International Communications in Heat and Mass Transfer 38 (2011) 1349–1353

Contents lists available at SciVerse ScienceDirect

International Communications in Heat and Mass Transfer

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

A numerical and experimental investigation of flow maldistribution in a micro-channel heat sink ☆

G. Kumaraguruparan [a], R. Manikanda Kumaran [b], T. Sornakumar [a], T. Sundararajan [b,*]

[a] Dept. of Mechanical Engg, Thiagarajar College of Engineering, Madurai, 625015, India
[b] Dept. of Mechanical Engg, Indian Institute of Technology Madras, Chennai, 600036, India

ARTICLE INFO

Available online 9 September 2011

Keywords:
Micro-channels
Mal-distribution
Flow separation
Inlet header

ABSTRACT

A study of flow mal-distribution in U-type micro-channel configuration is presented. Numerical simulations indicate that flow deceleration and associated pressure recovery in the inlet header lead to flow separation and recirculation which cause oscillations in channel-wise mass flow distribution. Increase in flow resistance by decrease in channel depth, width or number of channels or increase in channel length, results in a more uniform distribution. Mal-distribution increases at high flow rate or low viscosity due to the dominance of inertial phenomena. Experiments performed on a 25-channel setup illustrate that small manufacturing variations in channel dimensions introduce random fluctuations in flow distribution.

1. Introduction

Micro-channels are employed in the cooling of microelectronics, offshore gas/fuel processing, gas turbine blade cooling, PEM-fuel cells and in high temperature nuclear reactors [1,2]. Channels whose widths lie in the range of a few microns to a few hundred microns [3] are generally termed as micro-channels. The high volumetric heat dissipation rates encountered in many modern electronic devices necessitate the use of large heat transfer areas, which could be achieved by resorting to coolant flow through micro-channels [4,5]. Pressure drop and heat transfer studies of micro-channel flow have been performed in the recent past, using single channels or multiple parallel channels. The single phase experiments on single micro-channels [6–8] reveal that the experimental and theoretical friction factor data are in good agreement and there is no early transition to turbulent regime. But, experiments on multiple micro-channels exhibit inconsistent results with regard to friction factor variation and flow transition [9,10], which are attributed to mal-distribution effects not being properly accounted for. Flow mal-distribution can occur due to size variations between the channels because of manufacturing tolerances or due to changes in the viscosity of the fluid or because of poor manifold design [11]. The effect of viscosity variation on mal-distribution is incorporated by introducing a viscosity ratio in the correlations for friction factor and Nusselt number [12]. The types of flow manifolds and the associated mal-distribution patterns have been discussed by Bajura and Jones [13]. Pan et al. [14] have shown that in order to achieve a uniform flow distribution, the micro-channel depth, width and pitch should be made as small as possible; also, the channel length should be large. It has been observed that the optimal channel configuration depends on the mass flow rate and Reynolds number of flow [15,16].

Although a few studies are available in literature which explain the effects of channel geometry on flow mal-distribution in micro-channels, the detailed mechanisms which cause mal-distribution have not been identified. Further, the influence of factors such as the number of channels, fluid viscosity and manufacturing tolerances has not been highlighted. In the present study, an experimental and numerical study is performed to systematically investigate the causes for flow mal-distribution and to highlight the effects of parameters such as the number of channels, type of fluid used and manufacturing tolerances.

2. Mathematical model and numerical solution procedure

Subjected to continuum, steady, incompressible and laminar (for the small flow rates considered here) flow considerations, the equations that govern the three dimensional flow in the micro-channels can be written in flux form suitable for finite volume discretization over a cell, using Cartesian tensor notation, as follows:

Continuity equation:

$$\oint \left(V_j \cdot n_j\right) \cdot dA = 0 \tag{1}$$

☆ Communicated by A.R. Balakrishnan.

* Corresponding author at: Thermodynamics and Combustion Engineering Lab, Department of Mechanical Engineering, Indian Institute of Technology, Madras, Chennai, 600036, India.
E-mail address: tsundar@iitm.ac.in (T. Sundararajan).

0735-1933/$ – see front matter © 2011 Elsevier Ltd. All rights reserved.
doi:10.1016/j.icheatmasstransfer.2011.08.020

Nomenclature

A	Area
d	Depth
g	Acceleration due to gravity
L	Length
m	Total flow rate
n	Unit normal
N	Number of channels
Δp	Pressure drop
t	Time
V	Volume
W	Width
x_i, x_j	Cartesian coordinate direction

Greek symbols

p	Density
σ, τ	Normal and Shear stress

Subscripts

C	Channel
Cell	Computational cell

Momentum equation:

$$\rho \cdot V_{cell} \cdot \frac{\partial V_i}{\partial t} + \rho \cdot \oint \left(V_j \cdot n_j \right) \cdot V_i \cdot dA = \rho \cdot g_i \cdot V_{cell} + \oint \left(\sigma_{ij} n_j \right) \cdot dA \qquad (2)$$

Where the stresses are given by

$$\sigma_{ij} = -p\delta_{ij} + \mu \left(\frac{\partial V_i}{\partial x_j} + \frac{\partial V_j}{\partial x_i} \right)$$

with δ_{ij} denoting the Kronekar delta and i, j denoting Cartesian directional indices. Due to the low Knudsen number range ($Kn<0.001$) of the present study, no additional terms have been included to represent micro-phenomena in the governing equations and boundary conditions.

The geometrical dimensions of the channels along with the type of the fluids and their mass flow rates employed for the simulation are summarized in Table 1. A three-dimensional model of the micro-channel geometry is created using the software Gambit 2.1 and the governing equations of flow are solved using the finite volume based commercial CFD solver, Fluent 6.2. A segregated implicit solver with SIMPLE pressure-correction algorithm has been adopted to compute the flow field. The boundary conditions applied for the analysis are: prescribed mass flow rate at the header inlet; no-slip conditions at the walls of the header and the channels; at the header exit, the pressure is specified as atmospheric. For the default geometry, the effect of mass flow rate on the channel-wise flow distribution has been studied. Although majority of the cases studied here correspond to the laminar regime, in a few cases, the Reynolds number is of the order of 4000. In such cases, the one equation Spalart–Allmaras (SA) turbulence model has been adopted to compute the flow pattern. The SA model has been employed due its relative simplicity and effectiveness in capturing the turbulent features of boundary layer dominated internal flows.

Grid sensitivity for the numerical predictions has been investigated using various structured non-uniform meshes, with number of cells ranging from 615,646 to 1,366,790. Finer grids have been used within the channels in order to capture the maldistribution effects and the flow features accurately. The predicted channel-wise normalized flow rates for different meshes indicate that the numerical results are insensitive to grid refinement (with a maximum deviation of 0.5%) beyond a mesh of about 1,065,558 cells. Hence, for all the subsequent studies, a grid with 1,065,558 cells has been adopted. Simulations have been carried out until the residues fall below 1×10^{-6} for all flow variables and below 1×10^{-4} for continuity equation, during the iterative solution procedure.

3. Experimental setup and measurement procedure

The 25-channel heat sink set up employed in the experimental study is shown in Fig. 1. For each channel of this set up, the width and depth have been measured using a profilometer at five different axial locations. At each location, the dimensions are measured three times and averaged. It is to be noted, however, that there are size variations (up to about 5%) along the length of each channel and also between channels, due to manufacturing tolerances. A transparent acrylic sheet along with a slotted mild steel cover is fixed over the micro-channels to visualize the flow inside the channels and the header. Water is used as the working fluid in the experiments. In order to quantify the mal-distribution encountered in the micro-channel setup, water passing through each channel is collected individually without employing an outlet header.

Prior to the start of experiments, high pressure air is passed through the micro-channel heat sink, which is immersed in water. After ensuring that there is no air leak, $KMnO_4$-water solution is sent through the setup to guarantee that the fluid flows only through the micro-channels, without spilling over the channels. De-ionized, de-gassed water from the reservoir is pumped with the help of 0.5 HP self priming mono-set pump, through a 5 μm water filter. The water flow rate is controlled with the help of valves provided near the outlets. When flow is stabilized, the water from each channel is collected with the measuring jar for 3 to 5 min, and the mass of the water is measured by an electronic balance with a maximum error of about 0.2%. For a fixed flow rate, the experiment has been repeated a few times on different days to ensure the repeatability of the data

Table 1
Predicted pressure drop (Δp) and Mal-distribution factor (MF) for various cases studied.

Sl. No.	Values	Pressure drop (Δp) between inlet and outlet/Pa	Maldistribution factor (MF) in%
Default case	$w_c=0.5$ mm, $d_c=5$ mm, N = 25, $L_c=150$ mm, Fluid = Water m = 0.007 kg/s	942	3.06
W_c (mm)	0.05	808,782	0.11
	0.075	240,161	0.10
	0.1	101,576	0.15
	0.3	3938	0.80
d_c (mm)	0.1	1,172,970	0.02
	0.5	21,104	0.45
	1	6410	0.85
	2.5	1973	1.69
L_c (mm)	50	377	9.96
	100	661	5.46
W_f (mm)	0.5	945	2.79
	1	945	3.52
N	5	4706	0.98
	10	2305	1.04
	15	1543	1.23
	20	1165	2.45
m (kg/s)	0.0207	3346	8.20
	0.0562	13,301	25.54
	0.0780	21,904	24.88
Fluids	Ethylene glycol	11,972	0.08
	Air	95,352	69.82
	m = 0.0597	45099.61	32.12
	m = 0.0798	74467.89	35.8435
	m = 0.0956	102643.36	37.9433