20.2				
29	Corrected TD °C	20.2		Fouling allowance %		10				
30	Overall htc, clean W/m²K	53		Oversizing %		1				
31	Area provided m²	0.83		Area required, clean m²		0.75				
32	MASS & SIZE									
33	Mass: dry/operating/test kg					30 / 30 / 31		[2]		
34	Core dimensions mm					76 x 996 x 55		[2]		
35	<div><div><div>996</div><div>A in</div><div>B out</div><div>B in</div><div>A out</div><div>276</div></div><div><div>A in</div><div>B in</div><div>55</div></div></div>									

Figure 3.5: PCHE Data Sheet

3.3 Gas Booster

A gas booster that includes a 4AAD-2 Haskel Air Amplifier is used to both pressurize the air coming from the air supply and circulate it in the loop. This is an air driven amplifier. Since the air entering the amplifier is already pressurized to a certain level, the force needed to drive the amplifier decreases since that inlet pressure provides a substantial portion of the driving force and therefore the amplifier requires less air to drive the amplifier. The pressure of the air required for the cycle, as seen from Table 3.1, is 10 bars with a mass flow rate of 4.14 kg/h. Therefore, a release valve is fitted to the amplifier which is activated when the pressure reaches 15 bars. The release valve will reduce the pressure to 7 bars and the amplifier starts repressurizing. A pressure gauge fitted to the amplifier allows us to read the pressure of the air being pressurized. A pressure regulator is located to the outlet of the booster fixed at desired pressure needed for the system.

Figure 3.6: Gas Booster

3.4 Heater

In order to heat the air before entering the PCHE, a Watlow immersion heater is added to the loop. The heater is rated at 240V and 3000W and is used to heat the air from a temperature of 25°C to a maximum temperature of 150°C at 10 bar. Given that our desired temperature for the design point is 650°C, this heater will not allow our loop to reach the design point; however, the heater will enable the examination of off-design performance characteristics of the PCHE.

Figure 3.7: Watlow Immersion Heater

3.5 Instrumentation

3.5.1 Pressure

Given that the pressures needed in the loop need to remain constant at the desired pressure during operation, the loop was fitted with pressure gauges and regulators. The loop is fitted with a total of 4 MG 25 Swagelok pressure gauges, three for the air stream and two for the CO_2 stream, with a range of 0 - 25 bars with an accuracy of $\pm 2.5\%$. For the air stream, one pressure gauge was fitted to the air amplifier to measure the pressure of the air being pressurized, the next pressure gauge is located right after the gas booster and the final gauge is located after exiting the heater right before entering the PCHE. For the CO_2 stream, the first gauge is located after the valve and second gauge is located after the mass flow meter right before entering the PCHE. The Swagelok pressure gauge is shown in Figure 3.8.

Figure 3.8: Swagelok Pressure Gauge.

To measure the pressure drop across the heat exchanger, two ABB 266MSH differential pressure transmitters, with an accuracy of $\pm 0.06\%$ are used. They are connected to the inlet and outlet of both the air stream and the CO_2 stream. The value obtained represents the difference between the inlet pressure and the outlet pressure and hence the pressure drop across the PCHE. The pressure transmitter can be seen in Figure 3.9.

Figure 3.9: ABB 266MSH Differential Pressure Transmitter .

The pressure is regulated at two locations in the loop. The first location is the exit of the gas booster for the air to circulate at a regulated pressure and the second location is after the valve in the CO_2 stream to regulate the pressure of the CO_2 in that stream. The pressure regulators are Swagelok regulators fitted with MG 25 Pressure gauges and can be seen in Figure 3.10.

Figure 3.10: Swagelok Pressure Regulator for Air and CO_2

3.5.2 Temperature

The loop contains analog temperature gauges as well as Thermocouples. A total of five Swagelok dampened movement temperature gauges, with an accuracy of $\pm 1\%$, are located around the loop, three for the CO_2 stream and two for the air stream. For the CO_2 stream the first temperature gauge is located right after the valve, whereas the second is located after the pressure regulator and the third is located after the mass flow meter before entering the PCHE. For the air stream, the first gauge is located after the exit of the gas booster and the second is located after the air exits the heater before entering the PCHE. The Swagelok temperature gauge can be seen

in Figure 3.11.

Figure 3.11: Swagelok Temperature Gauge

A total of four J-Type 1/8" thermocouples are located in the loop. Two at the inlets of the PCHE and two at the outlets of the PCHE. The use of thermocouples is important at such a vital location to measure the inlet and outlet temperatures of the heat exchanger to the highest of accuracies. Figure 3.12 shows a 6 in long J-Type thermocouple.

Figure 3.12: 6 in J-Type Thermocouple

3.5.3 Mass Flow

Two Alicat Scientific MC Mass Flow Meters, with an accuracy of $\pm 0.2\%$, are located in the loop. One for the CO_2 stream, located after the second temperature gauge and the second is for the air stream located after the first temperature gauge. Figure 3.13 shows the mass flow meter.

Figure 3.13: Alicat Scientific MC Mass Flow Meter

3.5.4 Data Acquisition (DAQ)

The digital data recorded from this loop include pressure values from the differential pressure transducers, mass flow rate from the mass flow controllers and temperature values from the thermocouples. A Graphtec midi Logger GL820 DAQ system, with an accuracy of 0.1%, was used where the data was collected every 20 ms. The advantage of using this type of DAQ is that the data is observed on the screen and does not need a computer and a corresponding software to use. The power supply was controlled through a panel found next to the DAQ system. The DAQ can be seen in Figure 3.14.

Figure 3.14: Data Acquisition System

Chapter 4

Numerical Model

4.1 Introduction

The computational work in this thesis was performed with the commercially available software COMSOL Multiphysics to simulate fluid flow and heat transfer inside the PCHE. COMSOL is a finite element solver designed to provide and solve partial differential equations as algebraic equations. The domain is discretized into a set of control volumes where the equations of conservation of mass, energy and momentum are solved.

4.2 Theoretical Background

4.2.1 Conservation of Mass

The general conservation of mass equation used by COMSOL is

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho u) = 0 \quad (4.1)$$

where ρ is the density, t is the time and u is velocity

4.2.2 Conservation of Momentum

The Navier-Stokes equation can be seen as Newtons second law of motion for fluids as it governs the motion of the fluid. The equation is represented as

$$\rho \left(\frac{\partial u}{\partial t} + u \cdot \nabla u \right) = -\nabla p + \nabla \cdot (\mu(\nabla u + (\nabla u)^T)) - \frac{2}{3}\mu(\nabla \cdot u)I + F \quad (4.2)$$

where μ is the dynamic viscosity.

The Navier-Stokes equation comprises of four different forces where $\rho(\frac{\partial u}{\partial t} + u \cdot \nabla u)$ represents the inertial forces, $-\nabla p$ represents the pressure forces, $\nabla \cdot (\mu(\nabla u + (\nabla u)^T)) - \frac{2}{3}\mu(\nabla \cdot u)I$ represents viscous forces and F represents the external forces applied to the fluid. Given that the current conditions are steady state with no external forces present, the term F equals zero.

4.2.3 Conservation of Energy

The first law of thermodynamics or commonly known as conservation of energy is the law governing all heat transfer problems. The equation is represented as

$$\rho C_p \left(\frac{\partial T}{\partial t} + (u \cdot \nabla)T \right) = -(\nabla \cdot q) + \tau : S - \frac{T}{\rho} \frac{\partial \rho}{\partial T} \bigg|_p \left(\frac{\partial p}{\partial t} + (u \cdot \nabla)p \right) + Q \quad (4.3)$$

where C_p is the specific heat capacity at constant pressure, T is absolute temperature, q is the heat flux by conduction, τ is the viscous stress tensor, S is the

strain-rate tensor which is given by the equation

$$S = \frac{1}{2}(\nabla u + (\nabla u)^T) \quad (4.4)$$

and Q contains heat sources other than viscous heating.

The right side of the equation is comprised of four main terms. The first term, $-(\nabla \cdot q)$, represents conductive heat transfer, that is solved using Fouriers law of conduction which is represented as

$$q_i = - \sum_j k_{ij} \frac{\partial T}{\partial x_j} \quad (4.5)$$

where k is the thermal conductivity. The second term, $\tau : S$, represents viscous heating of a fluid. The operation $\cdot$ is a contraction which when in its expanded form is

$$a : b = \sum_n \sum_m a_{nm} b_{nm} \quad (4.6)$$

The third term, $-\frac{T}{\rho} \frac{\partial \rho}{\partial T} \Big|_p \left(\frac{\partial p}{\partial t} + (u \cdot \nabla)p \right)$, represents pressure work which is responsible for the fluid heating under adiabatic compression. Generally, this term is small for low Mach number flows. When inserting Equation 4.1 into 4.3 and ignoring both viscous heating and pressure work we obtain the more familiar equation

$$\rho C_p \frac{\partial T}{\partial t} + \rho C_p u \cdot \nabla T = \nabla \cdot (k \nabla T) + Q \quad (4.7)$$

4.3 PCHE Dimensions

As mentioned in section 2.2, the details of the inside geometry of the PCHE is usually not provided by Heatric due to it being proprietary information which makes it difficult to correctly model the PCHE. When the PCHE was purchased, Heatric provided a data sheet, Figure 4.1, which highlights the operating conditions of the PCHE.

SERVICE PER ITEM		SIDE A				SIDE B				Rev
1	FLUID CONDITIONS	In	Out	In	Out	In	Out	In	Out	
2	Fluid	Air				CO2				
3	Flow rate, total	4.14				4.69				
4	Flow rate, gas/vapour	4.14	4.14	4.69	4.69	4.69	4.69	4.69	4.69	
5	Flow rate, liquid	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	
6	Temperature	650.0	5.0	2.0	585.0	2.0	585.0	2.0	585.0	
7	Design temp: max/min	800 / 0				800 / 0				
8	Pressure: inlet/design	0.0 / 10.0				8.1 / 10.0				
9	Pressure drop: calc/allwd	0.05 / 0.05				0.00 / 0.05				
10	FLUID PROPERTIES	Liq	Vap	Liq	Vap	Liq	Vap	Liq	Vap	
11	Density	-	0.4	-	1.3	-	18.9	-	5.6	
12	Specific heat	-	1133	-	1007	-	918	-	1178	
13	Viscosity	-	0.0450	-	0.0179	-	0.0137	-	0.0409	
14	Thermal conductivity	-	0.0630	-	0.0245	-	0.0159	-	0.0592	
15	Dew point									
16	CONNECTIONS									
17	No. of nozzles	1	1	1	1	1	1	1	1	
18	Nozzle size	20 NB	20 NB	20 NB	20 NB	20 NB	20 NB	20 NB	20 NB	
19	Flange class	-	-	-	-	-	-	-	-	
20	Max strainer aperture	300	300	300	300	300	300	300	300	
21	MATERIALS									
22	Nozzle material	SS 304H	SS 304H	SS 304H	SS 304H	SS 304H	SS 304H	SS 304H	SS 304H	
23	Flange material	-	-	-	-	-	-	-	-	
24	Header material	-	-	-	-	-	-	-	-	
25	Header belt material	-	-	-	-	-	-	-	-	
26	Core material	SS 304H				SS 304H				
27	THERMAL DESIGN									
28	Design heat load	0.8	LMTD	20.2	20.2	20.2	20.2	20.2	20.2	
29	Corrected TD	20.2	Fouling allowance	10	10	10	10	10	10	
30	Overall htc, clean	53	Oversizing	1	1	1	1	1	1	
31	Area provided	0.83	Area required, clean	0.75	0.75	0.75	0.75	0.75	0.75	
32	MASS & SIZE									
33	Mass: dry/operating/test	kg	30 / 30 / 31	2	2	2	2	2	2	
34	Core dimensions	mm	76 x 996 x 55	2	2	2	2	2	2	
35										

Figure 4.1: PCHE Data Sheet

In terms of geometry, the data sheet only provides outside core dimensions which are $76 \times 996 \times 55$ mm and heat transfer area provided which is 0.83 m^2 , with no mention to any inside data such as channel shape, channel size, number of channels or channel zigzag angles. However, after contacting Heatric we were able to obtain this data, shown in Table 4.1, which helped us in creating the model.

Table 4.1: PCHE details provided by Heatric

	<i>Air</i>	<i>CO₂</i>
Channel Shape	Semi Elliptical	Semi Elliptical
Number of Channels	180	180
Number of Plates	10	10
Zigzag Angle	26°	26°
Channel Width (mm)	2.02	2.02
Hydraulic Diameter (mm)	1.09	1.09

From the data in Table 4.1, we were able to calculate the dimensions inside the PCHE.

The number of channels per plate is

$$\frac{\# \text{ of channels}}{\# \text{ of plates}} = \frac{360}{20} = 18 \text{ channels/plate} \quad (4.8)$$

The plate thickness is

$$\frac{PCHE \text{ thickness}}{\# \text{ of plates}} = \frac{55}{20} = 2.75 \text{ mm} \quad (4.9)$$

Total channel width per plate is

$$\text{Channel width} \times \# \text{ of channels per plate} = 2.02 \times 18 = 36.36 \text{ mm} \quad (4.10)$$

Total metal width per plate is

$$\text{Plate width} - \text{Total channel width per plate} = 76 - 36.36 = 39.64 \text{ mm} \quad (4.11)$$

Metal width per channel is

$$\frac{\text{Total metal width per plate}}{\# \text{ of channels per plate}} = \frac{39.64}{18} = 2.2 \text{ mm} \quad (4.12)$$

In order to simplify the model, the channel shape was assumed to be a semi-circle which has approximately the same hydraulic diameter as the semi- ellipses. The hydraulic diameter of a semi-circle in relation to the actual diameter is defined as

$$D_h = \frac{4A}{P} = \frac{4 \times 0.5 \times \frac{\pi D^2}{4}}{D + \frac{\pi D}{4}} \quad (4.13)$$

where D_h is the hydraulic diameter, A is the cross sectional area and P is the wetted perimeter.

Therefore, the diameter of our semicircular channel is

$$D_h = \frac{D_h(\pi + 2)}{\pi} = \frac{1.09(\pi + 2)}{\pi} \approx 1.8 \text{ mm} \quad (4.14)$$

The zigzagged channels in the PCHE play a vital role in its design. The zigzags allow for a larger area of heat transfer, since the travel length of the fluid increases, while remaining compact, decreasing the overall volume and footprint of the PCHE. The travel length is the length which is travelled by the working fluid inside the heat exchanger. This length is directly related to the overall length of the PCHE as well as the zigzag angles. Figure 4.2 shows a simplified diagram of the relationship between the travel length, the PCHE core length and the zigzag angles of the channels.

Figure 4.2: Simplified shape of PCHE channel [19]

The half bending angle, θ , is defined as the sine inverse of the core length divided by the travel length. Since we are given angle x as 26° , the half bending angle is

$$\theta = \frac{180 - (x \times 2)}{2} = \frac{180 - (26 \times 2)}{2} = 64^\circ \quad (4.15)$$

Therefore, the travel length is

$$Travel\ Length = \frac{PCHE\ Core\ Length}{\sin(Half\ Bending\ Angle)} = \frac{996}{\sin 64} = 1108.2\ mm \quad (4.16)$$

4.4 COMSOL Model

After obtaining the needed dimensions, a model for the PCHE was developed in COMSOL Multiphysics. As seen in Figure 4.3, all inlets and outlets of the PCHE are located on the sides showing a pure counter flow heat exchanger both horizontally and vertically. The plates of the PCHE are stacked in a way to have one hot plate per cold plate and given that the number of channels for the hot flow is equal to the number of channels for the cold flow, it is assumed that the channels are aligned directly one on top of the other. The solid domain of the model was modelled as stainless steel 304 and was assumed to have the same properties as the material of the actual PCHE, stainless steel 304H. whereas the fluid domains were modelled as air for the hot flow and CO_2 for the cold flow.

Figure 4.3: Uninsulated PCHE showing the inlets and outlets

4.4.1 Model Assumptions

As mentioned in section 4.3, the heat exchanger has a total of 180 channels. Modelling all these channels is computationally very expensive. Therefore, some assumptions and simplifications have been applied to the model to make it easier and faster to simulate.

The channels were assumed to be semicircular for ease of modelling and simulation. The hydraulic diameter of the channel of 1.09 mm is equal to that of the semi ellipses channel giving us a diameter of 1.8 mm for the semicircle.

A main simplification that was made was modelling only one column of the heat exchanger with a total of 20 channels instead of the whole PCHE with a total of 180 channels. As discussed by Figley [11], the results for modelling one column of the heat exchanger falls within 1% error when compared to modelling the whole heat exchanger. This has been tested and confirmed when modelling the whole PCHE, Figure 4.4, and when comparing the results to one column of the PCHE, Figure 4.5, where both had the same inputs.

Figure 4.4: COMSOL Model of the whole PCHE

Figure 4.5: COMSOL Model of one column of PCHE (left) and an enlargement of model channels (right)

The zigzag angles have not been implemented in this model, instead, the travel length calculated from the zigzag angles was used to model the straight channels. Since the total travel length has been used, the total heat transfer area of the PCHE has been accounted for which allows us to simulate the heat transfer behavior while decreasing computational time. A length of 1108.2 mm was used for the travel length as calculated in section 4.3.

4.4.2 Boundary Conditions

To solve this heat transfer problem, two main physics were used in COMSOL to set the boundary conditions. One is the Fluid Flow physics which dictates the fluids behavior such as turbulence regimes and flow rates and the other was the Heat Transfer physics where inlet temperatures of the flows were set. Both physics were coupled in COMSOL using a Non-Isothermal Flow Multiphysics coupling, which allows both physics to exchange information to calculate a solution. The Non-Isothermal Flow coupling allows for the change in temperature of the flow which in turn corresponds to a change in other material properties such as density and viscosity.

Fluid Flow

The fluid domain has been modelled as air for the hot flow and CO_2 for the cold flow. Since this is a counter flow heat exchanger, the inlet boundary conditions for each flow have been placed on opposite sides of the heat exchanger. The inlets of the fluid domain have been set as the pressure of the flow, in bars, whereas the outlets have been set as the mass flow rate, in kg/h, with suppressed backflow.

Satish [32] discusses the definition of a micro channel as having a maximum hydraulic diameter of 1 mm. Given that our hydraulic diameter is 1.09 mm, the turbulence regimes would not behave the same in the PCHE channels as they do in conventional sized pipes. As discussed in section 2.3, the flow can be observed to be turbulent with a Reynolds number as low as 400 in microchannels. Therefore, a turbulence model has been added to the fluid domain which simulates turbulence in the flow. The k -turbulence model was chosen which uses equations for the turbulent kinetic energy (k) and for turbulent dissipation (ϵ) thereby predicting the behavior of the fluids turbulence. In case the flow turns out to be laminar, COMSOL's turbulent flow physics successfully solves problems of flow in the laminar regime.

Finally, for ease of computation, the model was cut vertically in half and a symmetry boundary condition was added to the fluid domain.

Heat Transfer

The heat transfer physics is applied to both the solid and fluid domain. The steel portion of the model was set as adiabatic given that the PCHE is insulated and almost no heat is lost to the surrounding. Changes in conductivity, density and heat capacity of the steel occur based on functions present in COMSOLs material library [36]. As for the fluid domain, inlet temperatures for air and CO_2 have been applied as these are the conditions that are controlled experimentally. An outflow boundary condition has been added to the outlets of the fluid domain, to give COMSOL a direction for the heat flux in the axial direction. As was done with the Fluid Flow physics the model was cut vertically in half for ease of simulation and a symmetry

boundary condition was added to both the solid and fluid domain.

4.5 Grid Analysis

4.5.1 Mesh Creation

To simulate the PCHE, a mesh was created using COMSOL's mesh generator. In order to mesh the PCHE, a 2-D mesh is first created on one surface and then swept across the entire geometry creating a 3-D volume mesh. For 2-D meshes, the generator allows to use either triangular or quadrilateral shaped meshes. Given that the PCHE is essentially a fluid dynamics problem, a structured quadrilateral mesh has been chosen to solve the simulation.

There are two main locations where the mesh resolution is of great importance. The first location is at the boundary between the fluid and the solid domain. The mesh density must be high enough near the walls to accurately capture thermal boundary layers and hence capturing the temperature gradients as well as resolving the laminar sub-layer along the wall. To achieve that high density, the boundary layer tool in COMSOL's mesh generator is used. A 5-layer boundary was created along with a boundary layer stretching factor of 1.1, as seen in figure 4.6.

The second location is at the inlets and outlets of the channels. The inlet region experiences a rapidly changing flow behavior, given that the flow's velocity and temperature profiles as well as the boundary layers are developing. In order to make the mesh finer in the axial direction, to accurately capture these changing behaviors, the geometric sequence distribution tool was used when sweeping the 2-D mesh across the entire geometry. This tool allows for the concentration of a certain number of

elements at both ends of the PCHE and then expanding as they leave the inlet region.

Figure 4.7 shows the mesh distribution at the inlet of one channel.

Figure 4.6: Quadrilateral surface mesh with boundary layers

Figure 4.7: Quadrilateral surface mesh with boundary layers

4.5.2 Mesh Sensitivity Analysis

In order to choose the right number of mesh elements, a mesh sensitivity analysis was conducted on one of the tests while observing the change in the CO_2 outlet temperature. The test chosen had an air inlet temperature of $70^\circ C$, CO_2 inlet temperature of $16.9^\circ C$, air mass flow rate of 0.0431 kg/h per channel, CO_2 mass flow rate of 0.0726 kg/h per channel and both flows were set at 10 bars. Table 4.2 shows the outlet temperature of CO_2 when simulating the model starting at a mesh of 42,100 elements and going up to 1,609,100 elements whereas Figure 4.8 shows the trend of mesh elements vs temperature.

Table 4.2: CO_2 Outlet Temperature with Varying Number of Elements

	<i>Number of Elements</i>	<i>CO_2 Outlet Temperature $^{\circ}C$</i>	<i>Computation Time</i>
Mesh 1	42,100	46.82	12 min 54 s
Mesh 2	69,700	47.11	13 min 12 s
Mesh 3	148,600	47.51	17 min 1
Mesh 4	174,400	47.62	18 min 59 s
Mesh 5	301,200	47.72	24 min 27 s
Mesh 6	737,450	47.68	49 min 21 s
Mesh 7	1,609,100	47.69	2 h 1 min 14 s

Figure 4.8: Number of Mesh Elements vs Temperature

The results from Table 4.2 and Figure 4.8 show that the change in the outlet temperature of CO_2 plateaus at around 300,000 elements where there is a $0.03^\circ C$ difference between Mesh 5, generating 301,200 elements, and Mesh 7, generating 1,609,100 elements. Mesh 5 was therefore used to solve the rest of the tests which, when compared to Mesh 7, provides an accurate result with a relatively low computational time. Mesh 5 contains 107,608 quadrilateral elements, 10,904 edge elements, 68 vertex elements and a mesh volume of $61,970 \text{ mm}^3$.

Comparison With Literature

The mesh quality for the COMSOL model was compared to the mesh quality created by several researchers to confirm that the mesh distribution for the COMSOL model was adequate to obtain accurate results. For researchers such as Kar [16], Figley [17], Li et al. [33] and Meshram et al. [34], the mesh density at the inlet and outlets of the channel was very high as seen in Figures 4.9, 4.10 and 4.11. The reason they increased the density at the surface of the channels is to investigate the temperature profiles and gradients occurring at the inlet and outlet surfaces and hence making the mesh extremely fine at these surfaces while also increasing the mesh quality of the boundary elements.

Figure 4.9: Mesh quality for Meshram et al. Model [34]

Figure 4.10: Mesh quality for Li et al. Model [33]

Figure 4.11: Mesh quality for Figley Model [17]

However, when looking at researchers such as Kim et al. [22] and Lee et al. [35], their mesh quality is not as fine as the ones previously mentioned. The investigation of their research covered overall outlet temperature and pressure drop investigations and not the temperature profiles at the surfaces and hence the mesh they used was of lesser quality as seen in Figure 4.12. Given that our main investigative purposes is to examine the overall temperature behavior of the fluid as it exits the channel, the mesh created to fulfill this goal is not as fine as those used to investigate the temperature profiles on the channel surfaces however, it is fine enough to obtain the results we need with good accuracy and much less computational time.

Figure 4.12: Mesh quality for Kim et al. Model [22]

4.6 Solver

COMSOL uses numerical solution methods based on partial differential equations which allow for the representation of the problem as a system of algebraic equations. Linear algebraic equations are set in the form of $Au = f$ where u is the vector solution. Once A and f have been determined, u is calculated and a solution is produced. These linear algebraic equations are solved using two methods, the direct method and the iterative method. The direct method finds an approximate solution for u through matrix factorization where a number of operations take place based on the number of unknowns. Once all operations are executed, the solution, u , is obtained. The iterative method starts with an approximate initial guess and then proceeds to improve the guess by performing iterations. Unlike the direct method,

the iterations can be stopped at any residual error and a solution u would be available. However, if the iterations have been stopped too early, that could result in a solution with poor accuracy.

The direct method is usually computationally expensive for large 3-D applications where as the iterative method has a lower memory consumption and for large 3-D applications are better to obtain a solution. However, the iterative method is more challenging when trying to solve matrices arising from multiphysics problems. For that reason, the direct method has been used to simulate the PCHE model.

When simulating the PCHE model, the direct solver uses a segregated approach, where the main physics controlling the behavior of the solution are segregated and solved separately. This is usually used when solving turbulent flow problems in order to stabilize the solution process. The solver separates the problem into heat transfer, which solves for temperature profiles in the fluid and the solid domains, fluid flow, which solves for the velocity field and pressure distribution in the channels and the turbulent kinetic energy and dissipation rate from the $k - \epsilon$ model.

The direct method uses the Parallel Direct Solver (PARDISO) in order to obtain a solution. PARDISO uses LU factorization which allows for solving of systems of linear equations.

Chapter 5

Results and Discussion

5.1 Introduction

A set of tests have been conducted on the PCHE with a range of temperatures, pressures and flow rates to observe the thermodynamic behavioral changes of varying the operating conditions of the PCHE. Ideally, the tests conducted would be as close to the design point as possible, however the number of tests and ranges have been dictated by the limitations of the heater. The purchased Watlow heater can only withstand a maximum of 150°C at a pressure of 10 bars and when operating at a relatively low flow rate, the maximum temperature cannot be achieved. For that reason, a total of 18 tests have been carried out by varying the inlet temperature of air from 70°C to 100°C to 140°C . The CO_2 temperature entering the PCHE depends on the temperature of the CO_2 present in the tank since currently there is no temperature control for the CO_2 entering the PCHE. The flow has been alternated from 5 Liters Per Minute (LPM) to 10 LPM and the pressure has been alternated as well from 5 bars to 10 bars. Table 5.1 shows the test specifications. Test set

A combines the tests that have air and CO_2 flow of 10 LPM, test set B combines the tests that have air and CO_2 flow of 5 and 10 LPM respectively and test set C combines the tests that have air and CO_2 flow of 5 LPM.

Table 5.1: Test Specifications

Test Set	Test Number	Air Inlet Temp ($^{\circ}C$)	Air and CO2 Pressure (bars)	Air Flow Rate (LPM)	CO2 Flow Rate (LPM)
A	1	140	10	10	10
	2	100	10	10	10
	3	70	10	10	10
	4	140	5	10	10
	5	100	5	10	10
	6	70	5	10	10
B	7	140	10	5	10
	8	100	10	5	10
	9	70	10	5	10
	10	140	5	5	10
	11	100	5	5	10
	12	70	5	5	10
C	13	140	10	5	5
	14	100	10	5	5
	15	70	10	5	5
	16	140	5	5	5
	17	100	5	5	5
	18	70	5	5	5

Attempts were made to obtain data at 1 LPM for both air and CO_2 but each came with its own set of difficulties that did not allow us to obtain that data. When air is at 1 LPM, the temperature of the air cannot reach the temperatures desired even though the heater is set at a high temperature. After contacting Watlow, they advised against operating at such conditions for the fear of damaging the heating element in the heater. When CO_2 is at 1 LPM, it takes an estimated average of approximately 14 hours to reach steady state, and due to the presence of no fail safes on the heater and having limited access to the experimental facility, CanmetENERGY advised against leaving the loop running overnight to reach steady state. The same thing happened when trying to conduct tests at an air flow rate of 10 LPM and CO_2 flow rate of 5 LPM. An estimated time of 12 hours was needed to reach steady state. Therefore, we were limited with the temperatures, pressures and flow rates for operation, however we were able to obtain the data from the tests in Table 5.1 and analyze them to obtain important trends till at a later time when the loop is redesigned to fix the mentioned issues.

5.2 Data Analysis

REFPROP

REference Fluid PROPERTIES (REFPROP) was used to obtain the properties for both air and CO_2 to be used in any calculations that require those properties. REFPROP is a database of real fluid properties developed by the National Institute for Standards and Technology (NIST). REFPROP includes the most accurate data and

equations to calculate thermodynamic and transport properties of real fluids. The state equations used in REFPROP are based on a large number of correlations for different fluids which are able to capture the changes in fluid properties over a wide range of state points. REFPROP can accurately call a fluids property data after being inputted with two different properties such as temperature and pressure or entropy and enthalpy.

Fluid Flow Conversion

In order to conduct our calculations as well as input the flow rate values into COMSOL, the units for the flow needed to be converted from volumetric flow rate to mass flow rate. The Alicat flow meters read values for flow in LPM, temperature in $^{\circ}\text{C}$ and pressure in psia. The temperature and pressure obtained from the flow meter are then inputted into REFPROP to obtain the density which is then multiplied by the volumetric flow rate to obtain the mass flow rate.

Heat Rate Calculation

It takes an average of approximately 3 - 4 hours to reach steady state with each test in Table 5.1. Because of the slow temperature changes, it is difficult to know when the flow has reached steady state and therefore another approach to find that out is to calculate the heat rate for air and CO_2 and compare the values. In steady state, the heat rate value released by air should be equal to the heat rate received by CO_2 assuming an adiabatic system.

After converting the volumetric flow rate to mass flow rate in units of kg/s,

we obtain the inlet and outlet temperatures from the DAQ for air and CO_2 . The temperature values as well as the pressure of the flow are inputted in REFPROP to obtain the enthalpy values of the inlets and outlets of the PCHE. After using the equation

$$Q = \dot{m}\Delta H \quad (5.1)$$

where Q is the heat rate, $\dot{m}$ is the mass flow rate and ΔH is the difference in enthalpy, the values of Q for both air and CO_2 are obtained and subtracted from each other to calculate the difference, which in steady state is supposed to equal zero. Due to uncertainties that arise from instrumentation errors, the difference in heat rates will not equal zero, however the values of the heat rates are acceptable if they lie within the error calculated as will be shown in section 5.4.

5.3 Pressure Drop

The pressure drop values across the PCHE were recorded using the differential pressure transmitters. After calculating the Reynolds number of the tests, a plot highlighting the trend of the pressure drop with varying Reynolds number for both air and CO_2 was plotted for pressure at 5 bar and pressure 10 bar as seen in Figure 5.1 and 5.2. The plots are obtained from data points where air and CO_2 have a flow rate of 1, 5 and 10 LPM.

Table 5.2: Reynold's Number and Pressure Drop values for CO_2

Volumetric Flow Rate (LPM)	Reynold's Number	Pressure Drop (kPa)
10 Bars		
10	1632	1.88
5	786	0.61
1	215	0.06
5 Bars		
10	841	1.1
5	420	0.41
1	90	0.03

Table 5.3: Reynold's Number and Pressure Drop values for air

Volumetric Flow Rate (LPM)	Reynold's Number	Pressure Drop (kPa)
10 Bars		
10	648	1.48
5	316	0.59
1	70	0.15
5 Bars		
10	321	1.03
5	185	0.46
1	50	0.15

Figure 5.1: Reynold's Number vs Pressure Drop for Air

Figure 5.2: Reynold's Number vs Pressure Drop for CO_2

As expected, both trends show that with a higher flow velocity, the higher the pressure losses are. We can also observe that for the same volumetric flow, the pressure drop is higher for the 10 bar flow when compared to that of the 5 bar flow. These pressure drops occur due to both, the roughness of the channel from chemically etching the steel plate and the zigzag angles in the channels.

5.4 Heat Rate

As mentioned in section 5.2, the heat rate calculations for all the tests have been calculated and are found in Table 5.4 whereas Figures 5.3, 5.4 and 5.5 show the trends of all three data sets with varying air inlet temperatures. As seen in Table 5.4, when comparing the heat released by air and the heat absorbed by CO_2 , the values are approximately equal and within the errors provided for each calculation. This gives us confidence that each test has successfully reached steady state. The error calculated is based on the measurement uncertainties of the instruments used to gather the experimental data. A sample of the error calculation can be found in Appendix A.

Table 5.4: Calculated Heat Rate

Test Number	Heat Released by Air (W)	Heat Absorbed by CO_2 (W)
1	259.9 ± 6.94	247.75 ± 7.94
2	157.7 ± 6.49	162.8 ± 7.66
3	114.24 ± 6.54	116.99 ± 7.57
4	123.86 ± 7.02	114.1 ± 8.76
5	84.3 ± 3.57	77.4 ± 4.45
6	56.2 ± 3.65	51.98 ± 4.57
7	131.4 ± 3.5	134.83 ± 7.87
8	78.5 ± 3.4	94.4 ± 10.02
9	50.93 ± 3.26	58.23 ± 7.23
10	74.69 ± 2.08	79.9 ± 4.46
11	43.77 ± 2.08	50.8 ± 4.37
12	24.9 ± 1.77	27.2 ± 4.5
13	117.8 ± 3.52	125.5 ± 5.06
14	92.32 ± 3.58	91.66 ± 5.21
15	48.6 ± 3.42	52.6 ± 3.7
16	58.9 ± 1.9	51 ± 2.3
17	45.9 ± 1.88	43.3 ± 2.22
18	26.4 ± 1.85	28.1 ± 2.17

Figure 5.3: Air Inlet Temperature vs. Heat Rate for Test Set A

Figure 5.4: Air Inlet Temperature vs. Heat Rate for Test Set B

Figure 5.5: Air Inlet Temperature vs. Heat Rate for Test Set C

As seen from figures 5.3 - 5.5, the heat rate increases as the air inlet temperature increases. Air and CO_2 heat rates appear to be parallel and within the error bars confirming steady state has been reached. The trends also show that doubling the pressure from 5 to 10 bar almost doubles the heat rate. The effect of pressure on the rate of heat transfer arises from its effect on the Nusselt number. The Nusselt number can be calculated through several correlations as seen from section 2.3 however, the same variables are used for all of them. Pressure drop affects the Reynold's number, the Prandtl number and the density of the flow. Nusselt number then changes based on the changes that occur in these three values as seen from one correlation of Nusselt number [23]

$$Nu = 0.0345 Re^{0.86} Pr^{0.23} \left(\frac{\rho_w}{\rho_b} \right)^{0.59} \quad (5.2)$$

where Nu is the Nusselt number, Re is the Reynold's number and Pr is the Prandtl number.

The change in Nusselt number in turn affects the heat transfer coefficient of the flow with the direct relationship of

$$h = \frac{Nu k}{L} \quad (5.3)$$

where h is the convective heat transfer coefficient, k is the thermal conductivity and L is the characteristic length. Finally, the convective heat transfer coefficient is directly proportional to the rate of heat transfer through

$$q = h(\Delta T) \quad (5.4)$$

5.5 Temperature

Table 5.5 shows the operating conditions of the all the tests with the measured outlet temperatures.

Table 5.5: Test Condition with Outlet Temperatures

Test Number	Air and CO_2 Pressure (bars)	Air Mass Flow Rate (kg/h)	Air Inlet Temp ($^{\circ}C$)	Air Outlet Temp ($^{\circ}C$)	CO_2 Mass Flow Rate (kg/h)	CO_2 Inlet Temp ($^{\circ}C$)	CO_2 Outlet Temp ($^{\circ}C$)
1	10	7.83	140.4	23.4	12.41	23.0	99.6
2	10	7.58	100.1	26.7	12.47	26.1	76.4
3	10	7.58	70.0	18.1	13.07	16.9	51.5
4	5	3.87	140.1	26.6	6.41	25.0	95.7
5	5	4.10	100.4	27.5	6.48	26.7	74.5
6	5	4.25	70.5	23.5	6.86	22.2	52.8
7	10	3.96	141.8	26.0	12.82	24.8	65.4
8	10	3.78	100.8	27.6	12.33	26.9	56.5
9	10	3.87	71.0	24.6	12.58	24.4	42.3
10	5	2.32	135	20.9	6.55	20.3	69.3
11	5	2.11	96.6	22.8	6.55	22.7	54.0
12	5	2.05	68.0	24.9	6.44	24.4	41.5
13	10	3.83	141.5	33.2	6.05	26.4	105.9
14	10	3.96	100.2	18	6.34	17.0	72.8
15	10	4.07	70.6	28.5	6.19	27.9	60.7
16	5	2.16	122.9	26.2	3.20	25.6	90.0
17	5	2.16	102.1	27.2	3.20	26.1	80.1
18	5	2.16	71.9	28.5	3.20	26.9	62.2

5.5.1 Experimental and Numerical Results

Inlet temperatures of air and CO_2 , pressures of the flows and their mass flow rate were inputted into COMSOL and the outlet temperatures are obtained. To compare the results from the COMSOL model to those obtained experimentally, the two central channels from the model were investigated as representatives of the total outlet temperatures. The simulation results were compared to the experimental results and a percentage difference was obtained through

$$\% \text{ Error} = \frac{|T_{sim} - T_{exp}|}{T_{exp}} \times 100\% \quad (5.5)$$

Table 5.6 shows the simulation results, the experimental results and their percentage difference.

Table 5.6: Comparison of Experimental and Simulation Results

Test Number	Air Texp (°C)	Air Tsim (°C)	ΔT (°C)	Error (%)	CO ₂ Texp (°C)	CO ₂ Tsim (°C)	ΔT (°C)	% Error
1	23.4	23.85	0.45	1.92	99.6	90.3	9.3	9.34
2	26.7	26.8	0.1	0.37	76.4	71	5.4	7.07
3	18.1	17.45	0.65	3.59	51.5	50.1	1.4	2.72
4	26.6	27.5	0.9	3.38	95.7	89.9	5.8	6.06
5	27.5	29.1	1.6	5.82	74.5	71.8	2.7	3.62
6	23.5	23.8	0.3	1.28	52.8	52.6	0.2	0.38
7	26	25.1	0.9	3.46	65.4	64	1.4	2.14
8	27.6	27.35	0.25	0.91	56.5	53.1	3.36	5.95
9	24.6	24.45	0.15	0.61	42.3	42	0.3	0.71
10	20.9	21	0.1	0.48	69.3	63	6.3	9.09
11	22.8	23	0.2	0.88	54	49.6	4.4	8.15
12	24.9	24.7	0.2	0.80	41.5	41.1	0.4	0.96
13	33.2	29.2	4	12.05	105.9	95.3	10.6	10.01
14	18	19.1	1.1	6.11	72.8	69	3.8	5.22
15	28.5	30	1.5	5.26	60.7	57.9	2.8	4.61
16	26.2	33	6.8	25.95	95	85	10	10.53
17	27.2	32.5	5.3	19.49	80.1	74	6.1	7.62
18	28.5	31.2	2.7	9.47	62.2	56.6	5.6	9.00

As seen from Table 5.6, the results show that outlet temperatures at a lower pressure are lower than those at a higher pressure however the difference is very small. For example, when comparing the CO_2 outlet temperatures of tests 2 and 5, which have the same inlet conditions with the exception of the pressure, we find that the outlet temperature for test 2 is $76.4^\circ C$ whereas test 5 has an outlet temperature of $74.5^\circ C$ even though test 5 had a higher inlet temperature by $0.8^\circ C$. Since test 5 had a lower heat rate because of the lower pressure, the heat absorbed by the CO_2 was not high enough to reach the temperature reached by test 2.

The results also show that the simulation results are in line with the experimental results with a maximum temperature difference between experimental and simulation of $10.6^\circ C$ found in Test 13. The experimental setup of the PCHE does not allow us to measure temperatures anywhere but the inlet and outlet, therefore we cannot obtain a temperature distribution from the experimental results and compare with those obtained through simulation. An example of the simulated temperature distribution along with the experimental outlet temperatures for Test 4 is found in Figure 5.6 with the rest of the trends found in Appendix B.

Figure 5.6: Test 4 Temperature Distribution

Figure 5.6 is translated into the fringe plot seen in Figure 5.7 which shows the two most central channels of the model with the maximum amount heat transfer occurring at the inlet of the hot channel. The steel walls act as an intermediary to regulate the heat exchange. Although there is heat being exchanged between the fluids, there is also an aspect of axial conduction within the steel which exists due to the temperature gradient. As expected, the temperature gradient along the walls is steeper than the gradients within the fluids due to this axial conduction.

Figure 5.7: Isometric View of Temperature Distribution in Central Channels

Temperature Distribution

Unfortunately, there is no way to verify the accuracy of the temperature distribution inside the PCHE. However, the simulated distribution makes physical sense when compared to a typical counter flow heat exchanger. This can be seen by the rapid air temperature change happening at the hot inlet, where the most temperature difference exists and hence most heat transfer occurs, and as the temperature difference decreases so does the change in the hot stream. We can also observe a slight curve at the cold inlet which represents the thermal development of the flow which is too small to be seen at the hot inlet.

5.5.2 Hot Inlet Temperature Gradient

The acute drop in temperature seen in Figure 5.6 at the hot inlet occurs due to the very slow flow rate of 3.87 kg/h which constitutes 0.0215 kg/h per channel which allows a lot of temperature change to occur in a very small distance. When increasing the air flow, it is observed that this gradient decreases, as seen in Figure 5.8, which shows the temperature distribution for Test 1. Test 1 has an air flow rate of 7.83 kg/h which constitutes 0.0435 kg/h per channel which is still very slow however it shows the change in gradient of the hot stream.

Figure 5.8: Test 1 Temperature Distribution

Sensitivity studies were carried out to see the effect of the flow rate on that gradient observed at the inlet of the hot stream to confirm the mentioned reason for such a behavior. The flow rate for both streams was multiplied by 2, 5 and 10 and the temperature distribution was observed as seen in Figure 5.9 and an enlargement of the inlet is found in Figure 5.10.

As seen from Figures 5.9 and 5.10, The sensitivity studies confirm that the gradient is dependent on how fast the flow travels in the channel as we see a decrease in the gradient as the flow increases. This was also confirmed from the literature where Vanmeter [20] was successful in plotting temperature distribution results for CO_2 and water with varying flow which is highlighted in Figures 5.11 and 5.12.

Figure 5.9: Effect of Flow Rate on Temperature Drop Gradient

Figure 5.10: Enlargement of the Inlet of "Effect of Flow Rate on Temperature Drop Gradient"

Figure 5.11: Temperature Profiles for Vanmeter with CO_2 at (a) 100 kg/h (top) and (b) 200 kg/h (bottom) [20]

Figure 5.12: Temperature Profiles for Vanmeter with CO_2 at (a) 300 kg/h (top) and (b) 400 kg/h (bottom) [20]

As observed from Figures 5.11 and 5.12, the temperature drop gradient at the inlet of the hot channel decreases as the flow increases. What is also observed from Figures 5.9, 5.11 and 5.12 is that as the flow increases, the outlet temperature of the flow increases as well. These two are explained by the same reason. Even though the heat rate increases as the flow increases which would give the assumption of a higher drop in the outlet temperature value however, the travel length allocated in the heat exchanger does not permit that high drop. Because the flow was increased but not the travel length, there is not enough time for the flow to drop the temperature to its maximum. Therefore, the gradient at the inlet decreases and the temperature of outlet increases.

5.5.3 Effect of Flow on Temperature Behavior

Test sets A, B and C were created based on the volumetric flow rate of air and CO_2 . Test set A has both air and CO_2 flowing at 10 LPM, test set B has air flowing at 5 LPM and CO_2 flowing at 10 LPM and finally test set C has air flowing at 5 LPM and CO_2 flowing at 5 LPM. The effects the different flow arrangements have on our results were examined and as expected, they were found to directly affect the temperature behaviors. To further understand those effects, temperature behaviors from tests 2, 8 and 14 were compared. All these tests have the same pressure and inlet temperatures however have different flows where test 2 belongs to test set A, test 8 belongs to test set B and test 14 belongs to test set C.

Figures 5.13, 5.14 and 5.15 show the temperature distributions of tests 2, 8 and 14 respectively.

Figure 5.13: Test 2 Temperature Distribution

Figure 5.14: Test 8 Temperature Distribution

Figure 5.15: Test 14 Temperature Distribution

Test 2

Test 2, belonging to test set *A*, has 10 LPM of flow for both air and CO_2 . The temperature distribution, found in Figure 5.13, shows an acute drop at the hot inlet and then the gradient of the temperature drop decreases till it reaches an outlet temperature of $26.8^\circ C$. At the cold inlet, the flow starts at $26.1^\circ C$, with a ΔT between the cold flow and hot flow of $0.7^\circ C$. This ΔT then increases almost linearly as the cold flow increases in temperature till it reaches its outlet where ΔT increases exponentially to $29.1^\circ C$.

Test 14

Test 14, belonging to test set *C*, has 5 LPM of flow for both air and CO_2 . Given that both air and CO_2 are at equal volumetric flow, it is expected that the temperature behavior of test 14 would behave similarly to test 2 with some slight differences. The similarities are found in the gradient of the temperature drop and increase of the hot and cold flow respectively. Both test 2 and test 14 have approximately the same

gradient of temperature change as seen when comparing Figures 5.13 and 5.15.

One of the main differences that can be observed is that the change in ΔT between the cold and hot flow for test 14 is lower than that of test 2. As seen in Figure 5.15, the space between the flows increases from right to left slower than the space between the flows in Figure 5.13. This occurs due to lower flow rate of test 14 which allows more time for heat to be exchanged even though its heat rate is lower than test 2. Therefore, more heat travels from air to the CO_2 which gives us a ΔT for CO_2 of 52 $^{\circ}C$ for test 14 and ΔT of 44.9 $^{\circ}C$ for test 2.

Test 8

Test 8, belonging to test set *B*, has 10 LPM flowing for CO_2 and 5 LPM flowing for air. As expected, the temperature distribution behaves differently than tests 2 and 14. Because air has a lower volumetric flow rate than CO_2 , the amount of heat air can provide CO_2 decreases which allows the CO_2 to have a ΔT of only 26.2 $^{\circ}C$. CO_2 , being at a higher flow rate, demands much more heat than air can provide and therefore, as seen from Figure 5.14, air loses almost all of its heat in the first 600 - 700 mm of the heat exchanger after which the temperature plateaus in the last 400 mm showing almost zero heat transfer.

5.5.4 Overall Heat Transfer Coefficient

The overall heat transfer coefficient plays a vital role in the modeling of closed loop Brayton cycles where it is used to predict the energy changes that occur in the heat exchangers and therefore provide more accurate performance data for the loop. In the data sheet provided by Heatric, information related to the overall heat transfer

coefficient was included as seen in Figure 5.16 whereas the design point values of the PCHE can be found in Table 5.7.

Design heat load	kW	0.8	LMTD	°C	20.2	
Corrected TD	°C	20.2	Fouling allowance	%	10	
Overall htc, clean	W/m ² K	53	Oversizing	%	1	
Area provided	m ²	0.83	Area required, clean	m ²	0.75	

Figure 5.16: PCHE Partial Specifications

Table 5.7: PCHE Design Point

	<i>Air</i>	<i>CO₂</i>
Inlet Temperature (°C)	650	2
Outlet Temperature (°C)	5	585
Pressure (barg)	10	10
Mass Flow Rate (kg/h)	4.14	4.69

To confirm the numbers provided, the overall heat transfer coefficient was calculated for the design point using the following equation

$$U = \frac{q}{\Delta T_{lm}} A \quad (5.6)$$

where U is the overall heat transfer coefficient, q is the design heat load provided by heatric, ΔT_{lm} is the log mean temperature difference (LMTD) and A is the area required.

The LMTD is a logarithmic average of the temperature difference between the hot and cold stream in a heat exchanger and is calculated using

$$\Delta T_{lm} = \frac{\Delta T_1 - \Delta T_2}{\ln \frac{\Delta T_1}{\Delta T_2}} \quad (5.7)$$

where $\Delta T_1 = T_{hotinlet} - T_{coldoutlet}$ and $\Delta T_2 = T_{hotoutlet} - T_{coldinlet}$. Therefore, when using the numbers for Table 5.7, we obtain

$$\Delta T_1 = 650 - 585 = 65 \text{ } ^\circ C \quad (5.8)$$

and

$$\Delta T_2 = 5 - 2 = 3 \text{ } ^\circ C \quad (5.9)$$

Therefore, when using equation 5.7 we obtain an LMTD of

$$\Delta T_{lm} = \frac{65 - 3}{\ln(65/3)} = 20.2 \text{ } ^\circ C \quad (5.10)$$

which is equivalent to the LMTD value provided by Heatric. Using the LMTD, the heat load provided and the area, we use equation 5.6 to obtain the overall heat transfer coefficient of the design point

$$U = \frac{800}{20.2}(0.75) = 52.8 \text{ } W/m^2 K \approx 53 \text{ } W/m^2 K \quad (5.11)$$

which is equivalent to the overall heat transfer coefficient provided by Heatric. Therefore using the same equations, values for the overall heat transfer coefficients for the tests were obtain and are found in Table 5.8.

Table 5.8: Overall Heat Transfer Coefficients

Test Number	Overall Heat Transfer Coefficient, U, ($\text{W}/\text{m}^2\text{K}$)
1	39.7
2	34.5
3	24.7
4	12.8
5	15.6
6	11.9
7	9.9
8	10.1
9	13.5
10	7.7
11	9.6
12	5.5
13	9.6
14	15.4
15	21.1
16	8.4
17	8.8
18	8.3

After obtaining more data points through increasing the range of operation of the loop, more data pertaining to the overall heat transfer coefficient can be obtained and used in any off design modeling of the Brayton cycle loops.

5.5.5 Design Point Simulation

Table 5.7 shows the design point of the PCHE. These conditions were inputted into COMSOL and simulated results for the temperature behavior can be found in Figure 5.17.

Figure 5.17: Design Point Simulated Temperature Distribution

As seen from Figure 5.17, the outlet temperature for CO_2 is $273.6\text{ }^{\circ}C$ which is significantly lower than the design point outlet temperature of $585\text{ }^{\circ}C$ giving us an error of 53.2% and a ΔT of $311.4\text{ }^{\circ}C$.

It has been observed that with a higher air inlet temperature, the error between the simulated and experimental results increases. Therefore, a plot highlighting the error trend was plotted and can be found in Figure 5.18. Unfortunately, at the moment only four data points can be used for the trend which are the three air inlet temperatures used in the tests performed and the design point. After obtaining more data in the future, this plot can be modified to include more simulated data which would allow us to fine tune the COMSOL model to take into consideration the error increase that accompanies an input temperature increase.

Figure 5.18: Temperature Difference Trend with Variation of Air Inlet Temperature

The difference observed between the simulated results as well as the experimental results may be due to the assumptions made in the model and due to the fact that not all the geometric definitions of the PCHE are known. Simulation of the horizontal counter flow at the inlets as well as the zigzags in the channel may contribute in

improving the results of the model. Another possible reason is COMSOL may not represent accurately the real life thermodynamic changes that occur in the PCHE. COMSOL's calculation of these thermodynamic behaviors shows a difference between them and the calculated experimental values. Once more data is obtained to predict the error trend more accurately, the errors can be fed into COMSOL which would improve the model to predict more accurate results for high air inlet temperatures.

Chapter 6

Conclusions

6.1 Overview

This work presents the numerical and experimental results of testing the heat transfer behaviors of a Printed Circuit Heat Exchanger (PCHE) with air and carbon dioxide as the working fluids. The experiments are conducted in order to provide a baseline for testing the PCHE using Supercritical Carbon Dioxide ($S - CO_2$) as the working fluid which play a vital role in the performance of $S - CO_2$ Brayton cycles.

The PCHE used in this work was designed by Heatric and setup at Canmet Energy, a division of Natural Resources Canada. Geometric details about the interior of the PCHE were not provided by Heatric due to the information being proprietary. Only partial details were provided which allowed us to estimate the PCHE dimensions, where these dimensions can be found section 4.3.

A 3D COMSOL Multiphysics model was created based on these dimensions to simulate the thermodynamic behavior of the PCHE. Based on work performed by Figley [17], only one column of the PCHE with 20 channels was modelled which gives

approximately the same results as modelling the entire heat exchanger containing 180 channels.

18 data sets were tested varying the air inlet temperature from 70 °C to 100 °C to 140 °C. The pressure of both fluids was alternated between 5 bars and 10 bars while the flow was alternated between 5 LPM and 10 LPM. The tests ran till steady state was achieved and outlet temperatures of the fluids as well as the pressure drop across the heat exchanger was recorded.

The pressure drop was investigated by varying the flow of both fluids from 1 LPM to 5 LPM to 10 LPM. The trends showed that with an increase in Reynolds number, an increase in pressure drop where CO_2 had an increase of 0.06 kPa drop with 1 LPM to 1.88 kPa drop with 10 LPM for 10 bars. For 5 bars, the same volumetric flow showed a lower pressure drop with 0.03 kPa for 1 LPM going up to 1.1 kPa for 10 LPM. Air pressure drop behaved the same with lower numbers than CO_2 with a drop of 1.48 kPa for 10 LPM at 10 bar and 1.03 kPa for 10 LPM at 5 bar.

The heat rate for all tests was calculated and showed an increase in heat rate as the air inlet temperature increased. It was observed that tests with the same inlet temperatures and flow but with different pressures resulted in almost equal outlet temperatures for air and CO_2 but with different heat rates showing the heat rate is almost double for flows at 10 bar when compared to the flows at 5 bar.

The outlet temperatures obtained from the COMSOL model were compared to the temperatures obtained experimentally and the results seem to agree. An average of 5.65% error was found when comparing outlet temperatures for air and an average of 5.73% was found when comparing outlet temperatures for CO_2 with a

maximum temperature difference of 10.6 °C. The trends obtained from the COMSOL model showing the temperature distribution appear to be physical showing a maximum heat transfer occurring at the inlet and small curves at the inlet indicating thermal development. Due to the inlet having maximum temperature difference, a rapid change in temperature occurs that slows down as the temperature difference decreases. It was observed that the higher the flow rate, the smaller the gradient of that temperature drop.

6.2 Future Work

The current work is preliminary however, the scope of this work provides the initial steps taken to test the PCHE using Supercritical Carbon Dioxide. Further testing and analysis is required to understand the complete thermodynamic behavior of the PCHE. Some noteworthy items to include for future work are

- 1- Modify the COMSOL model to include zigzags in the channels to investigate their effects on the pressure drop of the flow.
- 2- Calibrate the COMSOL model to take the temperature differences originating from high input temperatures into consideration. Once completed, more accurate simulated data for high input temperatures would be obtained.
- 3- Further investigation of the behaviors of the heat transfer coefficient and the role it plays in the performance of the PCHE.

4- Creating an analytical model to verify the findings of the temperature trends obtained from the numerical model.

5- Purchasing a new heater to increase the range of operation to include higher temperatures, pressures and flow rates.

6- Redesigning the loop to withstand the pressures and temperatures of reaching the supercritical phase.

References

- [1] V. Dostal. *A Supercritical Carbon Dioxide Cycle for Next Generation Nuclear Reactors*. Ph.d. thesis, Massachusetts Institute of Technology, 2004.
- [2] Mechanical and Aerospace Engineering 4th Year Project - Brayton Cycle Design. Technical report, Carleton University - Ottawa.
- [3] S. A. Wright, T. M. Conboy, and G. E. Rochau. Overview of Supercritical CO₂ Power Cycles Development at Sandia National Laboratories. Technical report, 2011.
- [4] M. A. Jacobs. *Measurement and Modeling of Thermodynamic Properties for the Processing of Polymers in Supercritical Fluids*. Ph.d. thesis, Technische Universiteit Eindhoven, 2004.
- [5] Steven A. Wright, Ross F. Radcliff, Milton E. Vernon, Gary E. Rochau and Paul S. Pickard. Operation and Analysis of a Supercritical CO₂ Brayton Cycle. Technical report, 2010.
- [6] M. Persichilli, A. Kacimov, E. Zdaniewicz and T. Held. Supercritical CO₂ Power Cycle Developments and Commercialization: Why sCO₂ Can Displace Steam. In *Power-Gen India Central Asia*, April 2012.
- [7] T. J. Held. Initial Test Results of a Megawatt-Class Supercritical CO₂ Heat Engine. In *sCO₂ Power Cycle Symposium*, September 2014.
- [8] ECHOGEN Power Systems. Clean, Economical Power Generation From Waste Heat Using Supercritical CO₂. http://www.echogen.com/_CE/pagecontent/Documents/News/Echogen_brochure_2016.pdf/.

- [9] R. Dennis. DOE Initiative on SCO₂ Power Cycles (STEP) - Heat Exchangers: A Performance and Cost Challenge -. In *EPRI-NETL Workshop on Heat Exchangers for SCO₂ Power Cycles*, October 2015.
- [10] Heatric. Compact Heat Exchangers - Driving Overall Capital Savings in Footprint, Volume and Structural Support Costs. http://www.heatric.com/compact_heat_exchanger.html.
- [11] J.E. Hesselgreaves. *Compact Heat Exchangers - Selection, Design and Operation*. Pergamon, 2001.
- [12] David Southall and Stephen John Dewson. Innovative Compact Heat Exchangers. In *ICAPP*, June 2010.
- [13] Renaud Le Pierres, David Southall and Stephen Osborne. Impact of Mechanical Design Issues on Printed Circuit Heat Exchangers. In *SCO₂ Power Cycle Symposium*, May 2011.
- [14] M. Pievi. A Calculation Method for the Optimized Thermal-Fluid Dynamic Sizing of Heat Exchangers. *Heat Transfer Engineering*, 6, 2008.
- [15] A. Moisseytsev, J. J. Sienicki, D.H. Cho and M. R. Thomas. Comparison of Heat Exchanger Modeling with Data from CO₂-to-CO₂ Printed Circuit Heat Exchanger Performance Tests. In *ICAPP*, June 2010.
- [16] S. P. Kar. CFD Analysis of Printed Circuit Heat Exchanger. Master's thesis, National Institute of Technology Rourkela, 2007.
- [17] J. Figley. Numerical Modeling and Performance Analysis of Printed Circuit Heat Exchanger for Very High-Temperature Reactors. Master's thesis, The Ohio State University, 2009.
- [18] A. Daouk, O. Petel and H. Saari. Multiphysics Simulation of a Printed Circuit Heat Exchanger. In *COMSOL Conference, Boston*, October 2015.
- [19] H. Song. Investigations of a Printed Circuit Heat Exchanger for Supercritical CO₂ and Water. Master's thesis, Kansas State University, 2007.

- [20] J. Vanmeter. Experimental Investigation of a Printed Circuit Heat Exchanger Using Supercritical Carbon Dioxide and Water as Heat Transfer Media. Master's thesis, Kansas State University, 2008.
- [21] T. Ishizuka, Y. Kato, Y. Muto, K. Nikitin and T.L. Ngo. Thermal-Hydraulic Characteristics of a Printed Circuit Heat Exchanger. *Bull. Res. Lab. for Nuclear Reactors*, 30, 2006.
- [22] S.G. Kim, Y. Lee, Y. Ahn and J. I. Lee. CFD Aided Approach to Design Printed Circuit Heat Exchangers for Supercritical CO₂ Brayton Cycle Application. *Annals of Nuclear Energy*, 92:175–185, 2016.
- [23] V. Gnielinski. New Equations for Heat and Mass Transfer in Turbulent Pipe and Channel Flow. *International Chemical Engineering*, 16, 1976.
- [24] A. Efanov, N. Loginov and V. Privezentsev. Heat Transfer at Liquid Flow in Microchannels. In *Nuclear Reactor Thermal Hydraulics*, October 2005.
- [25] S. Mehendale, A. Jacobi and R. Shah. Fluid Flow and Heat Transfer at Micro- and Meso-Scales with Applications to Heat Exchanger Design. *Applied Mechanics Review*, 53:175–193, 2000.
- [26] X. Peng, G. Peterson and B. Wang. Heat Transfer Characteristics of Water Flowing Through Microchannels. *Experimental Heat Transfer*, 7:265–283, 1994.
- [27] X. Peng and G. Peterson. Convective Heat Transfer and Flow Friction for Water Flow in Microchannel Structures. *International Journal of Heat and Mass Transfer*, 39:2599–2608.
- [28] X. Peng, G. Peterson and B. Wang. Frictional Flow Characteristics of Water Flowing Through Rectangular Microchannels. *International Journal of Heat and Mass Transfer*, 17:73–82.
- [29] S. Liao and T. Zhao. Measurements of Heat Transfer Coefficients From Supercritical CO₂ Flowing in Horizontal Mini/Micro Channels. *Journal of Heat Transfer*, pages 413–420, June 2002.

- [30] S. Liao and T. Zhao. An Experimental Investigation of Convective Heat Transfer to SCO₂ in Miniature Tubes. *International Journal of Heat and Mass Transfer*, 45:5025–5034, 2002.
- [31] S. Mokry and I. Pioro. Heat Transfer Correlation for Supercritical Carbon Dioxide. In *SCO₂ Power Cycle Symposium*, May 2011.
- [32] S.G. Kandlikar. *Heat Transfer and Fluid Flow in Minichannels and Microchannels*. Elsevier B.V., 2006.
- [33] H. Li, Y. Zhang, L. Zhang, M. Yao, A. Kruizenga and M. Anderson. Pdf-Based Modeling on the Turbulent Convection Heat Transfer of Supercritical CO₂ in the Printed Circuit Heat Exchangers for the Supercritical CO₂ Brayton Cycle. *International Journal of Heat and Mass Transfer*, 98:204–218, 2016.
- [34] A. Meshram, A. K. Jaiswal, S. D. Khivsara, J. D. Ortega, C. Ho, R. Bapat and P. Dutta. Modeling and Analysis of a Printed Circuit Heat Exchanger for Supercritical CO₂ Power Cycle Applications. *Applied Thermal Engineering*, 2016.
- [35] S.M. Lee and K.Y. Kim. Design Optimization of a PCHE Using RSA Model. In *Transaction of the Korean Nuclear Society Autumn Meeting*, May 2011.
- [36] Cryogenic Technologies Group. Cryogenic Material Properties. http://www.cryogenics.nist.gov/MPropsMAY/304Stainless/304Stainless_rev.htm.

Appendix A

Error Analysis

This appendix calculates the error based on the measurement uncertainties of the instruments used to gather the experimental data. Equations A.1 through A.7 highlight the relationship between the error and its corresponding mathematical operation

If $Z = A + B$, therefore the error is,

$$(\Delta Z)^2 = (\Delta A)^2 + (\Delta B)^2 \quad (\text{A.1})$$

If $Z = A - B$, therefore the error is,

$$(\Delta Z)^2 = (\Delta A)^2 + (\Delta B)^2 \quad (\text{A.2})$$

If $Z = AB$. therefore the error is,

$$\left(\frac{\Delta Z}{Z}\right)^2 = \left(\frac{\Delta A}{A}\right)^2 + \left(\frac{\Delta B}{B}\right)^2 \quad (\text{A.3})$$

If $Z = A/B$, therefore the error is,

$$\left(\frac{\Delta Z}{Z}\right)^2 = \left(\frac{\Delta A}{A}\right)^2 + \left(\frac{\Delta B}{B}\right)^2 \quad (\text{A.4})$$

If $Z = A^n$, therefore the error is,

$$\frac{\Delta Z}{Z} = n \frac{\Delta A}{A} \quad (\text{A.5})$$

If $Z = \ln A$, therefore the error is,

$$\Delta Z = \frac{\Delta A}{A} \quad (\text{A.6})$$

Finally, if the error is $Z = e^A$, therefore the error is,

$$\frac{\Delta Z}{Z} = \Delta A \quad (\text{A.7})$$

The heat rate equation is defined by

$$Q = \dot{m} \Delta H \quad (\text{A.8})$$

Therefore, using equation A.3, we obtain

$$\left(\frac{\Delta Q}{Q}\right)^2 = \left(\frac{\Delta \dot{m}}{\dot{m}}\right)^2 + \left(\frac{\Delta[h_{in} - h_{out}]}{[h_{in} - h_{out}]}\right)^2 \quad (\text{A.9})$$

The error in the enthalpy term of equation B.9 uses equation A.2 to obtain

$$\Delta(h_{in} - h_{out}) = [(\Delta h_{in})^2 + \Delta h_{out}]^{\frac{1}{2}} \quad (\text{A.10})$$

As an example, the error for the hot stream of Test 1 was calculated. The conditions for air for Test 1 are found in Table A.1

Table A.1: Test 1 Air Conditions

<i>Pressure (Bar)</i>	<i>Air Inlet ($^{\circ}C$)</i>	<i>Air Outlet ($^{\circ}C$)</i>
10	140.4	23.4

The enthalpy is affected by both temperature and pressure which are measured by the J-type thermocouples having an accuracy of ± 2.2 $^{\circ}C$ and the Swagelok pressure gages having an accuracy of ± 0.625 bar respectively. Table A.2 shows the temperatures and pressures with their corresponding upper and lower limit

Table A.2: Enthalpy for Air

Pressure (bar)	Temperature ($^{\circ}C$)	Enthalpy (kj/kg)	Temperature ($^{\circ}C$)	Enthalpy (kj/kg)
9.375	138.2	535.7	21.2	416.3
10	140.4	537.8	23.4	418.4
10.625	142.6	540.0	25.6	420.5

$$\Delta h_{in} = 540.0 - 537.8 = 2.2 \quad (\text{A.11})$$

$$\Delta h_{out} = 420.5 - 418.4 = 2.1 \quad (\text{A.12})$$

$$\Delta(h_{in} - h_{out}) = [(2.2)^2 + (2.2)^2]^{\frac{1}{2}} = 3.04 \quad (\text{A.13})$$

$$\frac{\Delta \dot{m}}{\dot{m}} = 0.008 \quad (\text{A.14})$$

Therefore, with the value of Q being 259.9 W, ΔQ is equal to

$$\Delta Q = \left[0.008^2 + \left(\frac{3.04}{537800 - 418400} \right)^2 \right]^{\frac{1}{2}} \times 259.9 \quad (\text{A.15})$$

From these calculations we deduce that the error for the hot stream for Test 1 is ± 6.935 W.

Appendix B

3D COMSOL Results

Figure B.1: Test 1 Temperature Distribution

Figure B.2: Test 2 Temperature Distribution

Figure B.3: Test 3 Temperature Distribution

Figure B.4: Test 4 Temperature Distribution

Figure B.5: Test 5 Temperature Distribution

Figure B.6: Test 6 Temperature Distribution

Figure B.7: Test 7 Temperature Distribution

Figure B.8: Test 8 Temperature Distribution

Figure B.9: Test 9 Temperature Distribution

Figure B.10: Test 10 Temperature Distribution

Figure B.11: Test 11 Temperature Distribution

Figure B.12: Test 12 Temperature Distribution

Figure B.13: Test 14 Temperature Distribution

Figure B.14: Test 15 Temperature Distribution

Figure B.15: Test 16 Temperature Distribution

Figure B.16: Test 17 Temperature Distribution

Figure B.17: Test 18 Temperature Distribution