The 6th International Chemical Engineering Congress & Exhibition
16 - 20 November, 2009 Kish Island, I.R. Iran (ICHEC 2009)

Experimental and Numerical analysis of flow and heat transfer in a gas- liquid thermosyphon heat exchanger in a pilot plant

Zare Aliabadi[a,b], Hassan , Atashi, Hossein[b]
a: Department of chemical engineering, Islamic Azad University, Shahrood Branch, Shahrood, I.RIRAN
Corresponding author: Tel: +98915 5056588, Fax: +98511 8812295 and E-mail: h_a_zare@yahoo.com
b: Department of chemical engineering, University of Sistan and Baluchestan, Zahedan, I.R.IRAN
Noie, Seyed Hossein
Department of chemical engineering, Ferdowsi University of Mashhad, Mashhad, I.R.IRAN
Khoram, Mohammad
Department of chemical engineering, University of Sistan and Baluchestan, Zahedan, I.R.IRAN
Khoshvaght, Mohammad
Department of chemical engineering, Islamic Azad University, Shahrood Branch, Shahrood, I.R.IRAN

Abstract: *a numerical and experimental investigation of flow and heat transfer in a gas- liquid thermosyphon heat exchanger "THE" with built in heat pipes and aluminum plate fins for moderate Reynolds numbers has been carried out.. It's module is composed of 6"rows" and 15"columns" copper pipes with aluminum plate fins with dimensions of 130cm "height", 47cm "width" and 20cm "depth" . The tubes have been filling by water with filling ratio of 30%, 50% and 70%. The density and thickness of fins are 300 fin/m and 0.4mm, respectively. The configuration of tubes is in-line with 30mm pitch. This paper presents the distribution of temperature and thermal performance of "THE" by using CFD modeling. A good comparison of the present CFD modeling results for the modeling thermosyphon heat exchanger with experimental results of hydrodynamic and thermal behavior achieved.*

Key words: *Thermosyphon heat exchanger "THE", CFD modeling, In-line configuration, Pressure Drop, thermal performance*

Introduction

Heat recovery**,** one of the methods of energy conservation can be successfully implement when the investment cost of additional equipment required is acceptably low. Thermosyphon based heat exchangers are very simple devices that can be used to heat transfer between two fluid phases [1]. In this section, a brief review of some of experimental and theoretical research conducted present. Rosman et al. [2] investigated numerically and experimentally one and two –row fin tube heat exchanger. They used isothermal fins for their calculations, the reason being the narrow space between the two adjacent fins. Biswas et al. [3] and Tsai et al. [4] reported numerical investigations on related topics. In the above investigations, the enhancement of heat transfer from the fin surfaces was achieved by disrupting the growth of thermal boundary layer. Song Lin and et al. [5 have presented a design method by using CFD simulation of the dehumidification process with heat pipe heat exchanger. Their studies illustrate that the CFD modeling is able to predict the thermal performance of the dehumidification solution with heat pipe heat exchanger (HPHE). Tan and Liu [6] have used the ε-NTU method to analyze an air-to-air heat pipe heat exchanger. They also have presented an equation to determine the optimum position separating a heat pipe into evaporator and condenser regions in a heat pipe heat exchanger was formulated by minimizing the total thermal resistance of the heat path. Noie [7] has carried out an experimental study of the performance of an air-to-air thermosyphon-based heat exchanger utilizing water as working fluid to investigate its behavior under different operating conditions. Zare Aliabadi, H. and et. al. [8] carried out an experimental research to investigate the hydrodynamic and thermal performance of a gas-liquid "THE". The effects of various parameters such as the heat capacity ratio of high- and low-temperature fluid streams "Ce/C_c", the inlet hot air temperature, and the mass flow rate or the inlet hot air velocity on thermal performance of a gas-liquid "THE". In this research, we have investigated fluid dynamic and thermal performance of "THE" by using CFD modeling and experimental pilot plant results.

Statement of the Problem

Theory