

Numerical Modeling and Thermal Performance Evaluation of a Heat Exchanger Using Ethanol and Cold Water

Asif Ahmad¹ Deepak Awasthi²

^{1,2}Department of Mechanical Engineering

^{1,2}Pranveer Singh Institute of Technology, Kanpur, Uttar Pradesh, India

Abstract — This study presents a numerical investigation of heat transfer between ethanol and cold water flowing through a heat exchanger. A three-dimensional model of the heat exchanger was first prepared using Computer-Aided Design (CAD) software and subsequently used for numerical analysis. In the simulated configuration, cold water was supplied through the outer passage at a mass flow rate of 0.001 kg/s with an inlet temperature of 10 °C. At the same mass flow rate, ethanol was introduced into the inner passage at an initial temperature of 78 °C. Both fluids were allowed to leave the heat exchanger under atmospheric pressure conditions. The numerical analysis predicted an average outlet temperature of 25.67 °C for the cold water, whereas the ethanol reached an average outlet temperature of 43.64 °C. The obtained temperature variations demonstrate the heat exchange occurring between the two fluids and provide useful numerical data for evaluating and improving heat exchanger designs involving ethanol–water thermal systems.

Keywords: Heat Exchanger, Numerical Simulation, Computer-Aided Design (CAD)

I. INTRODUCTION

Heat exchangers are widely employed thermal systems that facilitate the transfer of heat between two or more fluids without necessarily allowing them to mix. They are essential components in numerous engineering and industrial processes, including power generation, refrigeration, chemical processing, and energy recovery. Depending on the required heat-transfer performance and operating conditions, heat exchangers can be developed in different geometrical configurations and flow arrangements to achieve effective thermal management [1]. Kareem et al. employed a two-phase flow model to investigate heat transfer characteristics in a counterflow shell-and-tube heat exchanger, comparing the thermal performance of various nanofluids with that of pure water. The shell was modelled as an enclosed cubic domain containing different working fluids, while the tube functioned as the primary heat exchange component. The study examined the effects of nanofluid composition, nanoparticle size, and volume fraction on heat transfer performance, aiming to enhance thermal exchange efficiency [2]. Power plants typically incorporate heat-exchange equipment, which can be quite complex in composition. The proposed method enables the calculation of the maximum heat flow rate, given that the thermodynamic properties of both heat carriers are known [3]. In the present research study, a numerical simulation of a counter-current double-pipe heat exchanger is performed to investigate the heat transfer characteristics and thermal behaviour of ethanol during the cooling process. The heat exchanger is designed to facilitate heat transfer between two fluids flowing in opposite directions, with hot ethanol passing through the inner pipe and cold water circulating through the outer pipe. The ethanol enters the heat exchanger at a temperature of 78°C, while the

cooling water enters at 10°C. The primary objective of this numerical investigation is to evaluate the thermal performance of the heat exchanger and predict the temperature distribution and heat transfer characteristics of both working fluids under the specified operating conditions. A computer-aided design (CAD) model of the double-pipe heat exchanger is developed and used for numerical simulation to examine the fluid flow and heat transfer behaviour within the system. The simulation provides insight into the temperature variation and velocity distribution of both ethanol and water along the length of the heat exchanger. A sectional cut plot of the CAD model is generated to visualise the flow characteristics, including the inlet and outlet velocities of both fluid streams. The counter-current flow arrangement enables heat transfer from the hot ethanol to the cold water, resulting in a gradual reduction in ethanol temperature and a corresponding increase in water temperature. The numerical simulation results indicate that the average outlet temperature of the cooling water reaches 25.67°C, representing a temperature rise of 15.67°C, whereas the average outlet temperature of ethanol decreases to 43.64°C, corresponding to a temperature reduction of 34.36°C. These findings demonstrate the heat exchange occurring between the two working fluids under the specified operating conditions. The numerical investigation provides useful information for design engineers in predicting the thermal behaviour of double-pipe heat exchangers and understanding the influence of counter-current flow on temperature distribution. Furthermore, the simulation approach can support the design, analysis, and optimisation of heat exchange systems for industrial applications involving ethanol cooling. **Figure 1** illustrates the step-by-step methodology adopted for the numerical simulation, including CAD model development, specification of boundary conditions, numerical analysis, and evaluation of the simulation results.

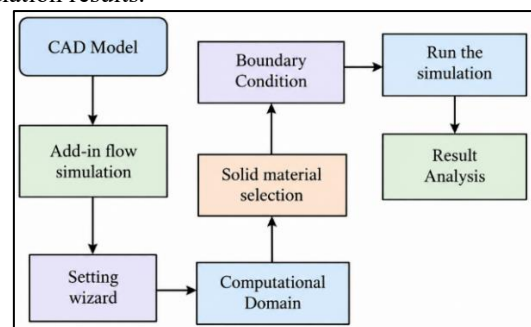


Fig. 1: Flow chart of numerical simulation procedure.

II. SIMULATION METHODOLOGY

A. Preparation of CAD Model

In this research study, a computer-aided design (CAD) model was developed utilizing CAD software as shown in Figure 2. The heat exchanger model comprises two pipes through

which two distinct fluids are circulated. The outer diameter and inner diameter of the outer pipe, through which cold water flows, are 60.452 mm and 52.501 mm, respectively. Conversely, the outer diameter and inner diameter of the inner pipe, which accommodates the flow of ethanol, are 42.164 mm and 35.052 mm, respectively.

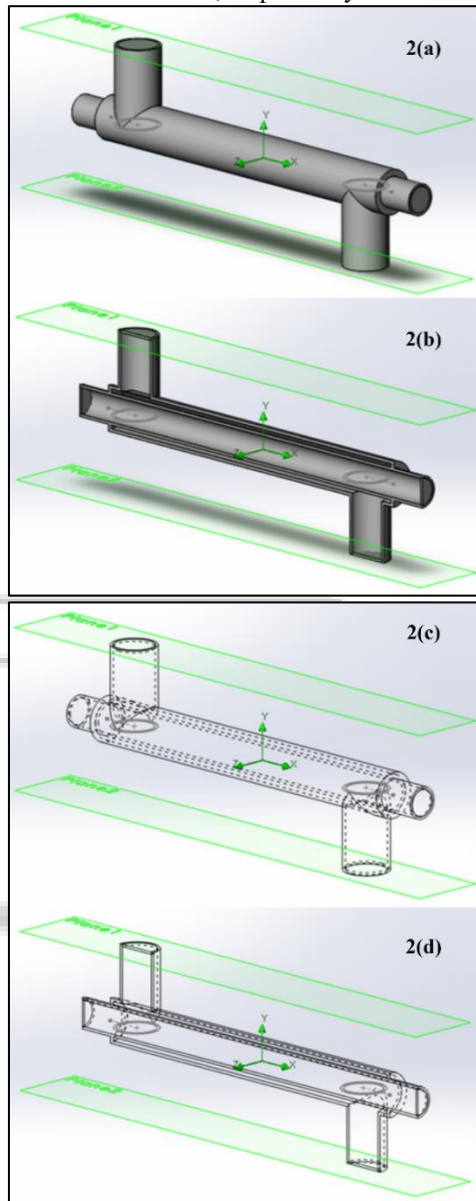


Fig. 2: (a) conceptual full section view, (b) conceptual half section view, (c) wire frame full section view and (d) wire frame half section view

B. Simulation setting

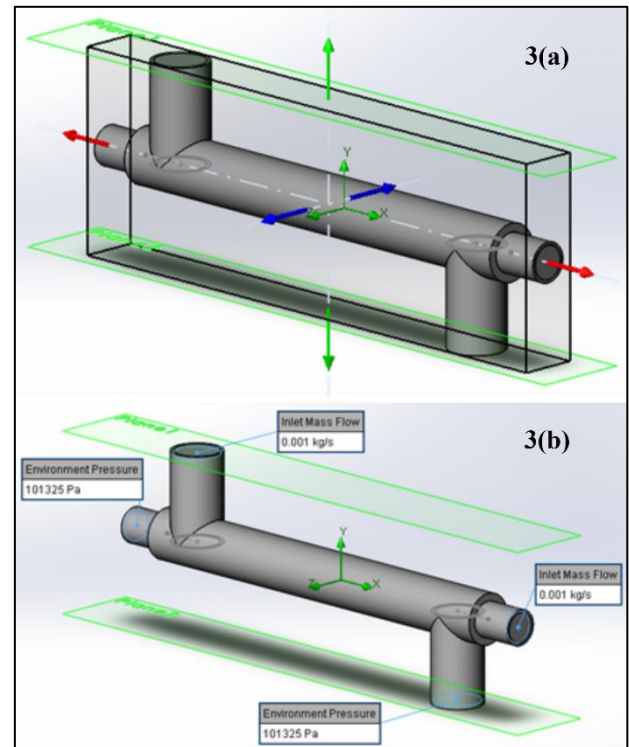


Fig. 3: computational domain (a) conceptual view, (b) surface parameter

Following the preparation of the CAD model, a flow simulation was incorporated using the appropriate add-in features of the software [4]. The computational domain for the simulation was established through a guided wizard within the CAD program as shown in Figure 3 (a). The material selected for the heat exchanger is copper, chosen for its excellent thermal conductivity properties. Subsequently, boundary conditions were defined for the simulation. Surface parameter i.e., inlet mass flow rate of ethanol was set to 0.001 kg/s, while the inlet mass flow rate of cold water was also established at 0.001 kg/s as shown in Figure 3 (b). The primary objective of the simulation is to determine the outlet temperatures of both the water and ethanol, allowing for an assessment of the heat exchange efficiency within the system. Following the development of the three-dimensional CAD model of the counter-current double-pipe heat exchanger, a computational fluid dynamics (CFD) analysis was performed using the flow simulation add-in integrated into the CAD software [4]. The numerical simulation was conducted to investigate the fluid flow behaviour, temperature distribution, and heat transfer characteristics of ethanol and cooling water within the heat exchanger. The simulation procedure involved defining the computational domain, assigning appropriate material properties, specifying the working fluids, establishing boundary conditions, and determining the parameters required to evaluate the thermal performance of the system. Initially, the computational domain was established using the guided simulation wizard available in the CAD software, as illustrated in Figure 3(a). The computational domain represents the region in which fluid flow and heat transfer are numerically analysed. In the present investigation, two separate fluid regions were considered: the inner pipe, through which hot ethanol flows,

and the outer annular passage, through which cold water circulates in the opposite direction. This counter-current flow arrangement facilitates heat transfer between the two working fluids through the separating pipe wall. The computational domain was configured to account for fluid flow and conjugate heat transfer, allowing the thermal interaction between the solid wall and the working fluids to be investigated. Copper was selected as the construction material for the heat exchanger because of its excellent thermal conductivity, which promotes heat transfer between the hot ethanol and cold water. The high thermal conductivity of copper reduces the conductive thermal resistance of the separating wall, facilitating the transfer of thermal energy from the ethanol to the cooling water. The material properties of copper were assigned to the solid components of the CAD

model to account for heat conduction through the pipe walls during the numerical simulation. After establishing the computational domain and assigning the material properties, the boundary conditions were specified to represent the operating conditions of the heat exchanger. The inlet temperature of hot ethanol was maintained at 78°C, while the inlet temperature of cold water was set to 10°C. The mass flow rate of ethanol at the inlet was specified as 0.001 kg/s, and an identical mass flow rate of 0.001 kg/s was assigned to the cooling water inlet, as illustrated in Figure 3(b). These boundary conditions were applied to investigate the heat transfer characteristics under equal mass flow rate conditions. The counter-current configuration was maintained by specifying opposite flow directions for the two fluids.

III. RESULT ANALYSIS

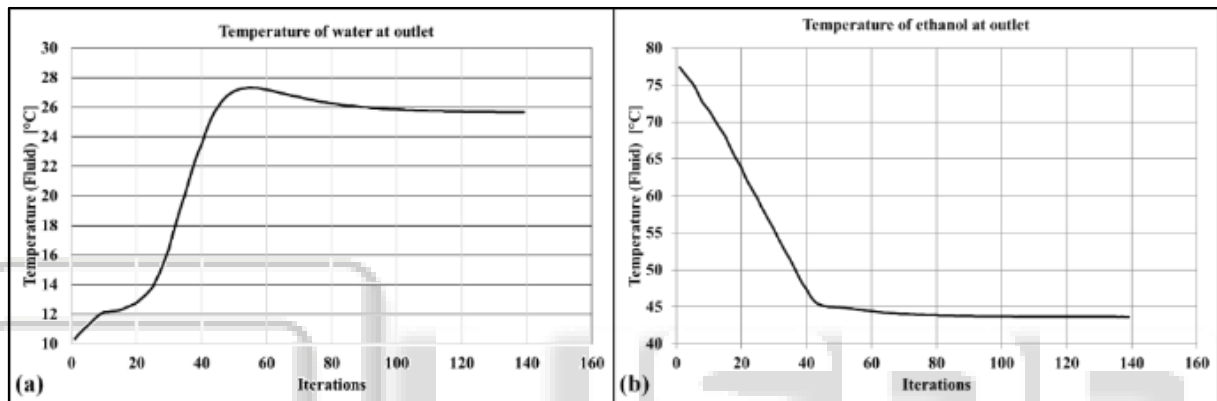


Fig. 4: temperature of water (a), (b) at the outlet.

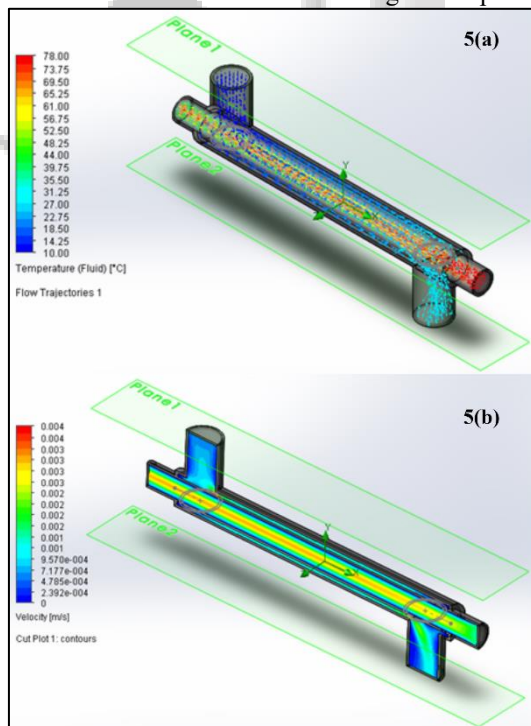


Fig. 5: Cut plot for (a) temperature & velocity (b) of fluid
The average outlet temperatures for water and ethanol are presented in Table 1. As illustrated in Figure 4 (a), the water temperature rises with each iteration, stabilizing after the 60th iteration. The predicted average outlet temperature is 25.67 degrees Celsius. As illustrated in Figure 4 (b), the temperature

of ethanol decreases steadily through the heat exchanger until the 40th iteration. Beyond this point, the temperature remains stable, with an average measured at 43.64 degrees Celsius.

Goal Name	Unit	Value	Average Value	Minimum Value	Maximum Value	Progress [%]
Temperature water outlet	°C	25.6668	25.7994	25.6668	26.1359	100
Temperature ethanol outlet	°C	43.6419	43.6901	43.6403	43.8159	100

Table 1: Temperature of water & ethanol at outlet

Figure 5(a) illustrates the flow trajectory plot generated from the simulation results. In this figure, a colour gradient is used to represent temperature variations, with red indicating the high-temperature ethanol entering the heat exchanger. Conversely, the light blue colour represents the water that is expelled after absorbing heat from the ethanol, resulting in a reduction of the ethanol's temperature due to the heat exchange process. Figure 5(b) presents a contour plot that visualizes the velocity distribution within the system. Here, the red colour indicates the highest velocities, while dark blue represents the lowest velocities. Light green denotes areas of average velocity, providing a clear understanding of how fluid flow varies throughout the system.

IV. CONCLUSION:

- 1) The temperature of the ethanol passing through the heat exchanger was determined to be 43.64°C.
- 2) The temperature of the water extracting heat from the ethanol as it flows through the heat exchanger was determined to be 25.67°C.
- 3) The average velocity of water at the outlet is 4.832×10^{-4} m/s, while the average velocity of ethanol is 0.001 m/s.

V. FUTURE SCOPE

The present study demonstrates the thermal behaviour of a counter-current double-pipe heat exchanger using numerical simulation, with ethanol and cold water operating at equal mass flow rates. The simulation provides useful information on the temperature distribution and outlet temperatures of both fluids under the specified operating conditions. However, the investigation can be further extended to improve the thermal performance and identify optimum operating conditions. In future work, Taguchi methodology can be employed to systematically investigate the influence of important process parameters such as ethanol inlet temperature, water inlet temperature, mass flow rates of both fluids, and heat-exchanger geometrical parameters [5-10]. An appropriate orthogonal array can be used to reduce the number of simulation runs while identifying the relative contribution of individual parameters to heat-transfer performance. Furthermore, Response Surface Methodology (RSM) can be integrated with numerical simulation to develop mathematical relationships between the selected input parameters and responses such as outlet temperature, heat-transfer rate, temperature difference, and thermal efficiency. RSM can also be used to determine the optimum combination of operating parameters through numerical experimentation. The optimized conditions obtained from Taguchi and RSM approaches can subsequently be evaluated using additional CFD simulations and, where feasible, validated experimentally [11, 12]. Future studies may also consider different flow rates, alternative working fluids, different pipe dimensions, wall materials, and flow arrangements to obtain a broader understanding of heat-exchanger performance. Since the present model uses copper because of its high thermal conductivity and applies equal mass flow rates of 0.001 kg/s for both fluids, these parameters provide suitable starting points for further parametric and optimization studies.

REFERENCE

- [1] Yuan, Y., et al., Numerical simulation and multi-objective optimization design of conjugate heat transfer in novel double shell-passes multi-layer helically coiled tubes heat exchangers. *Case Studies in Thermal Engineering*, 2024. 62: p. 105155.
- [2] Kareem, A.K., A.H. Alabbasi, and A.M. Mohsen, Simulation study of heat transfer behaviour of turbulent two-phase flow in a 3D cubic shell and tube heat exchanger using water and different nanofluids. *Results in Engineering*, 2024. 24: p. 102851.
- [3] Frid, S.E., A simplified method of calculating heat flow through a heat exchanger with a significantly temperature-dependent specific heat of heat transfer fluids. *Applied Thermal Engineering*, 2024. 257: p. 124349.
- [4] Kashi, E., F. Mirzaei, and F. Mirzaei, Analysis of gas dispersion and ventilation within a comprehensive CAD model of an offshore platform via computational fluid dynamics. *Journal of Loss Prevention in the Process Industries*, 2015. 36: p. 125-133.
- [5] Ahmad, A. and S. Alam, Integration of RSM with Grey based Taguchi Method for optimization of pulsed TIG welding process parameters. *Materials Today: Proceedings*, 2019. 18: p. 5114-5127.
- [6] Ahmad, A. and S. Alam, Parametric optimization of TIG welding using Response Surface Methodology. *Materials Today: Proceedings*, 2019. 18: p. 3071-3079.
- [7] Ahmad, A. and S. Alam. *PROCESS PARAMETER OPTIMIZATION OF PULSED TIG WELDING USING RSM*. 2019.
- [8] Ahmad, A., Salam, and P.R. Scholar. *A Review on Pulsed TIG Welding*.
- [9] Ahmad, A., *Application of Taguchi Method in Optimization of Pulsed TIG Welding Process Parameter. Quality Control - Intelligent Manufacturing, Robust Design and Charts*, 2020.
- [10] Ahmad, A. and S. Alam, Grey Based Taguchi Method for Optimization of TIG Process Parameter in Improving Tensile Strength of S30430 Stainless Steel. *IOP Conference Series: Materials Science and Engineering*, 2018. 404.
- [11] Ahmad, A., *Microstructure Analysis and Multi-objective Optimization of Pulsed TIG Welding of 316/316L Austenite Stainless Steel. Handbook of Smart Materials, Technologies, and Devices*, 2021.
- [12] Ahmad, A. and Tjprc. *Parametric Optimization of TIG Welding to Determine Welding Strength of S30430 Stainless Steel. in International Journal of Mechanical and Production Engineering Research and Development*. 2018.