Applied Thermal Engineering 31 (2011) 1588–1599

Contents lists available at ScienceDirect

Applied Thermal Engineering

journal homepage: www.elsevier.com/locate/apthermeng

Numerical study of fluid flow and heat transfer in a multi-port serpentine meso-channel heat exchanger

Masoud Dehghandokht [a], Mesbah G. Khan [b], Amir Fartaj [b,*], Sepehr Sanaye [a]

[a] Energy Systems Improvement Laboratory (ESIL), Dept. of Mechanical Engineering, Iran University of Science and Technology (IUST), Iran
[b] Dept. of Mechanical, Automotive, and Materials Engineering (MAME), University of Windsor, 401 Sunset Av., Windsor, ON, Canada N9B 3P4

ARTICLE INFO

Article history:
Received 16 July 2010
Accepted 24 January 2011
Available online 5 March 2011

Keywords:
Meso-channel
Heat exchanger
Numerical Simulation
Serpentine
Crossflow

ABSTRACT

Three dimensional numerical simulations, based on the Navier–Stokes equations and the energy equation, are performed for forced convection of hot water through a serpentine meso-channel heat exchanger with a cold air across flow. The results are compared for the same geometrical and experimental conditions. The numerical predictions of heat transfer rate, both pressure and the temperature drop in the water, and the core surface temperature at the serpentine bend are found to be in good agreement with experimental data. Numerical results show that the existence of serpentine bend in the meso-channel heat exchanger causes the heat transfer rate to increase. The multi-port slab has flat surfaces on the top and bottom faces; air flows in contact with the heat transfer surfaces and maintains a uniform temperature distribution. Moreover, the parallel channels located inside the meso-channel slab core help to distribute the heat fairly uniformly through all the channels.

1. Introduction

Microchannel and meso-channel heat exchangers have found applications in highly specialized areas such as microelectronics cooling, aerospace, biomedical processes, metrology, robotics, automotive industries etc. [1]. Shah [2], in studying the compact heat exchanger technologies and their applications, mentioned that “a heat exchanger referred to as a micro heat exchanger if the surface area density is above 10000 m^2/m^3”. Several researchers have compared micro- and meso-channel heat exchanges with conventional ones based on various fluid flow and heat transfer parameters.

Many researchers have studied the thermal and hydraulic performance of microchannel heat exchangers in recent times. Ramshaw [3] investigated the process intensification in chemical industries and found that the microchannel heat exchangers can greatly increase the heat exchange that can be accomplished per unit volume as an aim of process intensification.

Bier et al. [4] and Bier et al. [5] studied the microchannel heat exchanger with an active volume of 1 cm^3 and surface density of 14200 m^2/m^3 and 4000 channels per cubic centimeter. Their results demonstrated that a volumetric thermal power of 18000 MW/m^3 could be achieved with the magnitude of overall heat transfer coefficient of 20 kW/m^2 K and a temperature difference of 50 °C.

Peng et al. [6], Wang and Peng [7], Adams et al. [8] and Qu et al. [9] found that the flow through Microchannels can deliver up to 60 times higher heat transfer rates than the conventional channels and can easily achieve a heat flux level in excess of 100 W/cm^2. These microchannel heat exchangers however incur high fluid pressure drop, which is the major drawback. Conventional heat exchangers are found to have an average heat transfer surface area per unit volume of 50–100 m^2/m^3, and a heat transfer coefficient of up to 5000 W/m^2.K when using liquid as the working fluid.

Roetzel [10] calculated heat transfer coefficient of crossflow arrangement of heat exchangers in special cases when either only laminar length effects or only temperature dependences are present. A more general approximation method was described by Roetzel [10] for the calculation of the mean overall heat transfer coefficient and the overall pressure drop, which is valid for any flow arrangement and combined length and temperature effects. For the analyses of variable heat transfer coefficient and heat distribution three dimensional numerical simulation may be useful in general but not for design purpose because three dimensional CFD analysis is too time consuming and needs large computational capacity

Traditional design techniques for conventional heat exchangers such as ϵ–NTU relations and procedure proposed by Roetzel [10] cannot be directly applied to the micro- and hence small channel heat exchangers since for their design it was assumed a constant value of convective heat transfer coefficient. The value of heat transfer coefficient can be obtained using various correlations for individual ducts. The thermally fully developed flow may not exist

---

[*] Corresponding author.
E-mail address: fartaj@uwindsor.ca (A. Fartaj).

1359-4311/$ – see front matter © 2011 Elsevier Ltd. All rights reserved.
doi:10.1016/j.applthermaleng.2011.01.035