Apr 21st, 2:00 PM - 3:00 PM

Compact Steam Generator Numerical Analysis and Mechanics Design

Parker Rockholm
University of Missouri-Columbia

Congshan Mao
University of Missouri-Columbia

Compact Steam Generator Numerical Analysis and Mechanics Design

Parker Rockholm & Congshan Mao
University of Missouri
Dr. Yue Jin

Abstract

A heat exchanger is a device that is used to transfer heat between two fluids that are at different temperatures and are separated by a solid wall. This system is used in many engineering applications such as air conditions, power production, waste heat recovery and chemical processing. Heat exchangers are commonly classified by their flow arrangement and type of construction. A special and important class of heat exchanger is termed compact heat exchangers, but 'compact' does not refer to its size. These heat exchangers are used to achieve higher performance by using very large surface densities (heat transfer surface area per unit volume) and through selection of heat transfer geometries.

The aim of the present study is to investigate the design options available for creating compact heat exchangers (CHX) that can be used in commercial nuclear power plants, with the goal of enhancing their technology readiness level for practical deployment. Initially, the study will concentrate on creating efficient computational models using ANSYS Fluent that can accurately provide the working conditions of CHXs in their final application. These computational models will later be validated with physical experiments on CHX samples. Finally, the research will also examine the thermal stress performance of CHXs in different system and design conditions.

Biography

Parker Rockholm grew up in Alton, IL before attending the University of Missouri. He is majoring in Mechanical Engineering with minors in Aerospace Engineering and Mathematics. On campus, he has served as the social chairman of Sigma Nu Rho chapter. In this position, he drafted and managed a budget of $60,000 for a variety of events across two semesters. Namely, he created and executed the first annual charity concert that raised $22,270 for a veteran's homeless shelter. He is also a Co-founder and treasurer Missouri chapter of National Society of Sales Engineers where we discuss opportunities in Sales Engineering and network with local employers.

Parker has 2 years of experience working on a manufacturing team as an Engineering Technician at 3M. He worked in the Barrier Films Vacuum Coating department alongside Manufacturing and Process engineers to streamline and problem solve using the Lean Six Sigma and 5S methodologies. Upon graduation, Parker will be working as a Territory Sales Engineer for Cognex Corporation. Cognex Corporation is an American manufacturer of machine vision systems, software and sensors used in automated manufacturing to inspect and identify parts, detect defects, verify product assembly, and guide assembly robots.

Introduction

There are many systems in the aerospace industry that use heat exchanger technology to manage heat transfer and temperature regulation in various systems. Rocket engines operate at extremely high temperatures that require heat exchangers to actively cool the engine. In addition, many electric devices generate high heat flux upon operation which need to be cooled, since most of these devices have an optimal operating condition. Compact heat exchangers are uniquely qualified for these systems because they use less material to manufacture and can realize a higher thrust to weight ratio.

Recently, there has been a surge in interest regarding hypersonic civil transport and reusable access to space, but the hesitation of this technology is driven by mounting concerns over atmospheric carbon dioxide and the consequent climate impacts. This has sparked interest in advancing hypersonic precooled airbreathing propulsion technology, which is now touted as the most promising solution to Rocket-Based Combined Cycle (RBCC) and Turbine-Based Combined Cycle (TBCC) engines for high-speed, environmentally sound civil aviation. With its ability to offer a higher specific impulse across a wider Mach number spectrum, hypersonic precooled airbreathing propulsion technology is associated with reduced fuel usage, lower CO2 emissions, and higher compressor ratios. However, developing effective cooling mechanisms for the incoming airflow of such engines remains a major challenge, without causing structural failure due to significant thermal stresses. This remains a research and design problem to develop compact heat exchangers that can perform in these extreme environments.

Another application for compact heat exchangers is in future nuclear power generation systems. These heat exchangers produced the steam from the excess heat generated by the reactor, and this steam is used to turn a turbine that generates electricity. The improvement in this technology will lead to improved safety and reliability. Mature steam generators exist in a few forms within industry. once-through steam generator designed by B&W BWR [4]. This is a straight-tube and shell design with flat tube sheets and primary heads. This device works by forcing convection through heat transfer between the feedwater and the lower tube sheet.

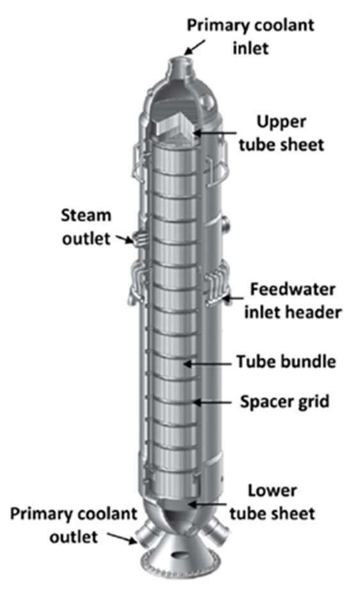


Figure 1: Cross section of the B&W OTSG tube and shell steam generator.

The increased demand from the nuclear and aerospace industries has sparked this investigation into the optimal design for compact plate-type heat exchangers. This technology promises much smaller and therefore fewer heavy designs. These heat exchangers typically use channels with hydraulic diameters in the range of 0.5 - 6 mm. Heat exchangers, like the ones manufactured by Heatric [4], are devices consist of many layers of metallic plates with a vertical array of tubes intersecting the layers normal to the plate surface. The tubes may be flat or circular, and the fins may be flat or corrugated. The types of heat exchangers that Heatric manufactures are Printed Circuit Heat Exchangers (PCHEs), Formed plate heat Exchangers (FPHEs), and Hybrid Heat Exchangers (H^2Xs). Each of these models consist of alternating fluid temperature layers as shown in Fig. 2. These devices can withstand the pressure and temperature of a conventional steam generator. Furthermore, the reliability of these designs demands further exploration of its behavior.

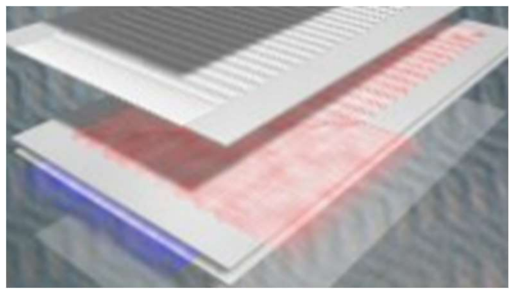
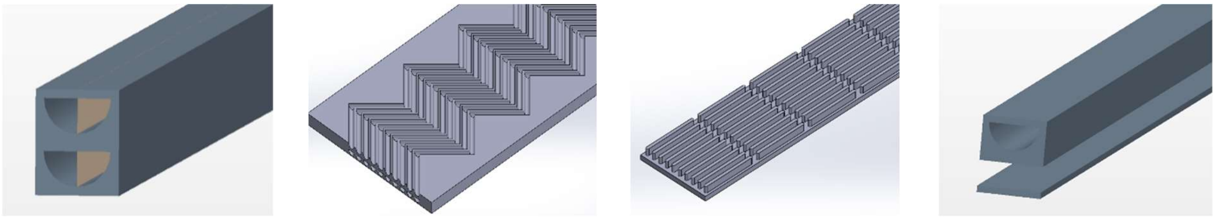


Figure 2: Visualization of cross flow channel layers

One of the most promising areas of progress is manipulation of the channel geometry. Several ideas for PCHE designs that optimize the heat transfer between the hot and cold channels as well as optimize the pressure drop across the flow channel have been proposed. These designs make use of a photo-chemical etching technique that removes material from the surface of a stainless-steel plate. Some proposed designs for experimentation are shown in Figs 4a and Fig 4b. However, the further reduced system volume and channel size as well as subsequently increased thermal and stress gradients pose certain risks for safe and efficient operation. Therefore, detailed, and comprehensive performance analysis and system optimization is necessary.

(a) Arrangement of Heatric plate-type heat exchanger **(b) Photograph of chemically etched plate**

Figure 3: Typical Heatric Compact Heat Exchanger Design



(a) Uniform Channel **(b) Zig-Zag Channel** **(c) Slotted fins** **(c) Plate and uniform channel**

Figure 4: Sketch of various CSG channel designs

The analysis of these CHXs is mostly limited to single-phase-to-single-phase heat transfer. During examinations there are depositions on the surface that block the flow through the channel and can interrupt the phase change process. Additionally, boiling phenomena in small geometries are intricate and not adequately represented by existing models. Early coalescence of bubbles in confined spaces dries out the channel, which casts doubt on the overall effect of mini-channel geometries on boiling heat transfer. Despite this, there is a scarcity of experimental data due to the difficulty of obtaining precise information and constructing dependable and universal correlations. This study intends to explore the behavior of the uniform channel geometry with the use of ANSYS fluent. First, the channel will be analyzed using single phase heat transfer, then proceed to study the 2-phase results of the CFD simulation. Later, these results will be compared to data collected from experimentation.

Factors Influencing Design

To start the design process of a new compact heat exchanger the design parameters must be assessed for practicality. The heat exchanger design problem is one of specifying the heat exchanger configuration and the heat transfer surface area required to achieve the desired outlet temperature. The first step of the analysis is to determine if the specified pressure and temperature change are possible with the materials that are qualified for manufacture. This is then corroborated with the process fluids. These fluids must be free off particulates that can cause fouling of the heat exchanger materials. Fouling is the deterioration of heat transfer structures from material deposits from the process fluids. These two factors will impact the choice of material and heat exchanger type. For example, if there is a specification for a high design temperature it will be necessary to use a more robust and costly alloy, then a FPHE solution may be the best fit because this type requires less raw material.

In industry applications with two streams, the process stream will be heated by a service or utility stream. In this case, its upstream and downstream temperatures and pressure drop will be specified closely. A fundamental part of the design problem is to find the most economic value for its flow rate and for the heater and cooler. Most often there is given flow rates and given inlet temperatures and the important design criterion is the outlet temperatures of the system [2] Fixing the thermal and pressure drop specifications, it is possible to determine the most effective configuration. From the thermal effectiveness method, the rate equation is defined by [3]

$$\dot{Q} = \dot{m}_h c_{p,h}(T_{h,o} - T_{h,i}) \quad (1)$$

$$\dot{Q} = \dot{m}_c c_{p,c}(T_{c,o} - T_{h,i}) \quad (2)$$

where $T_{h,i}$ and $T_{h,o}$ is the inlet and outlet temperature of the hotter fluid, $T_{c,i}$ and $T_{c,o}$ is the inlet and outlet temperature of the colder fluid, $\dot{m}$ is the mass flow rate of the hot fluid, and c_p is the specific heat of the fluid.

Commonly, the product $\dot{m}c_p$ is written as C for convenience. The curvature of the temperature distributions depends on whether the hot stream capacity rate C_h or cold stream capacity rate C_c is the smaller value. The smaller value of C is used to define the idealized performance of a heat exchanger with infinite surface area as shown in Eq. (3)

$$\dot{Q}_{max} = C_{min}(T_{h,i} - T_{c,i}) \tag{3}$$

This is best shown by the temperature curves seen in Fig. 5. The point of equal temperatures is called the pinch point. In this idealized case, it represents the point of zero temperature difference. In industry, the pinch is the point of minimum temperature difference.

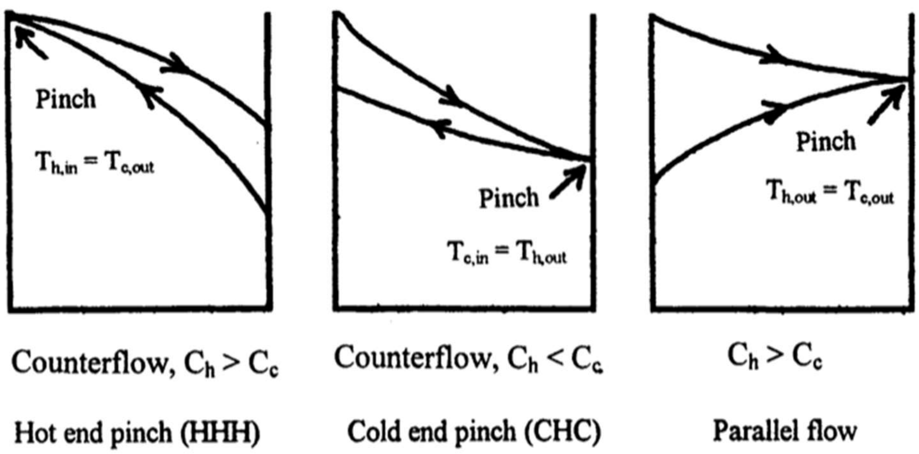


Figure 5: Idealized temperature distribution.

From the idealized calculations the effectiveness of the heat exchanger can be derived. The thermal effectiveness will influence the selection of configuration for the designed heat exchanger. Kays et al [3] define thermal effectiveness by

$$\varepsilon = \frac{\dot{Q}}{\dot{Q}_{max}} = \frac{C_h(T_{h,i} - T_{h,o})}{C_{min}(T_{h,i} - T_{c,i})} = \frac{C_c(T_{c,i} - T_{c,o})}{C_{min}(T_{h,i} - T_{c,i})} \tag{4}$$

Thus, the effectiveness of the exchanger can be found directly from the terminal temperature points. Another major relationship is the heat exchange equation as given by

$$\dot{Q} = (UA_s)\Delta T_m F_{GEOM} \tag{5}$$

Where UA is the product of the overall heat coefficient U and the surface area A_s. The symbol ΔT_m is the average temperature difference between streams, usually expressed as the Log Mean

Temperature Difference multiplied by a correction factor, F_{GEOM} that depends on flow configuration. Finally, the number of heat transfer units, NTU required is given by

$$NTU = \frac{UA_s}{C_{min}} \tag{6}$$

which is a measure of 'thermal length' of the exchanger. Experience shows that optimal heat exchanger designs maximize NTU and have F_{GEOM} approach 1 [1].

Concurrently, the allowable pressure drop should be specified appropriately. Optimizing the pressure drop is a game of balance. It is desirable to have a minimum pressure drop to decrease operating cost, but a very small pressure drop specification can be very difficult or impossible. Southhall et al. [1] define the pressure drop of a heat exchanger system by

$$\Delta P = 4f\left(\frac{1}{d_h}\right)\left(\frac{\rho u^2}{2}\right) \tag{7}$$

where f is the friction factor, d_h is the hydraulic diameter, ρ is the fluid density, and u is the fluid velocity. A low-pressure drop will require a short flow length and small fluid velocity. In consequence, this will affect the film heat transfer coefficient and thus reduce the overall heat transfer coefficient. Since F_{GEOM} is fixed by the configuration and the Log Mean Temperature Difference is fixed by the process' specification, Eq (5) shows that a small pressure drop can only be compensated by an increase in heat transfer area A_s.

Descriptions of tools

ANSYS Fluent is a computational fluid dynamics (CFD) software package developed by ANSYS, Inc. It is used for the simulation of fluid flow, heat transfer, and related phenomena. ANSYS Fluent offers a comprehensive suite of tools for modeling complex fluid systems and analyzing their behavior. The software uses numerical methods to solve the governing equations of fluid flow, including the Navier-Stokes equations, which describe the motion of fluids. ANSYS Fluent can simulate a wide range of fluid systems, including laminar and turbulent flows, compressible and incompressible flows, and multiphase flows. ANSYS Fluent is widely used in various industries, such as aerospace, automotive, energy, and manufacturing, for designing and optimizing products and processes. It can be used to simulate fluid behavior in a variety of applications, such as air flow over airplane wings, water flow through a turbine, or oil flow in pipelines.

Development of Numerical Models for CSGs

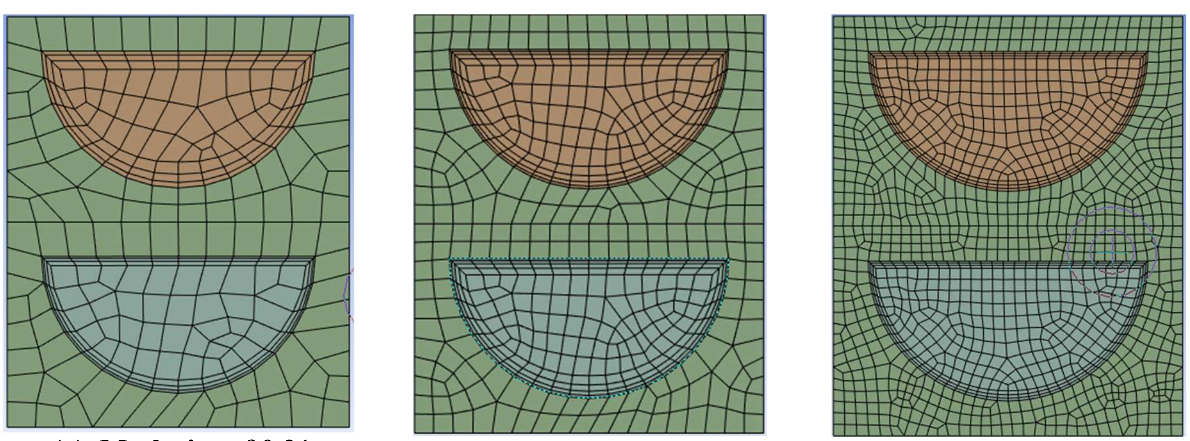
The current study aims to complete a deep analysis of the behavior of the uniform channel geometry. Instead of simultaneously simulating the entire flow over the plate, the model is truncated to include 1 hot and 1 cold channel in counter-current flow. The model is 3 mm tall and 2.5 mm wide, with two semicircular channels with a radius of 1 mm positioned 0.5 mm apart. When conducting the simulation, the inlets are defined by velocity and temperature while the outlets are pressure outlets.

A grid refinement study was first performed to determine the optimal meshing size for this mesh. In computational fluid dynamics (CFD) analysis, a grid refinement study is a process of systematically increasing the number of grid cells in a computational mesh to improve the accuracy of the simulation results. The computational mesh is a discretized representation of the geometry of the fluid domain being analyzed, and it is made up of many small cells or elements. The size and shape of these cells can have a significant impact on the accuracy of the CFD simulation results.

During the grid refinement study, we started with a coarse mesh with a relatively small number of cells and gradually increases the number of cells while keeping other simulation parameters, such as time-step and boundary conditions, constant. The simulation is run for each level of mesh refinement, and the results are compared to determine how much the solution changes as the mesh becomes finer. The process was repeated we reached a point of diminishing returns, where further refinement of the mesh no longer results in significant improvements in the accuracy of the simulation results.

During mesh construction, additional consideration was given to the turbulence model of the boundary layer of the channel. The accuracy of the turbulence model within the boundary layer region is measured by the y-plus value of each cell near the wall of the fluid mesh. Specifically, the y-plus value is a measure of the distance from the wall to the first cells center. If the y-plus value is low, the cell's flow is laminar, if the y-plus value is large the cell is turbulent. To account for this, an inflation layer was added to each mesh based on the present channel geometry. The inflation layer of each model had 4 layers with a growth rate of 1.5 and the first layer height was ¼ the mesh size. For this analysis, the realizable k-epsilon turbulence model with enhanced wall treatment was used. A benefit of this model is that it more accurately predicts rotation and recirculation.

The first coarse mesh created had a multizone geometry with cell size of 2.1 mm on the face normal to fluid flow, as well as edge size of 0.63 mm parallel to flow. The second fine mesh had a mesh size of 0.14 mm with an edge size of 0.48 mm, while the third finest mesh was 0.08 mm with an edge size of 0.36 mm. From smallest to largest mesh size, the cell counts were 2717044, 1054144, and 523020 cells respectively. Figure 6 shows the face of each mesh created for the grid refinement study. The smallest to largest cell count of each mesh is as follows.



(a) Mesh size of 0.21 mm **(b) Mesh size of 0.14 mm** **(c) Mesh size of 0.08 mm**

Figure 6: Cross section of each mesh included in the grid refinement study.

The single-phase analysis was conducted for each mesh size with both fluid channels specified as water and the solid material was 316 annealed stainless steel. The hot and cold inlet boundary conditions are presented in Table 1. Each inlet was treated as a velocity inlet and each outlet was treated as a pressure outlet. Each simulation was run for 800 iterations to allow for convergence of temperature.

Table 1: Inlet conditions for each channel.

Channel	Parameter	Value	Units
Hot channel	Velocity	2.3	m/s
	Pressure	12.88	MPa
	Temperature	552	°K
Cold channel	Velocity	3.19	m/s
	Pressure	16.3	MPa
	Temperature	603	°K

The two-phase simulation is currently being conducted using the mesh that was validated from the grid refinement study. The current study used a Eulerian mixture model with the Schiller-Naumann drag coefficient and the Manninen slip velocity model. This simulation assumes that the surface tension coefficient is constant at $\sigma = 0.021$ N/m. This simulation was run for 3000 iterations to allow for convergence. To ensure a valid comparison with the single-phase scenario, the same channel geometry was adopted. The hot channel system pressure was maintained at 16.3 MPa, while the cold channel system pressure was reduced from 12.88 MPa to 7.6 MPa to enable boiling under this pressure condition. The hot channel inlet temperature was specified at 340 °C (613.15 K) with an inlet velocity of 3.2 m/s, which is 10 degrees higher than that of the single-phase case. Meanwhile, the cold channel inlet temperature and velocity were set at 275 °C (548.15 K) and 0.101 m/s, respectively, due to the reduced volume of flow required to meet the heat transfer requirements.

Results and Discussion

The single-phase simulation was run for each of the 3 meshes. The outlet temperatures of each simulation are shown in Table 2. These results show that even the coarse mesh predicts the same outlet temperature of even the finest mesh that was simulated. Figure _ shows the axial temperature distribution within the 0.14 mm mesh. The behavior of the velocities shows a more dramatic difference in results. The outlet velocity of the hot channel increases with the decreasing mesh size while the outlet velocity of the cold channel decreases with decreasing mesh size. These velocity profiles also show a dramatic change in magnitude after 0.10 m into the channel. This suggests that these inlet conditions may be inaccurate for the proposed situation. The results of the grid refinement study show that if the outlet temperature is the only value of interest, a coarse mesh may be used. However, the finer meshes may be more viable to predict fluid behavior in the channel. Considering that the selected mesh will be used in two phase analysis, The 0.14 mm mesh size will yield the best balance of calculation speed and accuracy.

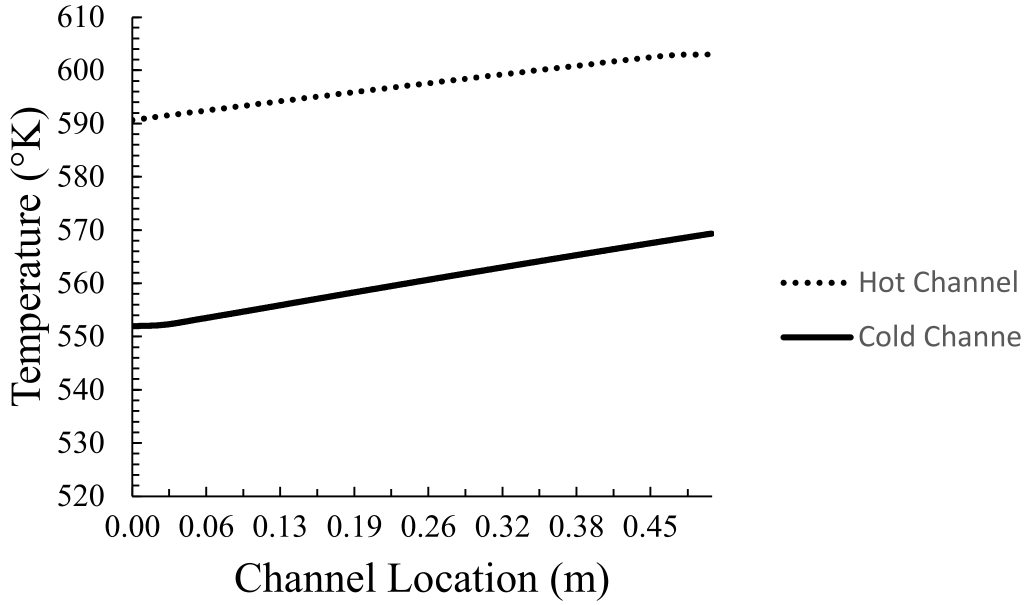


Figure 7: Axial temperature distribution of the 0.14 mm mesh.

Table 2: Outlet temperature of the hot and cold channels for each mesh.

Mesh Size	Hot Channel [°K]	Cold Channel [°K]
0.8 mm	589.8	570.4
1.4 mm	590.7	569.3
2.1 mm	590.3	569.8

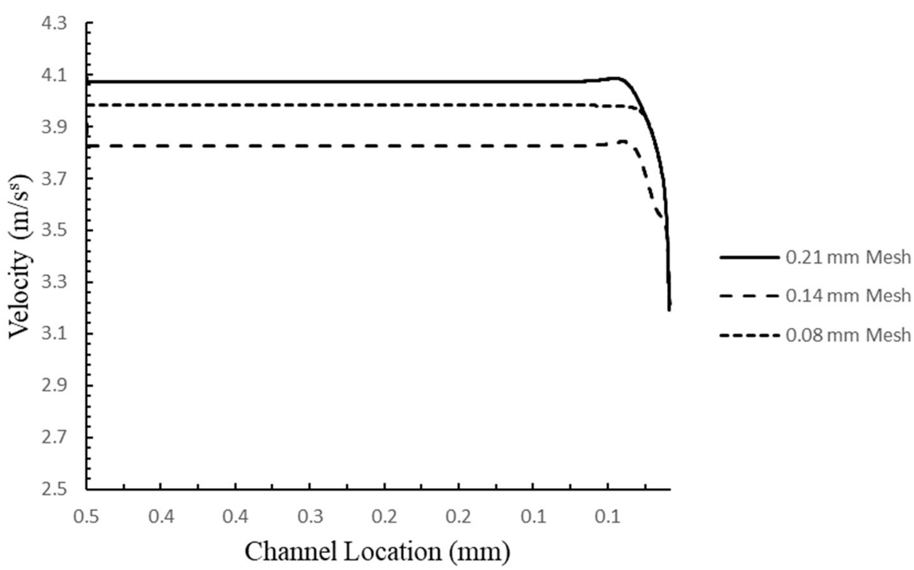


Figure 8: Velocity distribution in the hot channel for each mesh size.

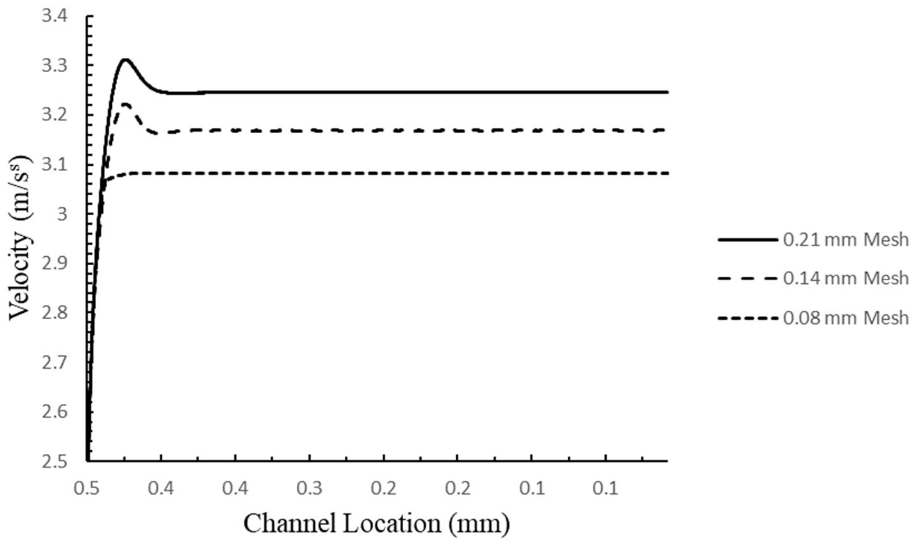


Figure 9: Velocity distribution in the cold channel for each mesh size.

Figure 10 shows a contour plot of the temperature for the 0.14 mm mesh halfway through the channel. An interesting part of this analysis is the dramatic difference between the top of the channel and the bottom of the channel. In future work, recursive boundary conditions may affect the outcome of this graph. Since these channels are meant to represent an array of channels it would be more accurate to consider the interactions between the channels surrounding this cell.

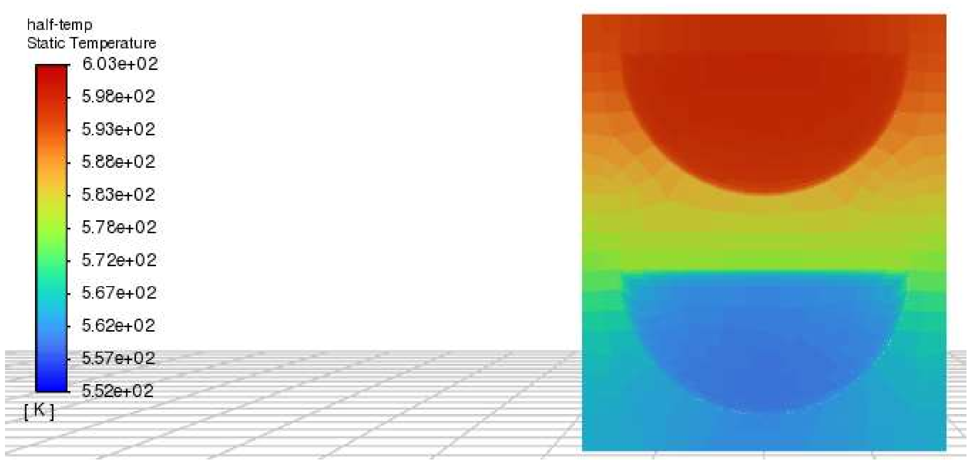


Figure 10: Temperature cross section halfway through the 0.14 mm mesh.

Figures 11 and 12 show the y-plus distribution at the inlets of the cold and hot channels. These distributions show an accurate performance of the turbulence model of this analysis. The realizable k-epsilon turbulence model with enhanced wall function calls for a y-plus value less than 5 (on the order of ~1).

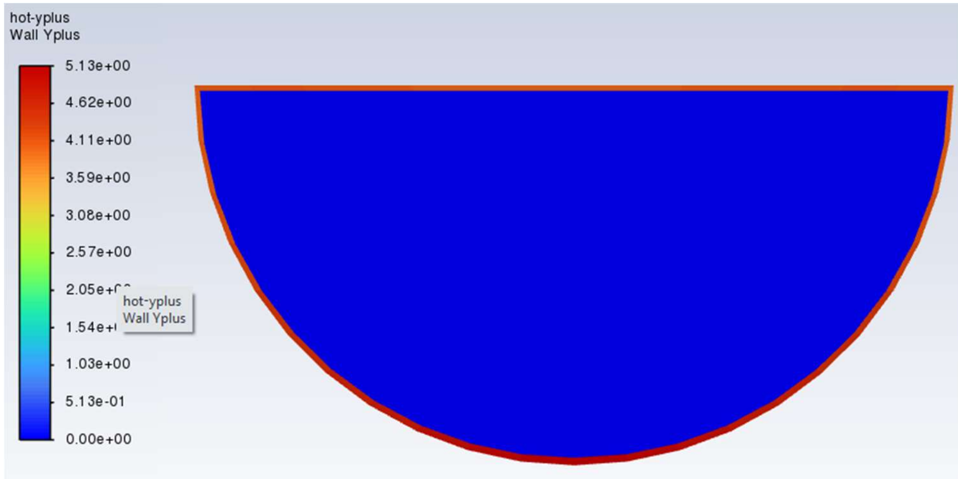


Figure 11: Distribution of Y-plus value in the hot channel of the 0.14 mm mesh.

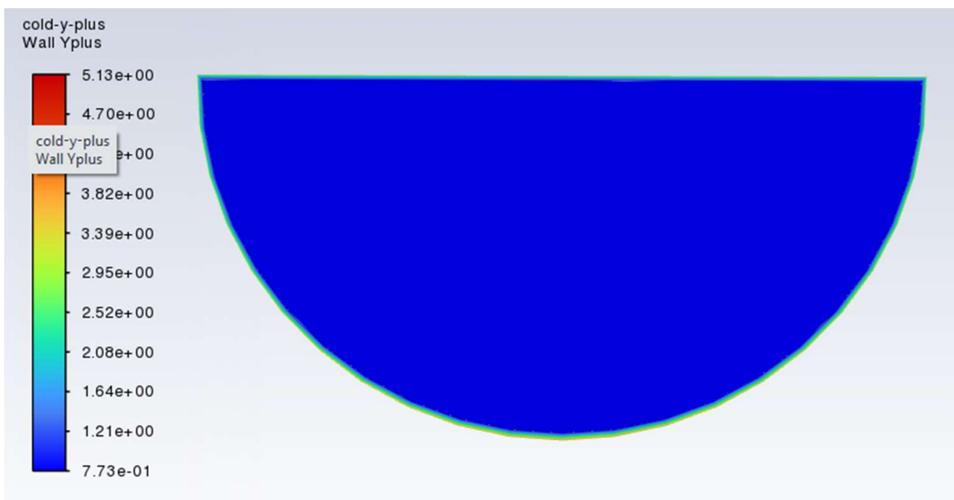


Figure 12: Distribution of Y-plus value in the cold channel of the 0.14 mm mesh.

Development of Experiments

The experimental validation of these results is currently underway. The current work is focused around exploring the behavior of two-phase flow within heat exchanger designs with channel size of mini/micro scale. The current test sections, made of either silicone or copper, have 4-6 micro-channels of 0.25-0.75 mm in width, manufactured by chemical etching or CNC machining, respectively. Figure 1 shows the micro-channel geometry on a silicone substrate The microchannel design is meant to meet the limited volume and high heat duty requirements of this experiment. The current configuration design is specifically aimed at investigating the small to micro scale two-phase flow mass and heat transfer process as well as interface behavior, which can be readily used to verify the CFD heat transfer closures as well as two-phase flow modeling capability. Moreover, the current experimental results will also provide useful information on the micro-channel design and optimization process.

To visualize the two-phase flow behavior as well as the detailed flow velocity distribution within the micro-channel. An advanced micro-PIV system was used in the current flow boiling experiments. Figure 2 shows the test configuration. The flow is driven by a syringe pump connected to the channel on the left side. The test section is heated at the bottom surface by three cartridge heaters lumped in a copper block. The water has photosensitive particles of 1 micro mixed into the fluid. A Dual pulse laser is directed onto a section of the channel that illuminates the particles and two high-speed cameras will record their movement.

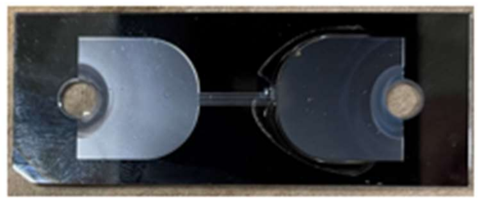


Figure 12: Photograph of microchannels subjected to heating element.

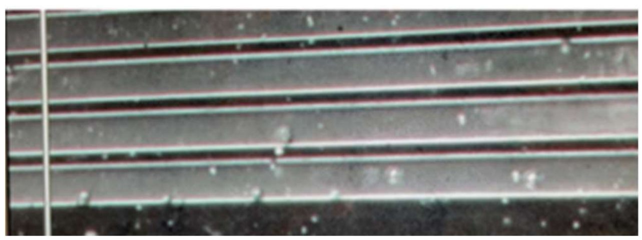


Figure 13: shows the obtained raw image for illuminated tracer particles in these 4 micro-channels.

In the current study, image processing software "Insight4G" is used to post process the raw images to obtain the final detailed velocity distribution. Once the detailed flow information is obtained, it will be compared with the CFD simulation results to further valid the existing numerical models used.

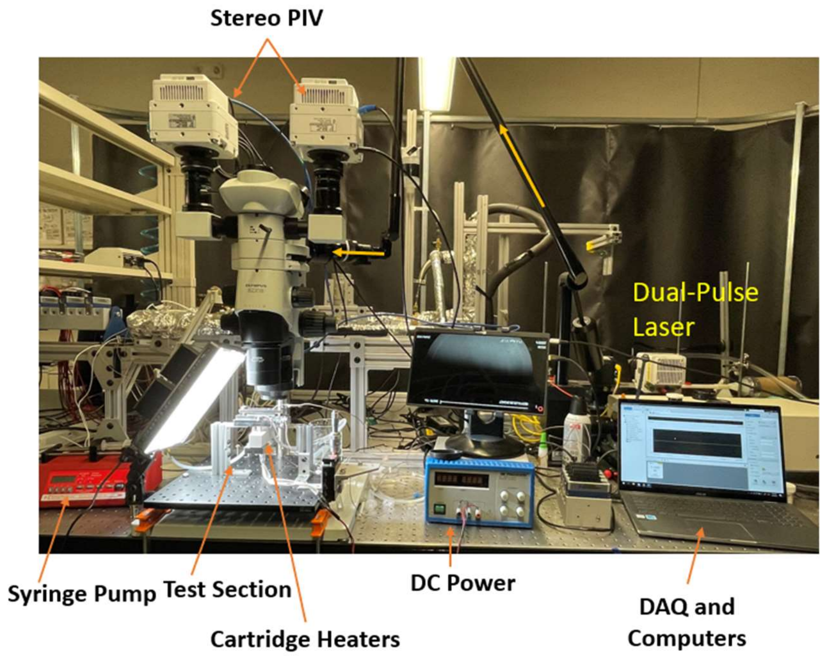


Figure 14: Photograph of experimental set up for study of microchannel flow.

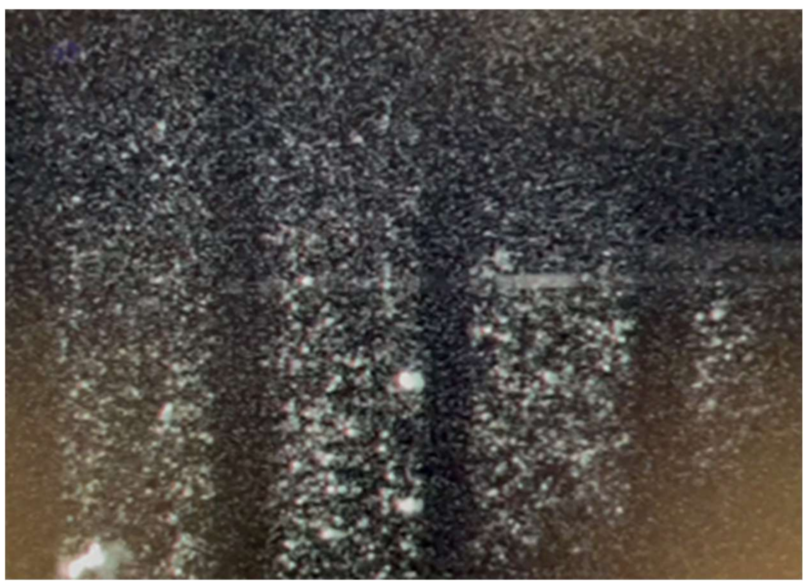


Figure 15: Photograph of photosensitive particles inside MCHX.

Conclusion

In conclusion, compact heat exchangers play a critical role in the aerospace industry by managing heat transfer and temperature regulation in rocket engines and electric devices. The development of hypersonic precooled airbreathing propulsion technology has created a demand for compact heat exchangers that can perform in extreme environments. In addition, the nuclear industry also

requires efficient and reliable heat exchangers for future power generation systems. The optimization of channel geometry in Printed Circuit Heat Exchangers (PCHEs) has shown promising results, but further research is necessary to ensure safe and efficient operation. As technology advances, the development of compact heat exchangers will continue to play a vital role in ensuring the reliability, safety, and efficiency of various industrial processes.

The current study has shown strong results for a single-phase simulation of uniform channel geometry. This study is not yet concluded, but theory shows that the multiphase solution will obtain a higher heat transfer than the single-phase case. Other geometries may also increase the heat transfer of these systems. Research should focus on geometries like the zig-zag channels to increase performance. Furthermore, the combination of two-phase flow and zig-zag geometry may be the highest performing configuration. These investigations will be continued with more simulations and later confirmed with experimentation.

Acknowledgements

I would like to express my sincere gratitude to all those who have contributed to the completion of this research project. First and foremost, I would like to thank my faculty advisor. Dr. Yue Jin for his invaluable guidance, support, and feedback throughout the project. I would also like to acknowledge the technical assistance provided by Congshan Mao who helped with data collection, analysis, and interpretation. His expertise and dedication were instrumental in the successful completion of this project. Finally, I am grateful for the financial support provided by the NASA-Missouri Space Grant Consortium Grant Award number 80NSSC20M0100. This funding allowed me to pursue this research project and provided the necessary resources to conduct the research activities. Once again, I express my heartfelt thanks to all those who contributed to this research project and helped make it a success.

References

[1] Southhall, D., Le Pierre R., Dawson S. J., "Design considerations for Compact Heat Exchangers," ICAPP, 2008.

[2] Bergman, T. L., Lavine, A. S., Incropera, F. P., & DeWitt, D. P. (2018). Fundamentals of Heat and Mass Transfer, 8th ed., Wiley Global Education US.

[3] Kays, W. M., and A. L. London, *Compact Heat Exchangers*, 3rd ed., McGraw-Hill, New York, 1984.

[4] Daniel Kromer, "MICROCHANNEL HEAT EXCHANGERS FOR INTEGRAL LIGHT WATER REACTORS," Ph.D. Dissertation, Georgia Institute of Technology, August 2019.

[5] Yue Jin, Koroush Shirvan, Patrick Fourspring, Kenneth Kimball, Compact Steam Generator Thermal-Hydraulic Assessment and Mechanics Design, in progress 2023.