27th ABCM International Congress of Mechanical Engineering
December 04-08, 2023, Florianópolis-SC, Brazil

COB-2023-0937
NUMERICAL MODELING AND QUALITATIVE APPROACH FOR GEOMETRIC DESIGN OF AN ULTRACOMPACT PRINTED CIRCUIT HEAT EXCHANGER FOR SUPERCRITICAL CO2

Bruno Henrique de Camargo Moreno
São Paulo's State University (UNESP), Brasil Sul Avenue, 56, City of Ilha Solteira, State of São Paulo, Brazil.
bruno.moreno@unesp.br

Jurandir Itizo Yanagihara
University of Sao Paulo (USP), Professor Mello Moraes Avenue, 2231, City of São Paulo, State of São Paulo, Brazil.
jiy@usp.br

Leandro Oliveira Salviano
São Paulo's State University (UNESP), Brasil Sul Avenue, 56, City of Ilha Solteira, State of São Paulo, Brazil.
leandro.salviano@unesp.br

Abstract. *The advancements in computational capabilities and in Computational Fluid Dynamics techniques have allowed the design of higher efficient heat exchangers for several applications. Moreover, there is an increasing demand on the part of industry to develop ecologically friendlier equipment, based on increments in performance. This work proposes the design of an ultra-compact heat exchanger, which will operate with a CO2/CH4 mixture near the supercritical point, considering a compressible, turbulent, and tridimensional flow. The goal is to develop an ultracompact heat exchanger to be installed in a compressor chain to improve the oil recovery process. In the current stage of project development, a qualitative analysis of the geometric shape of the heat exchanger waves is carried out, as a basis for future geometric modifications in the design of the final heat exchanger. Subsequently, the numerical model will be validated, and an optimization procedure will be developed considering a 3D-CFD modeling coupling methods for metamodeling and optimization. Existing CFD codes will be modified to include advanced models for calculating thermodynamic properties and turbulent flows to simulate the heat exchanger process and the fluid dynamics of the supercritical CO2. A sequential approach of Sensitivity Analysis methods using a screening method (Elementary Effects) followed by a quantitative method (Sobol Indices) to reduce the number of input parameters for optimization will be implemented, permitting large models to be optimized at lower computational costs. The main characteristics of dynamic flow and heat transfer will be investigated in detail, in order to develop and design an optimized ultracompact heat exchanger.*

Keywords: *Supercritical CO2; Heat transfer intensification techniques; Ultracompact plate heat exchanger; Optimization; Numerical simulation.*

1 INTRODUCTION

Supercritical CO2 has diverse applications, including surface leaching, chemical manufacturing, and use as a solvent in the chemical and food industries, replacing more environmentally impactful hydrocarbons. CO2 is cost-effective, chemically stable, non-flammable, and widely available, making it an attractive choice for various applications. Its accessible supercritical state, compared to other fluids like water, reduces compression power requirements in energy generation cycles, resulting in smaller machinery and components like boilers and heat exchangers, thus lowering operational costs (Marchionni et al., 2020). Carbon dioxide was one of the first fluids to be used as a refrigerant and has recently drawn the attention of the academic community and industry to its potential as a refrigerant fluid to replace chlorofluorocarbons and hydrofluorocarbons in the areas of refrigeration, air conditioning and heating systems (Chai et al., 2020). This potential is due to the properties of CO2 and its characteristic of being economic and environmentally sustainable.

Moreover, CO2 can be used in the oil recovery process (EOR) in extraction reservoirs. Large volumes of oil remain in the reservoirs when conventional oil extraction methods are used for this task. The recovery process of this remaining oil fraction can be improved by using the insertion of pressurized CO2 in a supercritical state on the reservoir, so that it extracts hydrocarbons from the oil to make it more easily miscible (Orr & Taber, 1984). The mechanisms to increase the oil extraction rate from reservoirs through the insertion of CO2 can occur by the increase of pressure in these reservoirs, through the reduction of the viscosity and density of the oil, as well as through the vaporization of petroleum hydrocarbons

(Perera et al., 2016). This process is named CO2-Enhanced Oil Recovery (CO2-EOR). It has emerged to capture CO2 from industrial activity, for example, which would be dumped into the atmosphere. This technology has great potential for application in the oil sector, since the market for using CO2 for oil recovery is vast, in addition to the fact that the revenues from the sales of captured CO2 emissions can also greatly accelerate the development process of this power generation industry, as well as the development of technologies to promote sustainability (Kuuskraa et al., 2013).

Several studies from literature have been modeling the CO2-EOR process. Ghanad Dezfully et al. (2015) treat the modeling of CO2 insertion into oil reservoirs as a mixture of gases under a supercritical state. It should be noted that, as it is a process of gas reinjection into oil reservoirs, this gas may be in a state of mixture with CH4. This is an aspect of CO2-EOR process modeling that will be addressed in this present study.

The work on which this project is focused is an application implemented in CO2-EOR systems. Supercritical CO2 mixed with CH4 is treated through a compression line for reinjection into oil reservoirs. An ultracompact heat exchanger is designed as intercoolers. Pei et al. (2014) present a representation of the compression line on which this work is based.

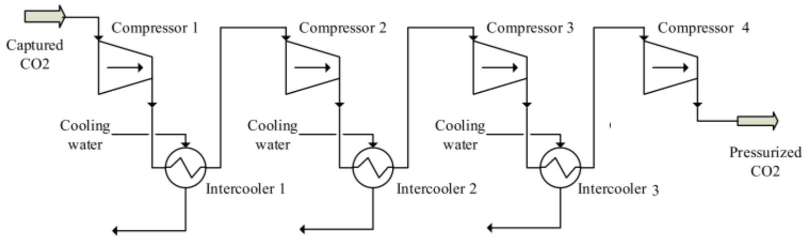


Figure 1. Intercooling compression strategy scheme (P. Pei et al., 2014).

Captured CO2 is directed to compressors and passes through cooling stages in compact heat exchangers. Research in the literature has extensively explored heat transfer phenomena involving supercritical CO2 (sCO2). Cheng et al. (2008) conducted a comprehensive analysis of heat transfer and pressure drop based on experimental data, providing correlations for cooling sCO2 in both macro and micro-channels. They discussed the physical and transport properties of CO2 at supercritical conditions and their impact on the heat transfer process. Huai et al. (2005) present experimental data about the flow and heat transferring characteristics of supercritical CO2 in horizontal circular channels, developing a new correlation for forced convection inside tubes for sCO2 under cooling conditions. Huang et al. (2016) present a review on the convection heat transfer process for water, carbon dioxide and hydrocarbon fuels at supercritical pressure, focusing on the thermophysical properties of fluids at supercritical pressures and relevant parametric effects, such as heat flux and pressure drop. Li and Yu (2021) present a review on the use of CO2 in heat exchangers, operating near the supercritical and transcritical points, presenting heat transfer characteristics involving sCO2, analyzing in detail this process in heat exchangers, with CFD simulations and experimental data, and developing an approach for a function to calculate sCO2 thermo-physical property constants in terms of temperature.

Wahl et al. (2021) investigated the heat transferring involving supercritical CO2 in a horizontal tube heat exchanger, with an internal diameter of 2 mm. The conclusion of this work points to a strong dependency between sCO2 heat transferring and fluid pressure and temperature. It was also identified that the cooling water temperature leads to a trade-off in heat transfer performance, and the CO2 mass flux influences the heat transfer substantially more near the pseudocritical temperature. Pei et al. (2019) demonstrate the benefits of using ultracompact heat exchangers in the heat treatment of sCO2, revealing the flow and heat transfer characteristics of SCO2 in a honeycomb ultracompact plate heat exchanger (UCPHE). Figley et al. (2013) present a numerical and thermohydraulic study of a type of ultracompact heat exchanger, which is named Printed Circuit Heat Exchanger (PCHE) and can be classified in the category of Ultracompact heat exchangers according to its high heat transfer density per area and reduced volume.

Figure 2 presents the base geometry model of the ultracompact heat exchanger of this project (HUANG et al., 2019).

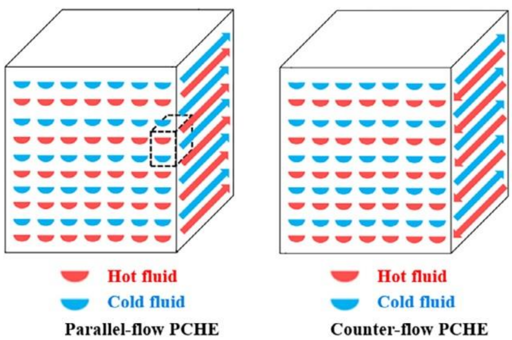


Figure 2. Schematic of an ultracompact PCHE-type heat exchanger (C. Huang et al., 2019).

The primary objective of this research is to design an optimal ultracompact heat exchanger for use in the gas compression process on an offshore gas-oil platform. This heat exchanger is specifically designed for the CO2/CH4 mixture as a working fluid, operating close to its supercritical point. The need for an ultracompact heat exchanger in offshore gas-oil applications arises from the fact that the centrifugal compressor has a temperature operating limit. The innovative aspect of this project lies in combining numerical modeling (CFD) with an optimization process using metamodeling. This approach aims to design efficient equipment with enhanced heat transfer capabilities while maintaining low to moderate pressure drop. Moreover, this research addresses the challenge of working with CO2 at its low critical point in heat transfer processes, representing a critical technological focus. The study explores the use of CO2 at its supercritical point and aims to overcome the limitations associated with its low critical point.

2 ULTRACOMPACT HEAT EXCHANGER DESIGN

Heat exchangers are used to transfer thermal energy between fluids at different temperatures through solid surfaces (Incropera et al., 1996). These devices vary in size and operate across a range of temperatures, making them versatile for numerous applications. They can exhibit mechanical properties suitable for high-pressure and high-temperature operations. Heat exchangers find use in diverse settings, from complex industrial processes, including those involving phase changes, to everyday equipment. They also play a crucial role in maintaining thermal comfort in environments, meeting strict efficiency and maintenance requirements to enhance cost-effectiveness and overall performance.

Compact Heat Exchangers are characterized by their density, denoted by the ratio of the heat transfer surface area to their volume, coupled with a significant heat transfer coefficient between the fluids (Q. Li et al., 2011). Therefore, these heat exchangers occupy less space, have a lower weight, and require smaller support structures when compared to traditional heat exchangers such as the shell-tube type. Researchers stated that two main factors drove the development of this type of heat exchangers. First, the industry's need for smaller and more thermally efficient equipment (e.g., the chemical and electronics industry), and second, the evolution of materials science, which allowed the fabrication of smaller objects, with high precision, low cost, and large quantities (Kew & Reay, 2011).

Other equipment has been developed based on the same concept of a compact heat exchanger with high volumetric density and high heat transfer rates. In recent years, Printed Circuit Heat Exchangers (PCHE) have been studied and evaluated as a promising option. In direct comparison with conventional heat exchangers, PCHE is more compact and has a firmer heat transfer core, which ensures stability and safety, due to the mechanical and structural properties given to the equipment by the characteristic of its compact geometry (Cui et al., 2018; Fopah-Lele et al., 2016). PHCE have characteristics of high heat transfer ability and can operate with high-pressure levels in their working fluid (C. Huang et al., 2019). Ultracompact heat exchangers have been studied as an alternative to increase heat transfer in systems in which equipment with reduced volume and high volumetric density are required (Liu et al., 2016; Nikitin et al., 2006). These qualities of compaction and high volumetric density give the PCHE wide applicability in the industry since these characteristics make it possible for the equipment to have high heat transfer capacities, which allows the miniaturization of circuits.

3 MESHING, NUMERICAL MODELING, VALIDATION METHODOLOGY AND A QUALITATIVE APPROACH

As referenced, this work proposes the design, development, and numerical evaluation of an ultracompact heat exchanger, with water as the cooling fluid of a CO2/CH4 mixture. A three-dimensional geometry design will be developed in the software ANSYS Design Modeler. Along with other geometric aspects of this model, these variables can be parameterized to evaluate their influence on the heat exchanger performance in a sensitivity analysis and optimization process, to obtain an optimal geometry that balances heat transfer maximization and pressure drop minimization. Different tube formats will be evaluated and compared, considering their effects on flow phenomena to determine the most efficient option. The initial focus was on creating a numerical model within a base geometry, validating the numerical approach, and ensuring that the mathematical conditions used for the final heat exchanger model were consistent with established references in the scientific literature.

The initial geometry and the validation data were based on the work conducted by Baik et al. (2017). They conducted a study on a printed circuit heat exchanger (PCHE) operating with CO_2 and water under various CO_2 thermodynamic states, including the supercritical state ($T_c = 30.98$ °C, $P_c = 7.38$ MPa). The study focused on developing and validating a PCHE design methodology for sCO2 applications, assessing the PCHE's performance under different CO2 states, and establishing friction factor and heat transfer correlations using experimentally validated results obtained through computational fluid dynamics (CFD). The PCHE's geometry involved semi-circular hot and cold flow tubes separated by a metal plate, with identical dimensions for both cold and hot flow tubes (Baik et al., 2017).

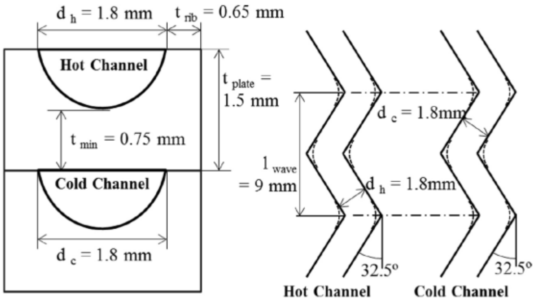


Figure 3. Geometry of the PCHE used for validation (Baik et al., 2017).

In this way, the preliminary geometry elaborated for this task consists of an ultracompact heat exchanger model of semicircular tubes, with dimensions of diameter d_c=1.8 mm, with longitudinal shape of zigzag tubes, built with an inclination in relation to an orthogonal reference of 32.5º, with a wavelength of 9 mm applied to each section, composed of 22 wave partitions of zigzag section along its longitudinal length and a height of 3 mm in its cross section used for calculation, which consists of a set of two semicircular tubes, the upper tube dedicated to the hot fluid flow (CO2) and the lower tube dedicated to the coolant fluid flow (H2O).

The test conditions presented by the authors were 26–43 ℃ and 7.3–8.6 MPa in temperature and pressure, respectively. In terms of non-dimensional numbers, the authors present the Reynolds number in a range of 15,000–100,000 and the Prandtl number in a range of 2–33. The authors incorporate boundary condition data obtained from experimental investigations on the ultracompact heat exchanger as parameters to define the boundary conditions for numerical simulations conducted using Computational Fluid Dynamics (CFD) techniques. The primary objective of the numerical simulations presented in the article is to discretize and analyze the flow and heat transfer phenomena within the ducts, as experimental measurements of such information were unattainable. In this regard, the authors present the mass flow rate data for both water and CO2, as well as the inlet and outlet temperature and pressure information obtained from experimental measurements. This data serves to provide context to the obtained results and guide the CFD calculations (Baik et al., 2017).

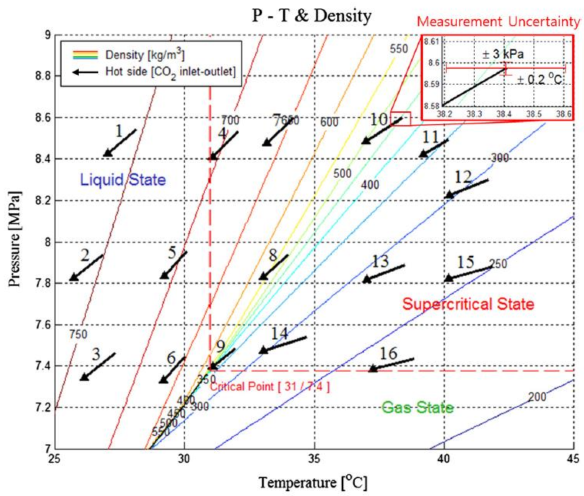


Figure 4. Inlet and outlet conditions for CO2 side (Baik et al., 2017).

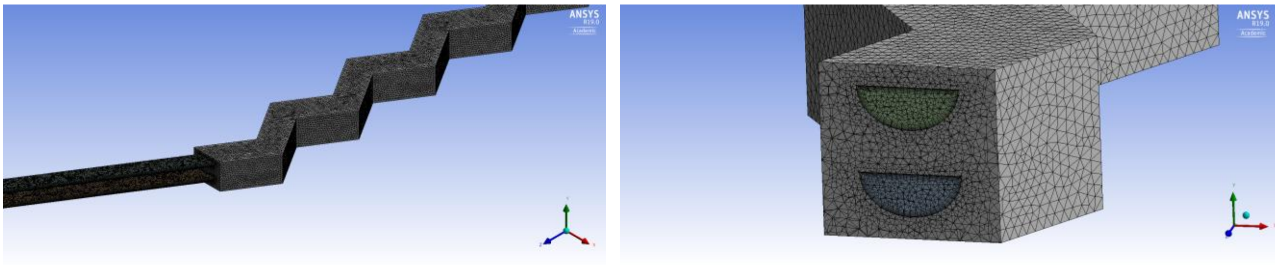
The boundary conditions presented in Figure 4 pertain specifically to experimentally measured data concerning the CO2 side. Each experiment conducted involved varying the mass flow rates of CO2 and H2O, resulting in significant changes to the thermodynamic inlet and outlet states of CO2. In contrast, for the water flow side, although the mass flow rate conditions varied in each case, the inlet temperature of water was maintained at a constant 15 ℃, and the flow on this side of the heat exchanger remained laminar.

Among the cases presented, the authors emphasize two specific cases, 7 and 15, for which the thermohydraulic characteristics of the model are evaluated through numerical CFD simulations based on the provided experimental boundary conditions. The reason is that both are the furthest cases from the pseudo-critical line within the supercritical state regime. Consequently, the properties of supercritical CO_2 behave similarly to normal gases for these cases, allowing the separation of the properties' effect (Baik et al., 2017).

The geometry employed for validation calculations was created using the ANSYS Design Modeler software, following the dimensions specified by Baik et al. (2017). Computational extensions were incorporated into the geometry to ensure an outflow condition for the flow in both tubes. The upstream extension of the primary domain was constructed with a length of 100 mm. This extension was implemented to ensure a uniform flow profile at the inlet of the primary domain, thereby mitigating the influence of purely numerical effects on the simulation results. The downstream extension of the primary domain was designed with a length of 400 mm. Its purpose is to prevent the occurrence of fluid recirculation or 'backflow,' which could introduce convergence instabilities in the numerical model. Ensuring the Neumann condition (outflow) is achieved is crucial for stable simulations.

For the present project, a hypothesis of steady-state and turbulent flow was adopted. Turbulence phenomena are addressed using the SST k-ω turbulence model. The SST k-ω turbulence model, a variant of the k-ω turbulence model proposed by Wilcox (1988), is employed as a two-equation transport model for solving the turbulent kinetic energy (k) and specific turbulent kinetic energy dissipation rate (ω). This model has demonstrated excellent performance for flows bounded by walls. The SST variant, known as Shear Stress Transport, combines Wilcox's original formulation for near-wall regions (in the viscous sublayer of the turbulent boundary layer) with the standard k-ε model (Launder & Spalding, 1974), which is suitable for flow calculations in regions further away from the wall, encompassing turbulent regions of the flow. The coupling function effectively bridges these two formulations. As referenced, the k-ω model excels in modeling flows near walls subject to adverse pressure gradients, boundary layer separations, and recirculation areas. To accommodate the high Reynolds numbers, present in the flow studied for numerical validation in this work, the computational mesh was highly refined to adequately capture heat transfer and CO2 flow phenomena in the numerical simulation.

The pressure-velocity coupling model used for this calculation was the Coupled scheme in ANSYS Fluent. This scheme allows for the simultaneous solution of the Navier-Stokes equations for fluid velocity and the continuity equation for pressure, considering the interactions between these two variables. In the pressure-velocity coupling of ANSYS Fluent, the Navier-Stokes equations are discretized and solved together with the continuity equation using an implicit iteration algorithm. In this method, the momentum and continuity equations are coupled through a linear system that is iteratively solved until a converged solution is achieved. The pressure-velocity coupling in the "Coupled" scheme ensures a consistent and accurate solution by considering the interdependency of pressure and velocity. By resolving the momentum and continuity equations simultaneously, the scheme allows for a more robust and converged solution.



(a) (b)

Figure 5. Geometry and mesh for the main domain and computational extensions (a) and Sectional cut of the main domain mesh highlighting local prismatic mesh refinement for boundary layer region (b).

The computational mesh was constructed using tetrahedral volumetric elements. Although hexahedral elements are a more computationally efficient choice, they can compromise mesh quality, especially at the high Reynolds numbers in CO2 flow. To address this, we applied local refinement with prismatic elements, incorporating 7 layers of prisms for Total Thickness inflation, resulting in elements smaller than 1.10^-4 mm in this region. Mesh quality was assessed using orthogonal quality and skewness metrics. The mesh achieved a minimum orthogonal quality of 0.20 and a maximum skewness of 0.79, both considered good. The total number of mesh elements was 20,805,484 million elements. Figure 5 illustrates a mesh cutout in the primary heat exchanger domain, highlighting the prismatic refinement used for boundary layer characterization. The top and bottom sides of the main domain are treated as periodic, and the sides of the domain are treated as adiabatic walls. The inlet condition is the velocity. The outlet boundary condition is stipulated as the Neumann condition, or outflow condition. The side boundaries of the domain are treated as no-slip walls.

3.1 A QUALITATIVE APPROACH

At the current development stage of the project, analyses are focused on conducting numerical validation of the calculations and simulations performed within the devised geometry and mesh, in line with the proposed mathematical modeling. The objective of this validation process is to obtain quantitative and qualitative results that align with a consistent physical phenomenology for the problem. Thus, having conducted a numerical simulation of the geometry under identical boundary conditions, the thermohydraulic parameters obtained should be compared with validated results of the literature, which are calculated from correlations, and these should lie within a certain percentage margin relative to these results.

The authors selected for the validation process provide appropriate numerical correlations for the proposed working range of the heat exchanger operation. Friction factor and heat transfer correlations for Nusselt were developed based on experimental data and computational analysis for the future PCHE design. Therefore, using the experimental and numerical data gathered by the authors of the base validation study, it is feasible to conduct the numerical validation of the work robustly and adequately.

Presented below are the numerical correlations for Nusselt number and Friction Factor used for comparing the numerical results of this study.

Table 1. Correlations of Nusselt Number and Friction Factor for the numerical validation (Baik et al., 2017).

	Water (50 < Re < 200)	CO_2 (15000 < Re < 85000)
Friction Factor	$f = 6.9982\ Re^{-0.766}$	$f = 0.0748\ Re^{-0.19}$
Nusselt Number	$Nu = 0.2829\ Re^{0.6686}$	$Nu = 0.8405\ Re^{0.5704}\ Pr^{1.08}$

Initially, calculations were conducted based on boundary parameters closely aligned with those presented in Figure 4. The intent of these calculations was to study the numerical convergence of the mathematical model, ensuring appropriate stabilization thereof, adjusting solution relaxation parameters when necessary. The authors of the work upon which this project's numerical validation is based state that they used the input parameters from tests 7 and 15 as the foundation for developing their numerical correlations. Test 1 describes the scenario with the lowest mass flow rates, while test 16 characterizes the case with the highest mass flow rate.

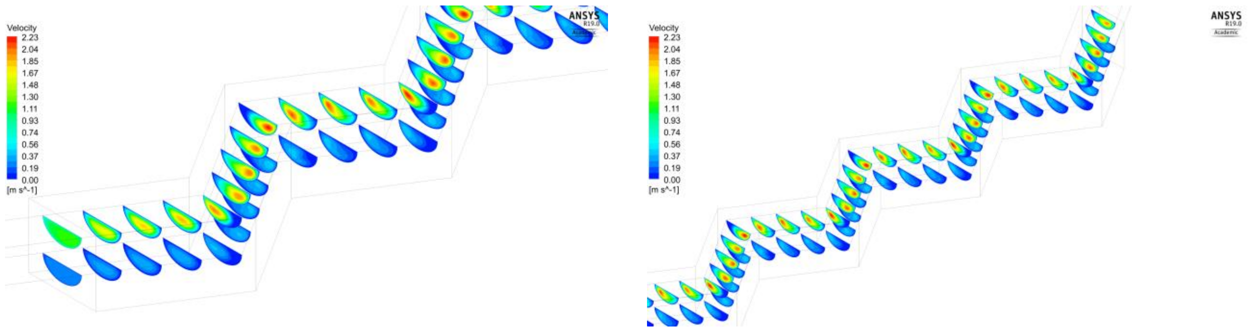
To verify the model's convergence robustness, an initial calculation was conducted using the following boundary conditions: an inlet velocity for CO_2 of 1.0 m/s, an inlet velocity for liquid H_2O of 0.2 m/s, an inlet temperature for CO_2 of 307.32 K, and an inlet temperature for water of 288.15 K. The temperature parameters were sourced from the paper by the authors chosen for validation (Baik et al., 2017). In their study, the authors stated that the water inlet temperature for all tests remained constant at 15°C. The CO_2 temperature, meanwhile, was extracted from case 7. The chosen inlet and outlet velocities were arbitrary, although they are close to the inlet flow velocities calculated from the data of case 7 presented, as this simulation aimed to qualitatively assess the validity of the results derived from a numerical simulation and modeling on the designed geometry.

The flow direction adopted for this initial simulation was parallel flow, because it is possible to observe the dynamics of both flows from the same inlet point, simplifying the comparison of their phenomenology.

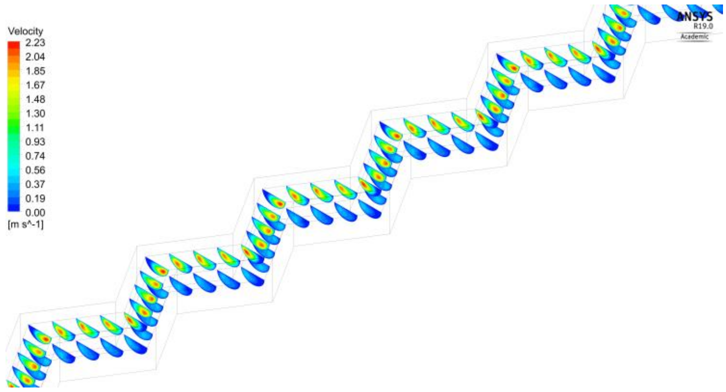
For the proposed qualitative analysis, the ultracompact heat exchanger domain was segmented into several subdomains by cross-sectional partitioning. This methodology was carried out by establishing transverse planes that intersect the heat exchanger's geometry, capturing the intersections of these planes with the tubes. A total of 176 partitions of the primary domain were selected as adequate for characterizing the flow phenomena and for subsequent thermohydraulic computations. These subdomains were established by segmenting the primary domain with 177 transverse planes. The subdivision criterion aimed to create subparts of the main domain with a length that was less than one-hundredth of the entire domain. The design involved dividing each wave of the heat exchanger into 8 sections, facilitated by the construction of 9 planes per wave to define them. Consequently, with a computational domain measuring 199.76 mm in length, and each wave spanning 9.08 mm, representing a domain with 22 waves, each of the segmented subdomains has a length of 1.135 mm. This corresponds to approximately 0.57% of the heat exchanger's total length.

This approach will also be beneficial for the quantitative analysis, since each of these planes will serve as capture points for properties, especially temperature and pressure. Using this method of segmenting the analysis domain into subdomains, a comparison of Nusselt and Friction Factor with values obtained from literature correlations will also be made to validate the numerical modeling. The importance of this methodology is emphasized, especially for the quantitative phase, given that the PCHE operates with supercritical CO2, in which abrupt property variations occur, such as Cp (specific heat at constant pressure). Considering these variations and given the total extent of the numerical domain, a traditional calculation approach, which assumes that these properties remain constant, becomes physically incompatible. Therefore, subdividing the larger computational domain into multiple subdomains is a strategy to mitigate the sensitivity to CO2 property variations.

Figure 6 depict visualizations of velocity profiles for the CO_2 and H_2O flows.



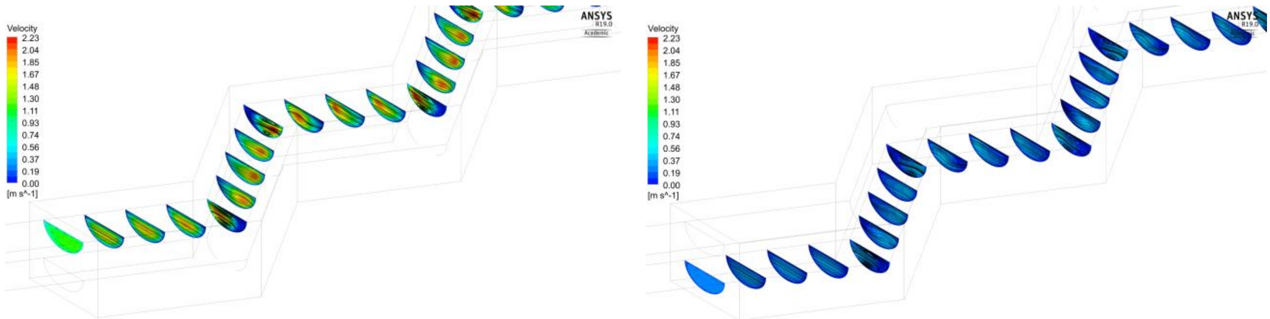
(a) (b)



(c)

Figure 6. Cross-sectional velocity profiles at the inlet portion of the ultracompact heat exchanger: (a) waves 1 and 2; (b) waves 9, 10, 11 and 12; (c) waves 19, 20, 21 and 22.

Figure 7 displays cross-sections in the inlet region of the heat exchanger domain, highlighting the flow streamlines of CO_2, in conjunction with the previously presented transverse velocity profiles, to illustrate the recirculation phenomena detected by the qualitative analysis.



(a) (b)

Figure 7. Cross-sectional velocity profiles and streamlines at the inlet portion of the ultracompact heat exchanger for (a) CO_2 and (b) H_2O flows.

The acute shape of the heat exchanger's waves notably influences the creation of recirculation regions within the flows. This effect is more pronounced as the flow velocity of the considered fluid increases, as observed when comparing both fluids. These recirculation regions (dark blue areas on the edges of the wave peaks) are formed both by the boundary layer detachment from the lower flow (which has been following the prior portion of the waves) and the potential collision of the flow with the walls defining the downstream part of the waves, which can generate recirculation in these regions. Practically, it can be asserted that such zones of recirculation and transverse vortices positively influence the flow's pressure drop, which is detrimental to the efficiency of systems in which such a heat generator is incorporated. This is because such losses may lead to an increased requirement for pumping potential of the fluids for the equipment's operation.

Figure 8 presents a solution to the potential increased pressure drop caused by the geometric feature of the sharp waves (Baik et al., 2017). This solution involves smoothing the curvature of the waves, allowing the flow to follow a regular path as it moves through them, without abrupt disruptions of contact between the fluid and the wall, which can create recirculation due to boundary layer detachment. It is noteworthy that, from a construction standpoint, the smoothed geometry is also more favorable compared to the sharp one, as it is easier to design and doesn't possess a geometric feature that could act as a stress concentrator.

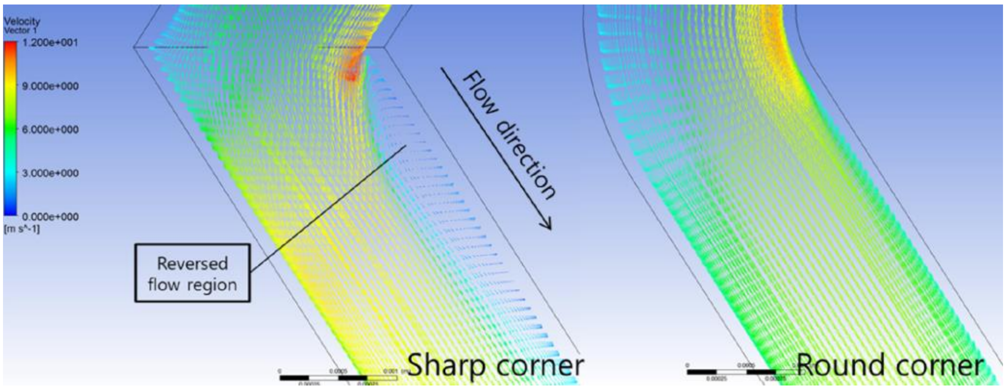


Figure 8. Differences in longitudinal velocity profiles resulting from modifications to the wave edges (Baik et al., 2017).

It is unequivocally evident that even in a flow with a lower velocity, such as the H_2O flow, the presence of sharp waves in the heat exchanger's geometry disturbs the flow, leading to the formation of vortices. To better characterize the type of vortices in these sections, a deeper analysis of them is essential, including a numerical examination of the heat exchanger's parameters of interest, to compute the effects of the vortex's generation on heat transfer and flow dynamics. Another pertinent comparison to be made, related to the solution provided by the authors of the reference validation work, is the juxtaposition of the same geometric cross-sections at identical points from two distinct wave shapes, one sharp and the other smoothed. This facilitates a comprehensive comparison, both numerical and qualitative, of all the effects and phenomena within these flow regions. Qualitatively deduced, the presence of smooth waves in heat exchangers can reduce pressure drops in their working fluids, rendering these devices more cost-effective to operate. Numerical aspects can validate this claim and may also shed light on the effects of geometric smoothing on heat transfer. Generally, it is anticipated that heat exchangers with sharp wave configurations would offer superior heat transfer capabilities, especially when examining these local wave portions independently. This is due to the previously mentioned phenomenon of boundary layer detachment in fluid-wall separation regions, resulting in a significant enhancement of the local convective heat transfer coefficient. Therefore, a forthcoming quantitative analysis can determine the most efficient geometry in terms of the trade-off between heat transfer and pressure drop. Such characterization could be based on a performance evaluation criterion selected for this analysis. As outlined above, it's essential to supplement this qualitative analysis with the numerical values corresponding to the validation of the mathematical model. In terms of the numerical model's convergence, it achieved stable convergence, attesting that the mesh characteristics and the set relaxation parameters for the analyzed case were both appropriate. For the next steps, numerical validation is essential, to compare the data derived from this model with values, correlations, and validated data from the literature, ensuring that the mathematical model is both robust and physically consistent.

4 CONCLUSIONS AND FURTHER WORK

In this study, the wave geometry format of an ultracompact printed circuit heat exchanger (PCHE) was qualitatively evaluated, representing a case of interest within the scope of development of an ultracompact heat exchanger for CO2-EOR process. This qualitative analysis serves as a comparative basis to guide potential geometric modification. The presented cases and other geometries will be intensively analyzed and will be subjected to an optimization process, with the aim of achieving a compact and thermally efficient heat exchanger design.

From the analyzed case, it can be concluded that sharp wave geometries in the heat exchanger potentially result in increasing quantity and intensity of boundary layer detachment zones, which lead to the generation of transversal vortices. These vortices contribute to elevating the pressure drop of the heat exchanger's working fluids. This heightened pressure drop adversely affects the system's operational cost as it leads to an increased requirement for fluid pumping work.

Heat exchangers with smoothed waves exhibit fewer recirculation zones since their geometry promotes smoother flow without abrupt detachments of fluid-wall contact. This feature means that smoothed waves positively influence the reduction of flow pressure drops and decrease the system's operational cost.

However, it's imperative to emphasize that vortex-generating elements can positively impact heat transferring. If longitudinal vortices are formed, they can act as enhancers of flow mixing, boosting the local convective heat transfer coefficients. Another factor contributing to a local increase in convective heat transfer coefficient is the boundary layer detachments themselves. They disrupt developed boundary layers, resulting in heat transfer spikes in these regions. Given this consideration, a quantitative approach becomes essential to compare the thermohydraulic performance of heat exchangers with smooth waves versus those with sharp waves. This comparison aims to determine which wave type offers a better cost-benefit in terms of its heat transfer capability relative to the pressure drop caused by its geometry.

Hence, to reinforce the presented conclusions and validate the numerical model, a comprehensive quantitative analysis of the heat exchanger, focusing on relevant thermohydraulic parameters, will be undertaken as the immediate next step in the proposed work's development, using selected experimental and numerical data from Baik et al. (2017). To achieve this, the thermohydraulic parameters of interest will be calculated and compared with the validation data to ensure a good level of agreement. This comparison will demonstrate that the simulation results accurately represent the real physical phenomena, thus establishing the reliability and robustness of the implemented computational model.

The thermohydraulic parameters to be analyzed based on the calculated simulation data are presented in the equations below.

$$Re=\frac{\rho\, u\, Dh}{\mu} \tag{1}$$

$$qw=\frac{1}{A}\int_A q_w\, dA \tag{2}$$

$$Tw=\frac{1}{A}\int_A T_w\, dA \tag{3}$$

$$Tb=\frac{\int_A c_p\, \rho\, u\, T\, dA}{\int_A c_p\, \rho\, u\, dA} \tag{4}$$

$$h=\frac{qw}{Tw-Tb} \tag{5}$$

Where Re is the Reynolds number, Dh is the hydraulic diameter of the channel, μ is the dynamic viscosity, ρ is the density of the fluid, u is the mass-weighted average velocity, qw and Tw are the average heat flux of the fluid. wall and the temperature of the liquid-solid interface, A is the internal surface area of the interface, Tb is the bulk temperature.

The Nusselt number, used for the dimensionless evaluation of the heat transfer in the system, will be described below, as well as the Fanning Friction factor, which will be used for the dimensionless evaluation of the fluid pressure drop.

$$Nu=\frac{h\, D_h}{\eta\, \lambda_f} \tag{6}$$

$$f=\frac{\Delta p_f\, D_h}{2\, \rho\, u^2\, l} \tag{7}$$

Subsequently, a mesh sensitivity analysis will be performed using the Grid Convergence Index (GCI) methodology proposed by Celik et al. (2008) to ensure the results' independence from the number of elements. This mesh refinement process aims to generate a robust mesh with an appropriate number of elements, which can constitute a representative numerical model of the physical phenomenon, while maintaining a manageable computational cost. The implementation of the sensitivity analysis for the model will be carried out in the next stage by performing a preliminary Design of Experiment (DoE) to evaluate the sensitivity of the system. The Response Surface Methodology (RSM) will then be trained and tested, and a parametric sensitivity analysis will be conducted using RSM. Finally, the evaluation of the optimized ultracompact heat exchanger will be conducted as the outcome of the development process.

5 BIBLIOGRAPHY

Baik, S., Kim, S. G., Lee, J., & Lee, J. I. (2017). Study on CO2 – water printed circuit heat exchanger performance operating under various CO_2 phases for S-CO_2 power cycle application. *Applied Thermal Engineering*, *113*, 1536–1546. https://doi.org/10.1016/j.applthermaleng.2016.11.132

Celik, I. B., Ghia, U., Roache, P. J., Freitas, C. J., Coleman, H., & Raad, P. E. (2008). Procedure for estimation and reporting of uncertainty due to discretization in CFD applications. *Journal of Fluids Engineering, Transactions of the ASME*, *130*(7), 0780011–0780014. https://doi.org/10.1115/1.2960953

Chai, L., Tsamos, K. M., & Tassou, S. A. (2020). Modelling and Evaluation of the Thermohydraulic Performance of Finned-Tube Supercritical Carbon Dioxide Gas Coolers. *Energies*, *13*(5), 1031. https://doi.org/10.3390/en13051031

Cheng, L., Ribatski, G., & Thome, J. R. (2008). Analysis of supercritical CO2 cooling in macro- and micro-channels. *International Journal of Refrigeration*, *31*(8), 1301–1316. https://doi.org/10.1016/j.ijrefrig.2008.01.010

Cui, X., Guo, J., Huai, X., Cheng, K., Zhang, H., & Xiang, M. (2018). Numerical study on novel airfoil fins for printed circuit heat exchanger using supercritical CO2. *International Journal of Heat and Mass Transfer*, *121*, 354–366. https://doi.org/10.1016/j.ijheatmasstransfer.2018.01.015

Figley, J., Sun, X., Mylavarapu, S. K., & Hajek, B. (2013). Numerical study on thermal hydraulic performance of a Printed Circuit Heat Exchanger. *Progress in Nuclear Energy*, *68*, 89–96. https://doi.org/10.1016/j.pnucene.2013.05.003

Fopah-Lele, A., Rohde, C., Neumann, K., Tietjen, T., Rönnebeck, T., N'Tsoukpoe, K. E., Osterland, T., Opel, O., & Ruck, W. K. L. (2016). Lab-scale experiment of a closed thermochemical heat storage system including honeycomb heat exchanger. *Energy*, *114*, 225–238. https://doi.org/10.1016/j.energy.2016.08.009

Ghanad Dezfully, M., Jafari, A., & Gharibshahi, R. (2015). CFD Simulation of Enhanced Oil Recovery Using Nanosilica/Supercritical CO2. *Advanced Materials Research*, *1104*, 81–86. https://doi.org/10.4028/www.scientific.net/AMR.1104.81

Huai, X. L., Koyama, S., & Zhao, T. S. (2005). An experimental study of flow and heat transfer of supercritical carbon dioxide in multi-port mini channels under cooling conditions. *Chemical Engineering Science*, *60*(12), 3337–3345. https://doi.org/10.1016/j.ces.2005.02.039

Huang, C., Cai, W., Wang, Y., Liu, Y., Li, Q., & Li, B. (2019). Review on the characteristics of flow and heat transfer in printed circuit heat exchangers. *Applied Thermal Engineering*, *153*, 190–205. https://doi.org/10.1016/j.applthermaleng.2019.02.131

Huang, D., Wu, Z., Sunden, B., & Li, W. (2016). A brief review on convection heat transfer of fluids at supercritical pressures in tubes and the recent progress. In *Applied Energy* (Vol. 162, pp. 494–505). Elsevier Ltd. https://doi.org/10.1016/j.apenergy.2015.10.080

Incropera, F., DeWitt, D. P., Lavine, A. S., & Bergman, T. L. (1996). *Fundamentals of heat and mass transfer* (Vol. 6). Wiley.

Kew, P. A., & Reay, D. A. (2011). Compact/micro-heat exchangers - Their role in heat pumping equipment. *Applied Thermal Engineering*, *31*(5), 594–601. https://doi.org/10.1016/j.applthermaleng.2010.08.003

Kuuskraa, V. A., Godec, M. L., & Dipietro, P. (2013). CO2 Utilization from "Next Generation" CO2 Enhanced Oil Recovery Technology. *Energy Procedia*, *37*, 6854–6866. https://doi.org/10.1016/j.egypro.2013.06.618

Launder, B. E., & Spalding, D. B. (1974). The numerical computation of turbulent flows. *Computer Methods in Applied Mechanics and Engineering*. https://doi.org/10.1016/0045-7825(74)90029-2

Li, Q., Flamant, G., Yuan, X., Neveu, P., & Luo, L. (2011). Compact heat exchangers: A review and future applications for a new generation of high temperature solar receivers. *Renewable and Sustainable Energy Reviews*, *15*(9), 4855–4875. https://doi.org/10.1016/j.rser.2011.07.066

Li, W., & Yu, Z. (2021). Heat exchangers for cooling supercritical carbon dioxide and heat transfer enhancement: A review and assessment. *Energy Reports*, *7*, 4085–4105. https://doi.org/10.1016/j.egyr.2021.06.089

Liu, H., Yu, Q. N., Zhang, Z. C., Qu, Z. G., & Wang, C. Z. (2016). Two-equation method for heat transfer efficiency in metal honeycombs: An analytical solution. *International Journal of Heat and Mass Transfer*, *97*, 201–210. https://doi.org/10.1016/j.ijheatmasstransfer.2016.01.020

Marchionni, M., Bianchi, G., & Tassou, S. A. (2020). Review of supercritical carbon dioxide (sCO2) technologies for high-grade waste heat to power conversion. *SN Applied Sciences*, *2*(4), 611. https://doi.org/10.1007/s42452-020-2116-6

Nikitin, K., Kato, Y., & Ngo, L. (2006). Printed circuit heat exchanger thermal–hydraulic performance in supercritical CO2 experimental loop. *International Journal of Refrigeration*, *29*(5), 807–814. https://doi.org/10.1016/j.ijrefrig.2005.11.005

Orr, F. M., & Taber, J. J. (1984). Use of Carbon Dioxide in Enhanced Oil Recovery. In *New Series* (Vol. 224, Issue 4649).

Pei, B., Chen, Z., Li, F., & Bai, B. (2019). flow and heat transfer of supercritical CO2 in the honeycomb ultra-compact plate heat exchanger. *The Journal of Supercritical Fluids*.

Pei, P., Barse, K., Gil, A. J., & Nasah, J. (2014). Waste heat recovery in CO 2 compression. *International Journal of Greenhouse Gas Control*, *30*, 86–96. https://doi.org/10.1016/j.ijggc.2014.09.001

Perera, M. S. A., Gamage, R. P., Rathnaweera, T. D., Ranathunga, A. S., Koay, A., & Choi, X. (2016). A Review of CO2 -Enhanced oil recovery with a simulated sensitivity analysis. In *Energies* (Vol. 9, Issue 7). MDPI AG. https://doi.org/10.3390/en9070481

Wahl, A., Mertz, R., Laurien, E., & Starflinger, J. (2021). Heat transfer correlation for sCO2 cooling in a 2 mm tube. *Journal of Supercritical Fluids*, *173*. https://doi.org/10.1016/j.supflu.2021.105221

Wilcox, D. C. (1988). Reassessment of the scale-determining equation for advanced turbulence models. *AIAA Journal*. https://doi.org/10.2514/3.10041

6 RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.