Available online at www.sciencedirect.com

ScienceDirect

Procedia Engineering 174 (2017) 188 – 194

Procedia Engineering

www.elsevier.com/locate/procedia

13th Global Congress on Manufacturing and Management, GCMM 2016

A Study on 3D Numerical Model for Plate Heat Exchanger

Ya-Nan Wang[a], Jong-Pyo Lee[a], Min-Ho Park[a], Byeong-Ju Jin[a], Tae-Jong Yun[a], Young-Ho Song[b], Ill-Soo Kim1[a,*]

[a]Mokpo National University, 1666 Yeongsan-Ro Chonggye-Myun Muan-Gum Jeonnam. Muan 58554, Korea
[b]DHP Engineering Co., Ltd, G, 105, 1022 Beon-gil, Gimhae-daero, Hallim-myeon, Gimhae-si, Gyeongnam, Gimhae 50850, Korea

Abstract

Generally, heat exchanger is a thermodynamic system which has been employed to transfer thermal energy between two or more fluids, between a solid surface and a fluid, or between solid particulates and a fluid, at different temperatures and in thermal contact. Recently, the growing requirements to save energy and reduce overall environmental impacts has placed greater emphasis on the use of heat exchangers with better thermal efficiency. Therefore, the plate-type heat exchanger that uses metal plates to transfer heat between two fluids has a major advantage over a conventional heat exchanger in which the fluids are exposed to a much larger surface area because the fluids spread out over the plates. However, fouling with the flow of fluid for a long period occurs on the plate surface. So, it might be necessary to carry out numerical simulation for the hot and cold fluid flow model which can reflect the real situation of hot and cold fluid heat transfer. In this paper, plate-type heat exchanger is simulated which based on the computational fluid dynamics method. The calculated model is established for fluid flow and heat transfer in cold and hot flow channels of a chevron-type plate-type heat exchanger. The flow pattern and heat transfer effect of the cold and hot fluids are analyzed under the water-to-water heat transfer by using the simulation software, ANSYS. At the same time, 3D numerical model of the local computation domains for the plate-type heat exchanger are simulated at different parameters which included the corrugated angle, corrugated depth and corrugated pitch. With the development of CFD(Computational Fluid Dynamics) technology, numerical guidance can be provided for the improvement of plate-type heat exchanger and optimization of the design.

Peer-review under responsibility of the organizing committee of the 13th Global Congress on Manufacturing and Management

Keywords: Plate-type heat exchanger; Fluid flow, Heat transfer; Design of optimization; 3D Numerical model

* Corresponding Ill-Soo Kim. Tel.:+82-061-454-3455; fax: +82-061-452-6376.
E-mail address: ilsookim@mokpo.ac.kr

Peer-review under responsibility of the organizing committee of the 13th Global Congress on Manufacturing and Management
doi:10.1016/j.proeng.2017.01.203

1. Introduction

Heat exchangers are commonly employed in a wide range of applications from heating and air-conditioning systems in a household to chemical processing and power production in large plants. In recent years, the energy crisis and the thrust for energy conservation have driven the development for high efficiency heat exchangers. Therefore, the related researches are being conducted in accordance with the requirements for economical, lightweight, and spaces saving heat exchangers.

Nomenclature

u	Velocity components in the x, y, z directions	C_1,C_2,C_3	Constant
Y_M	Fluctuations due to excessive diffusion in the compressive turbulent flow	B	Corrugated angle
G_k	Turbulent kinetic energy produce by the velocity gradient of the laminar flow	h	Corrugated depth
G_b	Turbulent kinetic energy generated by buoyancy	λ	Corrugated pitch
x, y, z	Three directions of Cartesian coordinates	Δp	Pressure drop
ε	Turbulent Prandtl number	S_k, S_ε	Corrugated pitch
α	Thermal conductivity	ρ	Density
ν	Viscosity	p	Pressure
t	Temperature		

Plate-type heat exchanger, one of the best devices for the advancement and technological development in compact heat exchanger consists of a stack of stamped heat exchange plates which are either brazed together or bolted together in a frame with gaskets due to its high thermal efficiency [1-2]. The plates of these heat exchangers comprise of some form of near-sinusoidal corrugations in a chevron pattern, a design commonly used for PHE(Plate Heat Exchanger) as it is considered the most successful type [3]. It is known that the number of plates is determined by the flow rate, physical properties of the fluids, pressure drop and temperature program [4]. The plate corrugations promote fluid turbulence and support the plates against differential pressure. The plate and the pressure plate are suspended from an upper carrying bar and located by a lower guiding bar, both of which are fixed to a support column. Connections are located in the frame plate in the frame and pressure plates. Nevertheless, the Reynolds numbers used in this type of equipment due to the relatively high pressure drop must be lower than those for shell and tube heat exchangers for equivalent flow rates. Furthermore, the applicable Reynolds numbers must be less than 2,000 as this equipment is used as a reflux condenser. On the other hand, the high fluid velocity inside the passage has the opposite result on fouling as it promotes particle detachment. Thus, the lack of resources in literature has taken place in direction for the development of an artificial design model for a PHE, even though there are studies referring to the effect of several geometrical parameters on the heat transfer coefficient and the friction factor [5, 6]. Mehrabian and Poulter [7] studied the flow inside a PHE with corrugations, while Hossain and Sadrul Islam [8] researched similar control volumes for three different corrugation shapes. In recent years, the developed computational fluid dynamics have been concentrated on the development of an effective tool for evaluating the performance of heat exchangers. In the literature, most works have conducted numerical analysis of the plate-type heat exchanger. However, this assumption is not realistic in general applications. Furthermore, the works of numerical simulation documented in the literature are limited because the plate-type heat exchanger is very difficult to be reproduced accurately. In this paper, plate-type heat exchanger is simulated based on the computational fluid dynamics method, and calculation model is established for fluid flow and heat transfer in cold and hot flow channels of a chevron-type plate heat exchanger by using the simulation software, ANSYS. At the same time, with the development of CFD technology for the plate-type heat exchanger are simulation at three different process parameters (corrugated angle, corrugated depth and corrugated pitch), to provide numerical guidance for the improvement of plate-type heat exchanger and the optimization design by discussion for the pressure distribution and pressure drop.