International Journal of Innovative Research in Technology and Science
ISSN: 2321-1156 www.ijirts.org Volume 12 Issue 2, March 2024

Simulation-Based Analysis of Additively Manufactured Printed Circuit Heat Exchangers

Sunil Shah, Vineet Kumar Dwivedi
Faculty of Engineering and Technology
SAM Global University, Bhopal, India

Selection and peer review of this article are under the responsibility of the scientific committee of the International Conference on Current Trends in Engineering, Science, and Management (ICCSTEM-2024) at SAM Global University, Bhopal.

Abstract - Compact heat exchangers often incorporate semicircular flow channels in thermal processing applications involving viscous media. However, due to manufacturing constraints, these channels may have semielliptical shapes. With small hydraulic diameters and relatively large length-to-diameter ratios (L/dh), these channels typically experience laminar flows that are fully developed hydrodynamically and thermally. Initially, we investigated fully developed laminar flow and heat transfer in a straight, circular, smooth duct subjected to constant heat flux using FLUENT. The accuracy of the solutions was validated by calculating the Fanning friction factor, Nusselt number, and Colburn factor. Subsequently, we extended our analysis to include three-dimensional, periodic straight and sinusoidal ducts with semielliptical cross-sections of varying aspect ratios. Employing Computational Fluid Dynamics (CFD) with FLUENT, we examined the impact of Reynolds number ($50 \leq Re \leq 500$) and aspect ratio for straight ducts and amplitude-to-wavelength ratio (A/L=0.3 & 0.5) for sinusoidal channels (with L/D=4.5) on heat transfer enhancement and pressure drop. We studied velocity and temperature fields, Fanning friction, Colburn, and goodness factors. Notably, the interruption of boundary layers near solid surfaces, replaced by fluid from the core, increases temperature gradients, leading to higher overall heat transfer coefficients and pressure drop penalties in sinusoidal ducts than in straight ducts. Finally, we established correlations between Fanning friction factor, Colburn factor, Reynolds number, and geometrical parameters for the analysed configurations.

Keywords: Compact Heat Exchanger, Laminar Flow, Printed Circuit Heat Exchanger, CFD, Heat Transfer Enhancement, Semielliptical Duct

1. INTRODUCTION

In contemporary thermal processing applications, compact heat exchangers facilitate efficient heat transfer, particularly in handling viscous media. Among these, printed circuit heat exchangers have garnered significant attention for their ability to achieve compactness and high heat transfer rates. Typically, these heat exchangers feature semicircular flow channels, optimising heat transfer while minimising space requirements. However, practical manufacturing constraints often necessitate deviations from ideal geometries, leading to the adoption semielliptical shapes for these flow channels. The intricate design of printed circuit heat exchangers poses interesting challenges and opportunities for analysis and optimisation. Given their small hydraulic diameters and relatively large length-to-diameter ratios (L/dh), the flow characteristics within these channels tend to exhibit laminar behaviour. In such laminar flows, hydrodynamically and thermally fully developed conditions prevail, providing a stable foundation for understanding heat transfer phenomena. The present study elucidates the fully developed laminar flow and heat transfer behaviour within printed circuit heat exchangers. We begin our investigation by examining the fundamental flow and heat transfer case in a straight, circular, smooth duct subjected to constant heat flux. Computational simulations using FLUENT software are employed to analyse key parameters such as the Fanning friction factor, Nusselt number, and Colburn factor, serving as benchmarks for validating our numerical approach. Building upon this foundational understanding, we extend our analysis to encompass more complex geometries, including three-dimensional periodic straight and sinusoidal ducts with semielliptical cross-sections. These geometries represent practical variations encountered in printed circuit heat exchangers, allowing us to explore the effects of Reynolds number variations and aspect ratios on heat transfer enhancement and pressure drop. Computational Fluid Dynamics (CFD) techniques enable us to delve deep into the intricacies of fluid behaviour within these complex geometries. By varying parameters such as Reynolds number and aspect ratio, we aim to uncover insights into the underlying mechanisms governing heat transfer and pressure drop characteristics. Specifically, we investigate the impact of Reynolds number variations (ranging from 50 to 500), aspect ratios for straight ducts, and amplitude-to-wavelength ratios for sinusoidal channels. Our study focuses on elucidating the flow and heat transfer characteristics and understanding the practical implications for heat exchanger design and operation. We examine velocity and temperature fields, as well as the fading friction factor, Colburn factor, and goodness factor, to comprehensively analyse heat transfer performance. Furthermore, we explore the underlying mechanisms driving heat transfer enhancement and pressure drop penalties, particularly in sinusoidal ducts, where the interruption of boundary layers significantly influences overall heat transfer coefficients.

2. LITERATURE SURVEY

The supercritical carbon dioxide (sCO2) Brayton cycle is being explored for sustainable energy production and emission reduction in the solar and nuclear sectors. The primary advantages of this cycle are its high thermal efficiency and compact system design [1–10]. In order to achieve these benefits, an efficient design of a high-performance heat exchanger is needed to recover the significant amount of unused heat from the turbine flow in a low-pressure-ratio cycle. The heat exchanger must endure high temperatures and pressures due to the working conditions of the sCO2 Brayton cycle. Several design studies have concentrated on high-performance heat exchangers [1–3,11,12]. The Printed Circuit Heat Exchanger (PCHE) is highly regarded as a top contender for fulfilling the requirements. PCHEs are created by chemical etching and diffusion bonding. Plates are etched to create flow channels, stacked and joined together through diffusion. Each flow channel has a semicircular cross-section with a diameter of approximately 1 mm. The design of PCHE is based on the acquired information. Friction and heat transmission at the channel surface.PCHE has numerous flow channel geometry and can be used in industry applications. Many researchers have evaluated the thermal-hydraulic performance of PCHE with helium, sCO2, or water as a working fluid. Kim et al. [13] conducted experiments to investigate the thermal-hydraulic performance of a Printed Circuit Heat Exchanger (PCHE) utilising helium as the working fluid. Their studies spanned a range of Reynolds numbers from 350 to 1200 (primarily at the laminar regions) and inlet temperatures from 25 to 550 °C at the hot side and 25–100 °C at the cold side throughout a range of operating pressures of 1.5–1.9 MPa. Correlations for friction factor and Nusselt number were derived from their observations of pressure losses and fluid temperatures at the inlet and exit of both the hot and cold sides. Mylavarapu et al. [14] evaluated the performance of two PCHEs in a high-temperature helium test facility. Ranges of inlet temperature, pressure and Reynolds number varied from 85 to 390 °C, 1.0–2.7 MPa and 950–4100, respectively, at the cold side and 208–790 °C, 1.0–2.7 MPa and 900–3900, respectively, at the hot side of the PCHEs. Measurements of pressure losses and fluid temperatures were evaluated to establish hydraulics resistance and heat transfer characteristics of the channel surface. Ishizuka et al. [15] studied the thermal-hydraulic properties of sCO2 in a PCHE with flow channels of 1.15 mm in hydraulic diameter. Their experiment encompassed ranges of Reynolds number of 2400–6000 on the hot side and 5000–13,000 on the cold side of the PCHE. The working fluid is in a gas subcritical state or gas-like supercritical state distant from the pseudo-critical point. Fluid temperatures at both the hot and cold sides were measured. Their experimental results showed that overall heat transfer coefficients varied over 300–700 W m−2 K−1. These results developed a local heat transfer coefficient correlation as a function of the average Reynolds number. As a basic design of PCHE, straight channels have been examined experimentally or numerically. Park et al. [16] evaluated the heat transfer of a PCHE with straight channels, which is employed as a pre-cooler of the sCO2 Brayton cycle. Their experiments addressed three operating conditions of the coolant: trans-critical, near-critical, and supercritical. They concluded that applying the average temperatures at the entrance and outlet of the flow channel in computing the heat-transfer coefficient is unsuitable due to the large change in characteristics of CO2 near the critical point. A discretisation method for examining the data was proposed. Chu et al. [17] examined the thermo-hydraulic performance of a PCHE with straight channels. sCO2 flows at the hot Fig. 1. 3-D model of a PCHE side, and water flows at the cool side. Fluid temperatures and pressure losses at both sides were measured throughout a range of CO2 operating pressures. Heat-transfer rate and friction factor were determined from the experimental data for their PCHE design. Ren et al. [18] examined the local heat transfer characteristics of sCO2 flowing in horizontal semicircular straight channels of a PCHE during cooling under forced and mixed convective conditions. A correlation was created for the local heat transfer coefficient, which accounts for the thermophysical property fluctuations and the buoyancy effects. In addition, the authors presented a generalised mean temperature difference method for building PCHEs. Seo et al. [19] performed studies with water flow through straight channels of 0.6685 mm in a PCHE over a range of Reynolds numbers from 100 to 850. The hot-side entrance temperature varied between 40 and 50 °C, whereas the cold-side temperature was fixed at 20 °C. The average heat transfer rate and hydraulics resistance features were examined through the analysis of measurements of fluid temperatures and pressure losses. Kruizenga [20] tested PCHE with straight channels under horizontal, uphill, and downward flow conditions to assess the effect of buoyancy forces on sCO2 heat transfer. Under their test conditions, they observed little or no effect of flow orientation on heat transmission, even at places with large buoyancy forces (Gr/Reb 2.7>10−5). To increase the heat transfer performance of PCHE, some researchers have investigated the thermal-hydraulic performance of PCHE with zigzag channels or sinusoidal wavy channels.

2. METHODOLOGY

In the methodology section, the user will follow a set of procedural steps to address the identified features of the problem. First, they will create the model geometry and grid. Next, they will initiate the appropriate solver for either 2D or 3D modelling and import the grid, followed by a check of the grid's integrity. Then, they will select the solver formulation and choose the equations to be solved, such as laminar or turbulent flow, chemical species or reactions, and heat transfer models. They will also identify additional models required, such as fans, heat exchangers, or porous media. Material properties will be specified by setting boundary conditions and adjusting solution control parameters. The flow field will be initialised, and a solution will be calculated. Afterwards, the results will be examined and saved. If necessary, the grid will be refined, or revisions to the numerical or physical model will be considered. These steps ensure a systematic approach to addressing the problem at hand.

2.1. Boundary Conditions

The governing differential equations of a CFD program need to be provided with boundary and beginning (for transient solutions) conditions for comprehensive solutions over the space and time domain of interest. The boundary conditions often involve velocities (or flow rate in lieu of them), pressure and temperature over the bounding surfaces. The method of addressing a fluid flow problem is generally viewed as the extrapolation of a data collection defined on the bounding contours or surfaces into the domain interior. It is, thus, important for the user to supply physically realistic and well-posed boundary conditions to ensure accurate and stable solutions.

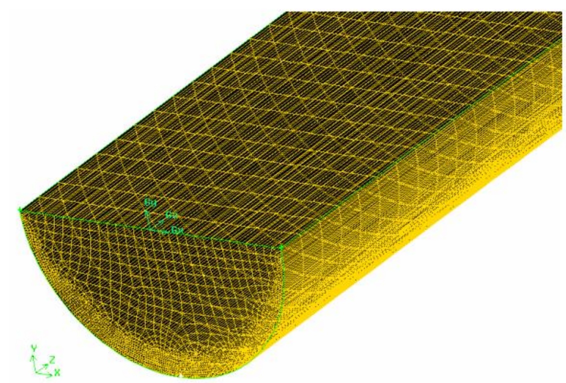

Figure 1. Meshing for the whole computational domain

Table 1. Thermo-physical properties of air for laminar flow in a circular duct

Fluid -air
Density=1.225 kg/m^3
Thermal conductivity=0.0242 w/m-k
Viscosity=1.7894e-05kg/m-s
Specific heat at constant pressure=1006.43 J/kg-k

A solid wall represents an impenetrable and no-slip boundary for a viscous fluid. All velocity components vanish on a stationary solid border. Newton's law of viscosity determines the shear stress. For investigation of heat transfer, isothermal or continuous heat flow boundary conditions are commonly applied. The SIMPLE algorithms evaluate the coupling between pressure and Velocity. The under-relaxation factor momentum and Energy have been 0.7 and 1, respectively. The second-order upwind scheme has been taken to discretisation momentum and energy equation. The 3-D double precision solver has been chosen. Convergence criteria have been taken as 1e-06 for continuity and Energy. The following grid-independent test was done using different grid sizes. Nusselt Number variation has been observed.

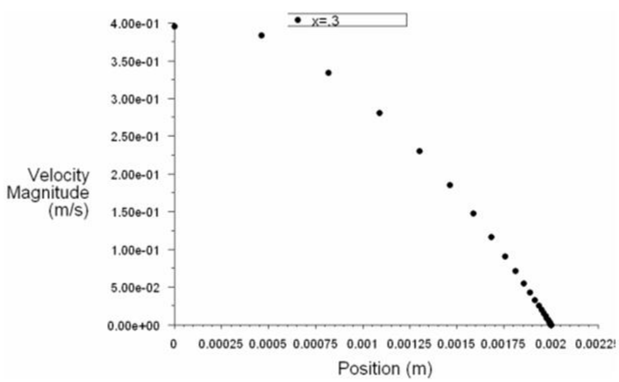

Figure 2. Variation of Velocity at a section along the Y-axis in a circular

3. RESULT AND DISCUSSION

A circular conduit with a length (L0) of 400 mm and a diameter of 4 mm was chosen for the analysis. Due to symmetry, the geometry was simplified, resulting in a diameter (d) of 2 mm. A finer mesh was applied near the wall to capture the velocity gradient. The Reynolds number (Re) was set to 50. A velocity of 0.2 m/s was assigned as the inlet velocity (u) to establish a Velocity Inlet condition at the Duct's inlet. Friction and Colburn factors were computed to assess the fluid flow and heat transmission characteristics. A relatively large length compared to the diameter was selected to ensure fully developed flow conditions and obtain comprehensive results.

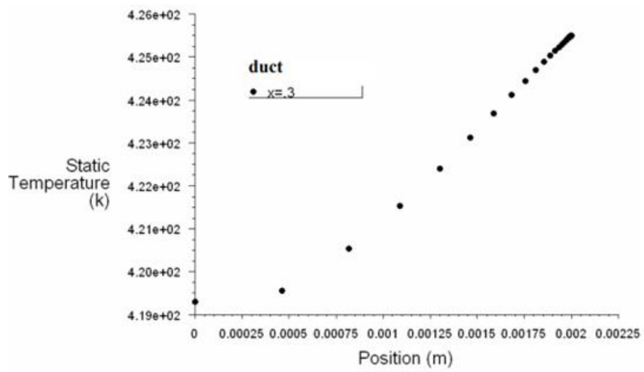

Figure 3. Variation of temperature at a section along the Y-axis. in a circular duct

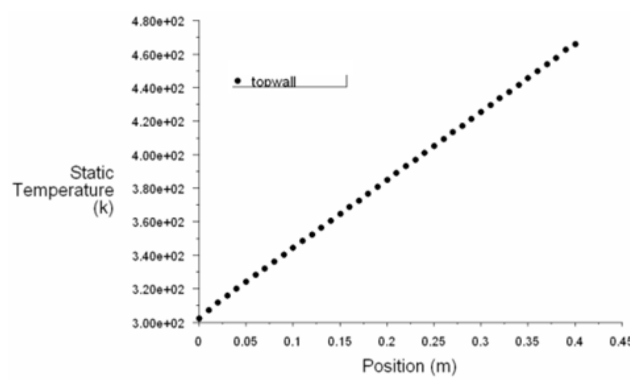

Figure 4. Variation of wall temperature along the length of the circular duct

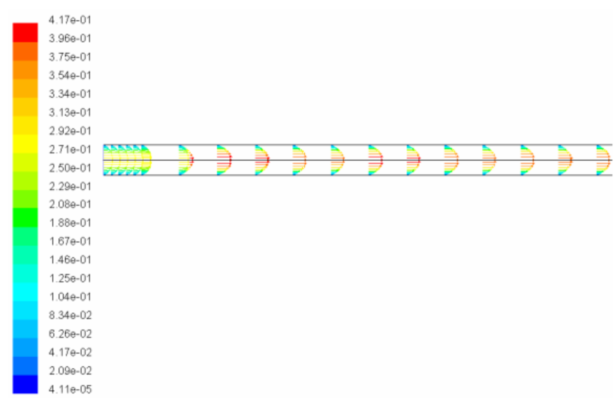

Figure 5. Display of velocity vectors showing the flow development.

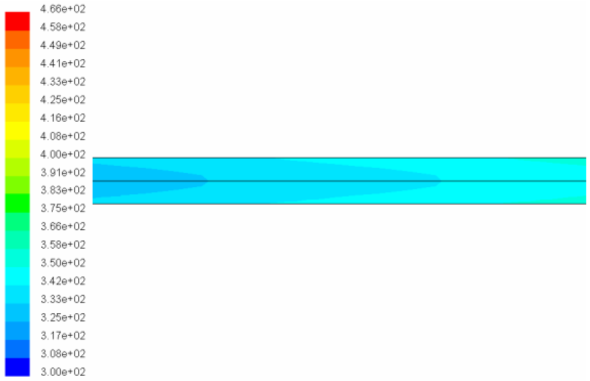

Figure 6. Temperature contour along the length of the Duct

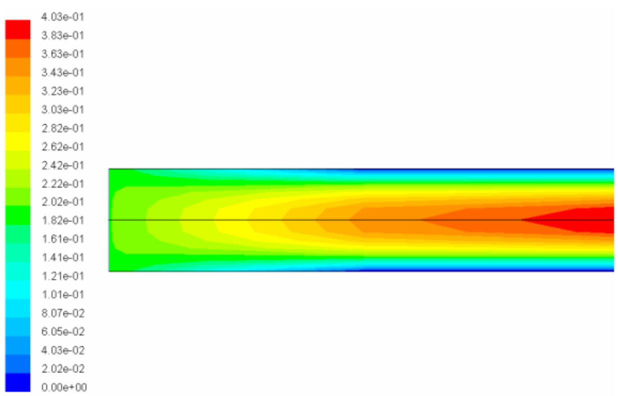

Figure 7. Velocity contour along the length of the Duct

4. CONCLUSION

In conclusion, our study delved into the intricacies of laminar flow and heat transfer in compact heat exchangers, particularly focusing on the influence of geometric variations on thermal performance. Through Computational Fluid Dynamics (CFD) simulations, we explored the behaviour of both straight and sinusoidal ducts, revealing significant insights into heat transfer enhancement and pressure drop characteristics. Our findings underscored the importance of geometric considerations, such as aspect ratio and channel shape, in optimising heat exchanger performance for dense media applications. Moreover, the established correlations between key parameters provide valuable guidelines for designing efficient heat exchangers tailored to specific operational requirements. Overall, this research contributes to advancing the understanding and design capabilities of printed circuit heat exchangers, facilitating the development of more efficient and effective thermal processing systems.

REFERENCES

[1]. R.F. Service, Fossil power, guilt-free, Science (356) (2017) 796–799.

[2]. L. Irwin, Y.L. Moullec, Turbines can use CO2 to cut CO2, Science (356) (2017) 805–806.

[3]. B.D. Iverson, T.M. Conboy, J.J. Pasch, A.M. Kruizenga, Supercritical CO2 Brayton cycles for solar-thermal Energy, Appl. Energy (111) (2013) 957–970.

[4]. B. Yildiz, K.J. Hohnholt, M.S. Kazimi, Hydrogen production using high-temperature steam electrolysis supported by advanced gas reactors with supercritical CO2 cycles, Nuclear Technol. (155) (2005) 1–21.

[5]. S.J. Bae, J. Lee, Y. Ahn, J.I. Lee, Preliminary studies of compact Brayton cycle performance for small modular high temperature gas-cooled reactor system, Ann. Nuclear Energy (75) (2015) 11–19.

[6]. S.M. Besarati, D.Y. Goswami, E.K. Stefanakos, Developing a solar receiver based on compact heat exchanger technology for supercritical carbon dioxide power cycles, J. Solar Energy Eng. (137) (2015) 031018-1-8.

[7]. P. Hejzlar, M.J. Driscoll, J. Gibbs, Y. Gong, S.P. Kao, Supercritical CO2 Brayton Cycle for Medium Power Applications: Final Report. MIT-ANP-PR-117, 2007.

[8]. M.J. Li, H.H. Zhu, J.Q. Guo, K. Wang, W.Q. Tao, The development technology

and applications of supercritical CO2 power cycle in nuclear Energy, Solar Energy and other energy industries, Appl. Therm. Eng. (126) (2017) 255–275.

[9]. K. Brun, P. Friedman, R. Dennis, Fundamentals and Applications of Supercritical Carbon Dioxide Based Power Cycles, Woodhead Publishing, United Kingdom, 2017.

[10]. Y.L. Moullec, Conceptual study of a high-efficiency coal-fired power plant with CO2 capture using a supercritical CO2 Brayton cycle, Energy (49) (2013) 32–46.

[11]. C. Turchi, A composite that takes the heat, Nature (562) (2018) 346–347.

[12]. M. Caccia, M. Tabandeh-Khorshid, G. Itskos, Ceramic-metal composites for heat exchangers in concentrated solar power plants, Nature (562) (2018) 406–409.

[13]. I.H. Kim, H.C. No, J.I. Lee, B.G. Jeon, Thermal hydraulic performance analysis of the printed circuit heat exchanger using a helium test facility and CFD simulations, Nuclear Eng. Des. 239 (2009) 2399–2408.

[14]. S.K. Mylavarapu, X. Sun, R.E. Glosup, RN Christensen, MW. Patterson, Thermal hydraulic performance testing of printed circuit heat exchangers in a high-temperature helium test facility, Appl. Therm. Eng. (65) (2014) 605–614.

[15]. T. Ishizuka, Y. Kato, Y. Muto, K. Nikitin, T.L. Ngo, Thermal-hydraulic characteristics of a printed circuit heat exchanger in a supercritical CO2 loop, Bull. Res. Lab. React. (30) (2006) 109–116.

[16]. J.H. Park, J.G. Kwon, T.H. Kim, M.H. Kim, J.E. Cha, H.J. Jo, Experimental study of a straight printed circuit heat exchanger on supercritical CO2 near the critical point with water cooling, Int. J. Heat Mass Transf. 150 (2020) 119364.

[17]. W.X. Chu, X.H. Li, T. Ma, Y.T. Chen, Q.W. Wang, Experimental investigation on sCO2-water heat transfer characteristics in a printed circuit heat exchanger with straight channels, Int. J. Heat Mass Transf. 113 (2017) 184–194.

[18]. Z. Ren, C.R. Zhao, P.X. Jiang, H.L. Bo, Investigation on local convective heat transfer of supercritical CO2 during cooling in horizontal semicircular channels of printed circuit heat exchanger, Appl. Therm. Eng. 157 (2019) 113697.

[19]. J.W. Seo, Y.H. Kim, D. Kim, Y.D. Choi, K.J. Lee, Heat transfer and pressure drop characteristics in straight microchannel of printed circuit heat exchangers, Entropy 17 (2015) 3438–3457.

[20]. A.M. Kruizenga, Heat Transfer and Pressure Drop Measurements in Prototypic Heat Exchangers for the Supercritical Carbon Dioxide Brayton Power Cycles, University of Wisconsin-Madison, USA, 2010.