Analysis of A Counter Flow Parallel-Plate Heat Exchanger

Ruoxu Jia and Junling Hu

Department of Mechanical Engineering, University of Bridgeport

Abstract

Heat exchangers are used widely in many industries for heat recovery or cooling purposes. This paper developed a numerical model to simulate a counter flow parallel heat exchanger. A representative repeating unit cell of the multi-channeled heat exchanger was taken as the computational domain, which includes a cold channel and a hot channel separated by plates. The model was simulated in COMSOL for an oil to water heat exchanger. Higher temperature oil and relatively lower temperature water entered two separate parallel channels in opposite directions. The detailed distributions of temperature, velocity, and pressure were used to analyze the performance of the heat exchanger. It was found the model can be used to provide guidance for designing an optimal heat exchanger.

Introduction

Currently, heat exchangers have a wide range of industry applications. They are widely used in space heating, refrigeration, power plants, petrochemical plants, petroleum refineries and sewage treatment. There are many types of heat exchanger designs for various applications. The major types of heat exchanger include double pipe, shell-tube, plate and shell, plate fin, and phase change heat exchangers. The flow in a heat exchanger can be arranged as parallel flow, counter flow, and cross flow. New heat exchangers have been designed for emerging thermal engineering fields, such as miniaturized heat exchanger for cooling electronics components and systems, miniaturized heterogeneously catalyzed gas-phase reactions, thermoelectric generators, etc. New materials, such as polymers, have been explored to develop polymer heat exchangers for better fouling and corrosion resistance .

Parallel-plated heat exchangers have been studied analytically and experimentally to provide formulations for heat exchanger design. Vera and Linan analyzed multilayered, counterflow, parallel-plate heat exchangers numerically and theoretically. They developed a two-dimensional model to find analytical expressions and their approximations for the fully developed laminar counter flow in long parallel-plate heat exchangers. Kragh et al. developed a new counter flow heat exchange for ventilation systems in cold climates. The efficiency of the new heat exchanger was calculated theoretically and measured experimentally.

Zhan et al. used an experimentally validated model to understand the influence of operational and geometric parameters of the cross-flow and counter-flow exchangers on the different metrics of cooling performance. Overall the counter-flow exchanger demonstrated better cooling effectiveness and higher cooling capacity than the cross-flow system. However, the energy efficiency (COP) of the counter-flow system is often seen to be lower than that of the more conventional cross-flow dew point system. The shape of the cross section of the heat exchanger also has a significant effect on efficiency. Hasan et al. studied the effect of channel geometry on the performance of a counter-flow MCHE. The influences of channel shapes such as circular, square, rectangular, isotriangular, and trapezoidal were evaluated by numerical simulations. In their studies, decreasing the volume of each channel or increasing the number of channels increased the heat transfer, but the required pumping power and pressure drop were also increased. The channel with a circular shape resulted in the best overall performance. Recently, CFD analysis of heat exchanger have been used to help design heat exchangers and to analyze their thermal performance, effectiveness and temperature distributions.

Problem Specification

Figure 1 shows a schematic sketch of a multilayered counter flow parallel-plated heat exchanger. Two fluids with different temperatures marked with different colors in Fig. 1 enter numerous channels in separate layers. Each channel is formed by thin folded plates and separation plates between cold and hot fluids. The thickness of the plates is t and the channels have a square shape with a size w and length L. A unit cell consists of a cold channel and a hot channel is taken as the computational domain for CFD analysis, as shown in Fig. 2.

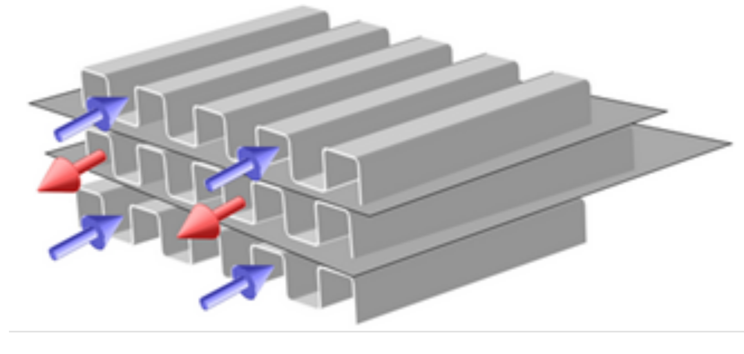

Fig.1 Schematic of a counter flow heat exchanger.

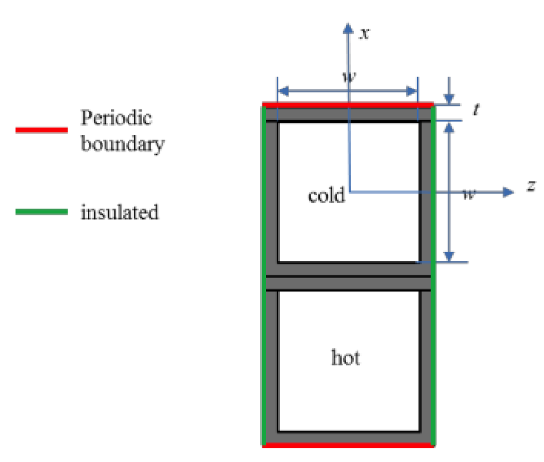

Fig.2 side view the computational domain consisting of one cold and one hot channels.

Table.1. Translation of Design requirements

Parameter	Symbol	Value	unit
Channel length	L	0.2	m
Channel width	W	0.02	m
Channel thickness	t	0.002	m
Density of water	ρ_w	997	kg/m^3
Density of oil	ρ_o	876	kg/m^3
Density of steel	ρ_s	7850	kg/m^3
Thermal conductivity of water	k_w	0.607	W/m·K
Thermal conductivity of oil	k_o	0.145	W/m·K
Thermal conductivity of steel	k_s	44.5	W/m·K
Heat capacity of water	c_{pw}	4180	J/kg·K
Heat capacity of oil	c_{po}	1964	J/kg·K
Heat capacity of steel	c_{ps}	475	J/kg·K
Dynamics viscosity of water	μ	0.891×10^{-3}	Kg/m·s
Dynamics viscosity of oil	μ	0.2177	Kg/m·s
Prandtl number of water	Pr_w	6.14	
Prandtl number of oil	Pr_o	2963	
Inlet velocity of water	V_{inw}	0.005	m/s
Inlet velocity of oil	V_{ino}	0.04	m/s
Reynolds number of water	Re_w	224	
Reynolds number of oil	Re_o	6.44	
Inlet temperature of water	Tin_w	300	K
Inlet temperature of oil	Tin_O	330	K

CFD Simulation

This paper simulated an oil to water heat exchanger. Hot oil at 330K enters hot channel with an inlet velocity of 0.04 m/s. Cold water at 300K enters cold channel with an inlet velocity of 0.005 m/s. The channels have a dimension of 2 cm × 2 cm × 50 cm. Channel wall thickness is 2 mm. Table I listed the important parameters used in this simulation. The properties of oil and water were set as a function of temperature in the simulation. The properties of water at 25^{o}C and those of oil at 40^{o}C were listed in Table I to calculate Reynolds numbers for each channel. The Reynolds number in the channels are found to be 224 and 6..44 for the respective cold and hot channels.

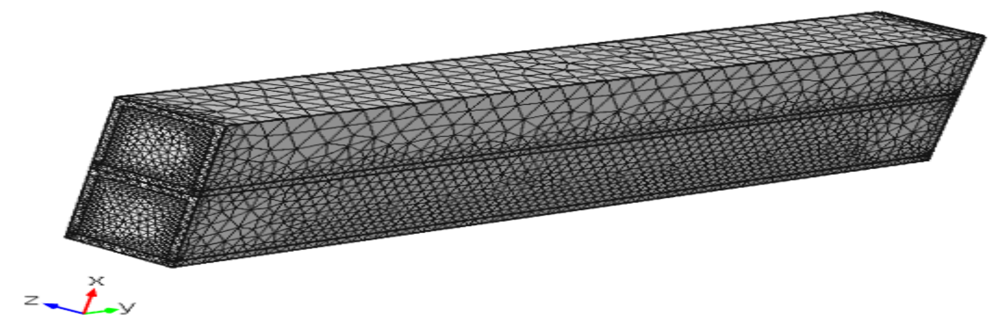

Fig.3 Computational mesh

Simulation Results

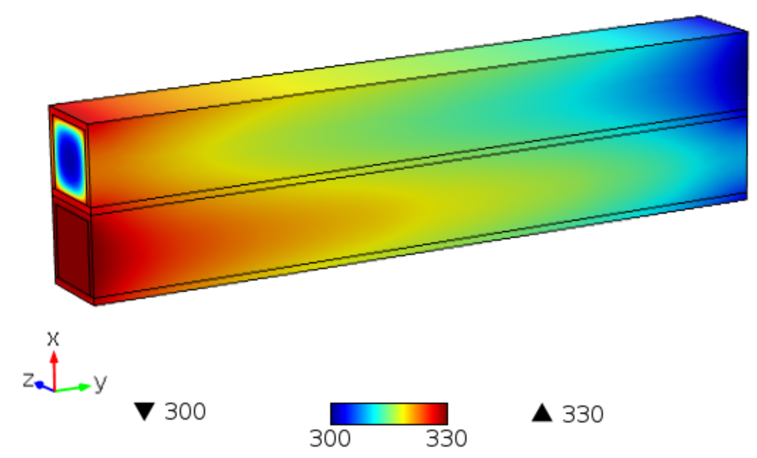

Fig. 4 Surface temperature of the channels and walls

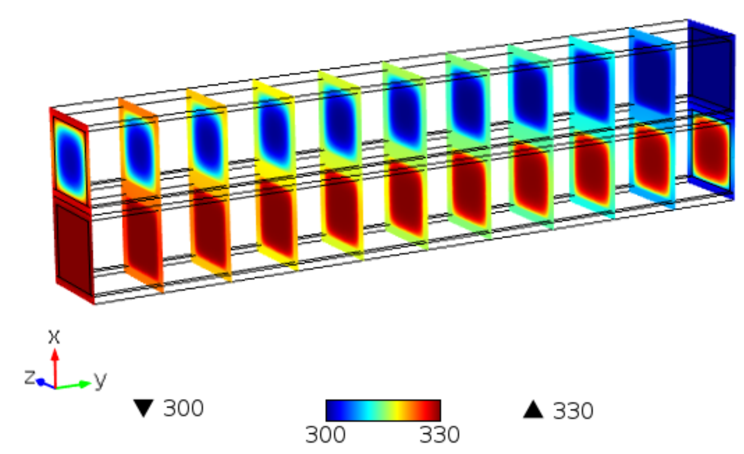

Fig.5 Temperature fields in the x-z cut-planes along channel length.

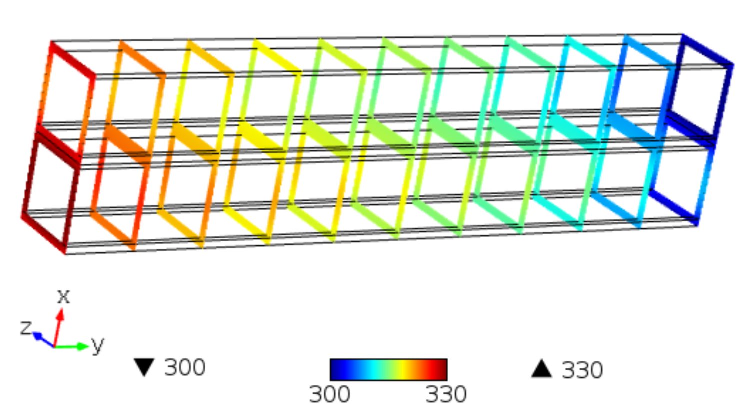

Fig.6 Temperature fields in the solid walls in the x-z cut-planes along channel length

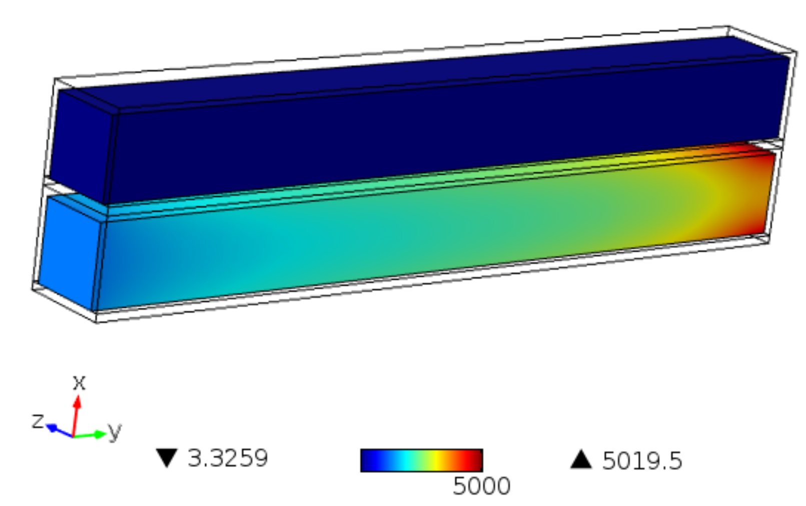

Fig.7 Prandtl number of fluids

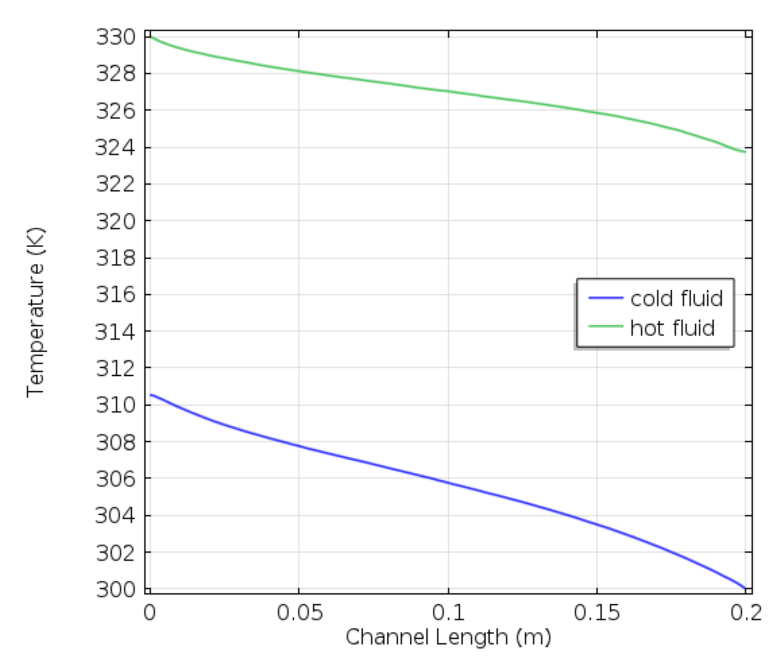

Fig.8 .Average fluid temperatures along the channel length.

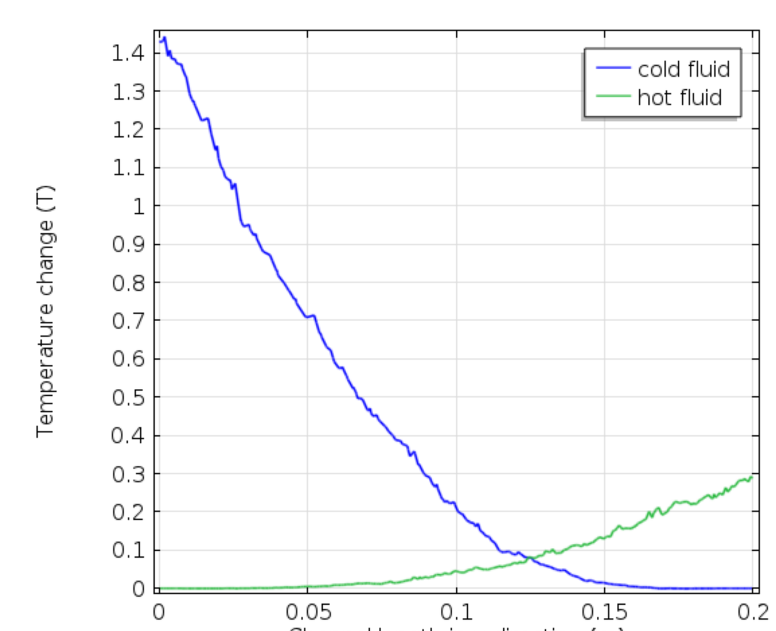

Fig.9. Fluid temperature changes along channel centerlines.

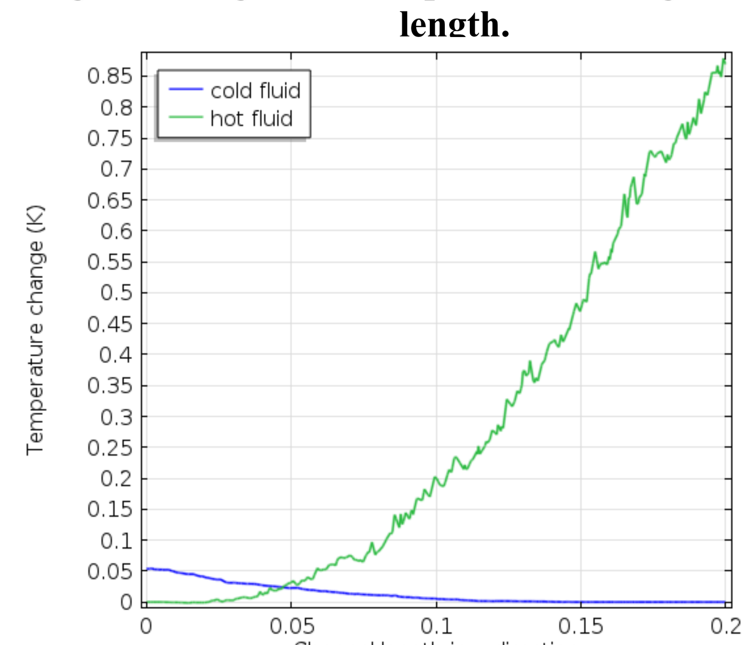

Fig.10. Fluid temperature changes along channel centerlines for the case with modified inlet velocities.

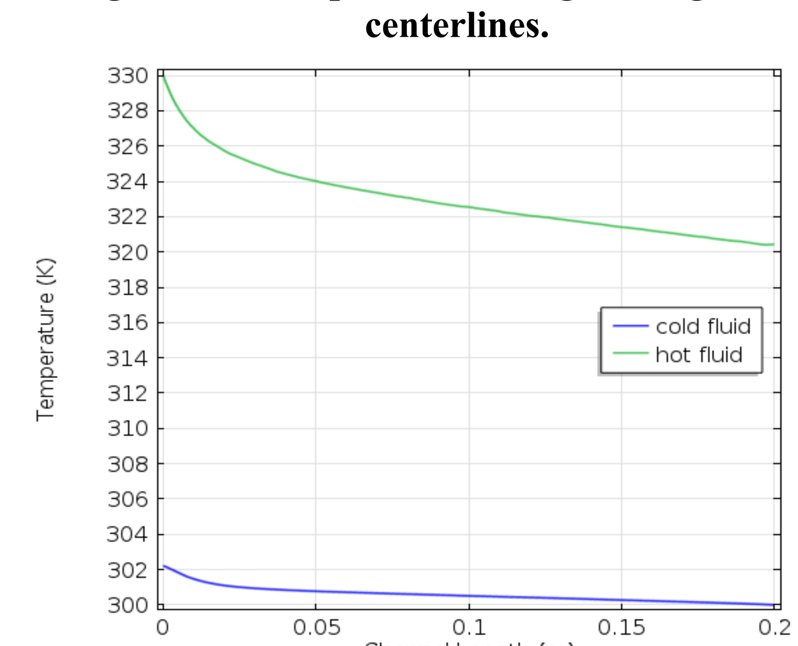

Fig.13 Average fluid temperatures along the channel length for the case with modified inlet velocities.

Conclusion

In this study, a 3D model of a multilayered counterflow parallel heat exchanged was developed to simulate the heat transfer and fluid flow pattern in a unit cell of one cold channel and one hot channel. The model was simulated in COMSOL with oil and water as two working fluids. The detailed temperature, velocities, and pressure distributions in the channels can be used as guidance for an optimal heat exchanger design.