Available online at www.sciencedirect.com

ScienceDirect

Procedia Engineering 157 (2016) 428 – 435

Procedia Engineering

www.elsevier.com/locate/procedia

IX International Conference on Computational Heat and Mass Transfer, ICCHMT2016

Experimental and numerical investigation of flow distribution within the heat exchanger with elliptical tubes

Paweł Ocłoń*, Stanisław Łopata, Kamil Chłosta

Institute of Thermal Power Engineering, Faculty of Mechanical Engineering, Cracow University of Technology, Al. Jana Pawła II 37, 31-864 Krakow, Poland

Abstract

The goal of this article is to validate the CFD simulations of the working fluid flow distribution in individual tubes of fin-and-tube heat exchanger. For this purpose, the CFD simulations have been conducted for the design of the inlet and outlet manifolds of the heat exchanger tube elliptical, and the results compared with the values determined empirically, by measuring the mass flow flowmeter installed in the lower part of the heat exchanger tube. A comparison of the results of numerical calculations with the measurement results is presented. The turbulence model k-ε, k-ω and Shear Stress Transport model SST was considered in the computations. It has been verified that the numerical simulation results (mass flow rates) allow to produce the results in good agreement with experimental data.

Peer-review under responsibility of the organizing committee of ICCHMT2016

Keywords: heat exchanger, oval-tube, elliptical tube, CFD, turbulence models, k-ε, k-ω, SST

1. Introduction

In recent years, the fin-and-tube heat exchanger attracted broad scientific attention due to the high heat transfer efficiency achieved within a compact size [1-10]. Fin-and-tube heat exchangers are widely used in a variety of applications in the waste heat recovery units (WHRU), HVAC&R systems, automotive applications, chemical processing plants and other units and systems. The main advantage of fin-and-tube heat exchangers is a vast heat transfer area, which is utilized through extended surfaces (fins) [11-13]. Fins are usually used to achieve efficient

* Corresponding author. Tel.: +48-628-35-59; fax: +48-628-35-60.
E-mail address: poclon@mech.pk.edu.pl

Peer-review under responsibility of the organizing committee of ICCHMT2016
doi:10.1016/j.proeng.2016.08.386

heat transfer between the gasses (flue-gas, air) and liquids (water, refrigerants, and oils). Hence, the fin-and-tube heat exchangers perform well mainly in a situation when a significant difference in the heat transfer coefficients (HTC) exists (i.e., low HTC for gas and high for a liquid). In industrial applications, the fin-and-tube heat exchangers are used as coolers, condensers, evaporators, and heaters. Elliptical tubes are preferred to circular ones, due to larger heat transfer efficiency and lower pressure drop [1,4].

When designing a heat exchanger one of the major challenges for designers is to correctly realize the flow and thermal processes, which ensure safe operation of these devices [14-21]. Today, an integral part of the design of heat exchangers is a flow modeling in programs that use numerical values to obtain the distribution of velocity and temperature in the issues under consideration. For the analysis of the tested fin-and-tube heat exchanger with elliptical tubes, a numerical model of flow domain was developed. The simulations were conducted in ANSYS CFX code [22]. The mass flow rate in individual heat exchanger tubes was determined by using the flowmeters of oscillatory type (Sontex Superstatic 749). Attempts have been made to validate the respective turbulence models with the experimental data. The turbulence model, which allows one to obtain the results more similar to those derived from experimental studies, was indicated. Furthermore, it was found that the CFD simulations allow to correctly determine the flow rate of the medium in each tube of heat exchanger.

Nomenclature

n_p	rotational speed of pump impeller, rot/min
$\dot{m}$	mass flow rate per heat exchanger tube, kg/s

2. Experimental setup and measurement results

This paper analyzes the flow in heat exchanger tubular space, shown in Figure 1. The heat exchanger consists of twenty elliptical tubes, which are arranged in two rows in staggered formation. Tubes are fixed to the inlet and outlet chambers, whose construction is shown in Figure 1 and Figure 2. For the purpose of CFD analysis and results interpretation the tubes of the heat exchanger are numbered in the manner shown in Figure 3.

The measurements were carried out successively for different rotational speeds of the pump, ie. 2500,3000,4000,4850 [rot / min]. A steady-state operation of heat exchanger was considered. Each of the flow meters read the volume of the fluid that has passed through a given a tube of heat exchanger. Action was repeated a total of three times in one measurement. For the first time, read the value of the volume, which has passed from the time of the pump to start the measurement, then at intervals of 30 minutes the fluid volume was read two times. The same procedure was repeated for various rotational speed of the pump impeller.