УДК 536.7

CFD-SIMULATION OF THERMO-HYDRODYNAMICS PROCESSES IN RECTANGULAR MICROCHANNEL ARRAYS

Lobasov A.S.
Scientific supervisor – Associate professor Petrischeva G. P.

Siberian Federal University

In recent years, there is a rapid growth of applications which require high heat transfer rates and fluid flows in relatively small passages. Some examples which demand such flow conditions are electronics cooling, space thermal management, MEMS devices for biological and chemical analyses etc. The development of new applications requiring cooling of components in a confined space has motivated researchers to focus on the prediction of the thermohydrodynamic performance of mini and microchanels.

Nominally, microchannels can be defined as channels whose dimensions are less than 1 mm and greater than 1 μm. Currently most researched microchannels fall into the range of 30 to 300 μm. Despite the fundamental simplicity of laminar flow in straight ducts, experimental studies of microscale flow have often failed to reveal the expected relationship between the transport parameters. Furthermore, data of simultaneously developing flows, which inherently provide high species transport coefficients, are not very abundant.

Thus, in this paper modelling of liquid flow in the rectangular parallel microchannel block with the various cross-section area has been carried out. In table 1 the geometrical sizes of researched microchannels are resulted. The microblock geometry for configuration S5 is shown in figure 1. In figure 2 the velocity module distribution is shown. In figure 3 comparison of numerical modelling results for various microchannel configurations with the experimental and analytical data is shown.

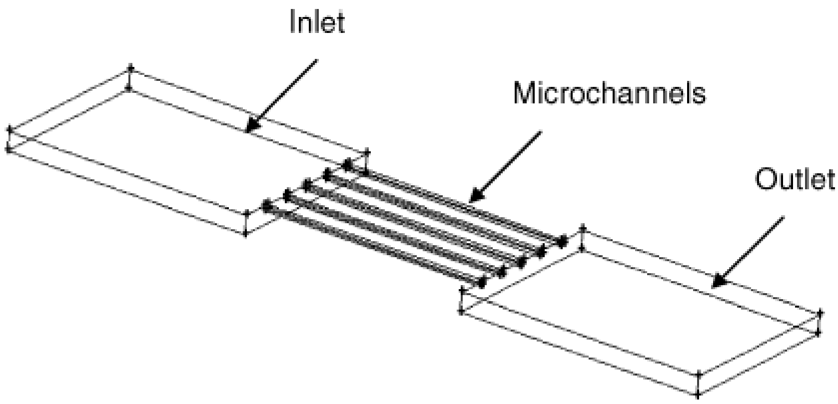

Figure 1 – Microblock geometry

Table 1 – Researched microchannels geometry

Test number	Number of channels	W, μm	H, μm	L, mm	D_h, μm	α, H/W
S5	5	480	460	25,4	470	0,96
L1	5	222	597	41,0	324	2,69

The numerical results in the present work were analyzed in the framework of conventional theory. The pressure drop and flow rate were measured to obtain the two most often used nondimensional parameters, the Reynolds number and the Darcy friction factor:

$$\mathrm{Re} = \frac{\rho \cdot U \cdot D_h}{\mu}$$

$$f = \frac{2 \cdot \Delta P \cdot D_h}{L \cdot \rho \cdot U^2},$$

where ρ – fluid density, U – average velocity in microchannel, D_h – hydraulic diameter, μ – fluid viscosity, ΔP – pressure difference.

Water with constant properties (density – 1000 kg/m^3, dynamic viscosity – 0.001 Pa·s) was used as a working liquid. Series of calculations in which Reynolds number varied has been lead for each microchannel configuration. The researched value was the Darcy friction factor.

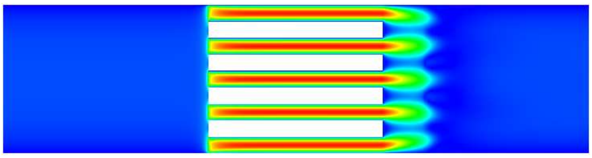

Figure 2 – The velocity module distribution

For fully developed laminar flow in rectangular channels of channel aspect ratio α, the following expression was used to predict the friction constant:

$$f = \frac{96}{\mathrm{Re}} \cdot \left(1 - \frac{1.3553}{\alpha} + \frac{1.9467}{\alpha^2} - \frac{1.7012}{\alpha^3} + \frac{0.9564}{\alpha^4} - \frac{0.2537}{\alpha^5}\right) \qquad (1)$$

For fully developed turbulent flow, predictions were obtained following the Blasius solution:

$$f = \frac{0.316}{\mathrm{Re}^{0.25}} \qquad (\mathrm{Re} < 20000)$$

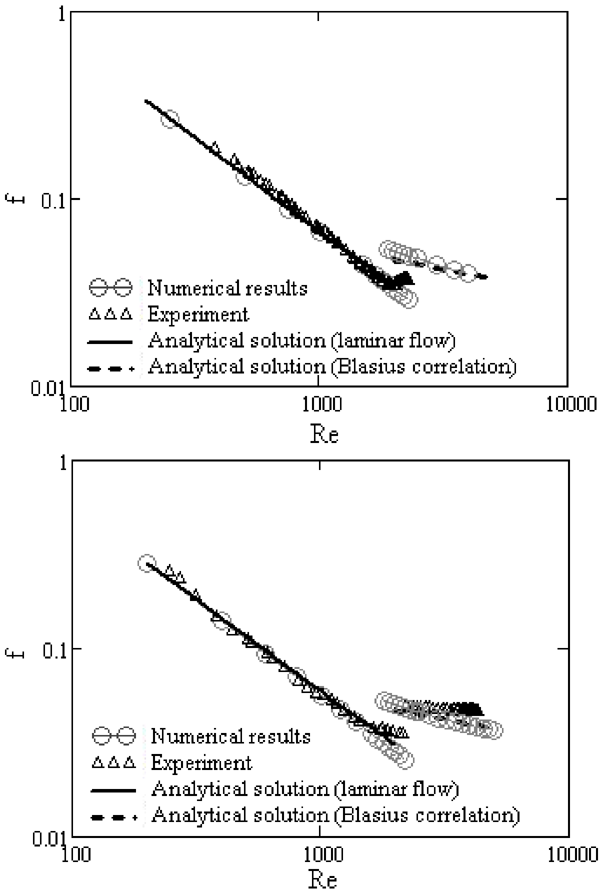

Figure 3 – Comparison of numerical modelling results with the experimental and analytical data

In most practical applications, the microchannel is not long enough for the flow to become fully developed under laminar flow conditions. In such cases, the following expression for apparent friction factor accounts for both the developing and fully developed laminar flow regions in the channel:

$$f_{app} = \frac{\left[\left(\dfrac{3.2}{\left(x^{+}\right)^{0.57}}\right)^{2} + \left(f\,\mathrm{Re}\right)_{fd}^{2}\right]^{\frac{1}{2}}}{\mathrm{Re}}$$

where $(f\cdot \mathrm{Re})_{fd}$ is calculated as in Equation (1) and the entrance length x^{+} is defined as

$$x^{+} = \frac{L}{D_h \cdot \mathrm{Re}}$$

The analysis of the received results has shown very good coordination between numerical modelling and the experimental and analytical data.

Besides in the given work modelling of microchannel block (geometry and microphoto are shown in figure 4) has been carried out.

The sizes of the microchannel: height – 0.772 mm, width – 1.1 mm, length – 50 mm, amount of microchannels in the block – 15. Water with constant properties (density – 1000 kg/m^3, dynamic viscosity – 0.001 Pa·s) was also used as a working liquid. Series of calculations in which Reynolds number varied has been lead for each microchannel configuration. The researched value was the Poiseuille Number:

$$Po = f \cdot \mathrm{Re}$$

The velocity module distribution is shown in figure 5. In figure 6 comparison of numerical modelling results for various microchannel configurations with the experimental and analytical data is shown.

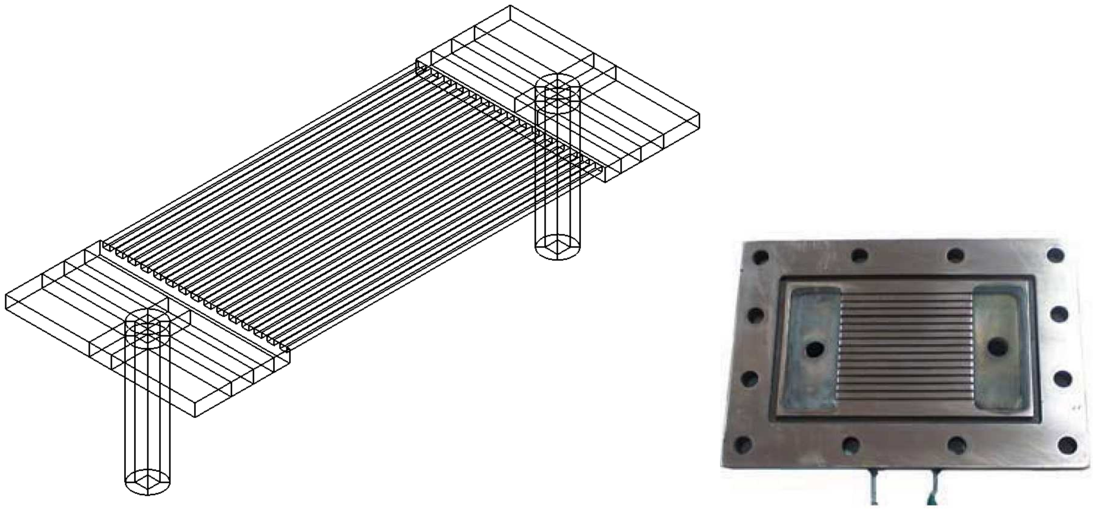

Figure 4 – Microblock geometry

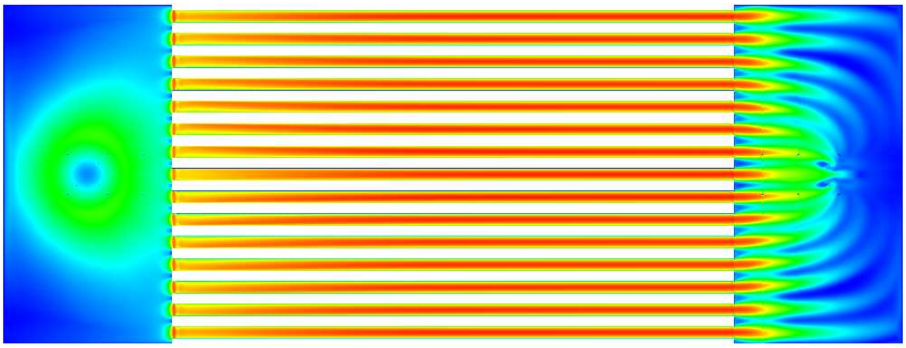

Figure 5 – The velocity module distribution

The local heat transfer coefficient for single-phase forced convection flow in mini-channels can be calculated using:

$$\alpha = \frac{q}{T_w - T_f}, \quad (2)$$

where T_w – average wall temperature, T_f – the average of the inlet and outlet fluid temperature.

The corresponding Nusselt number is given by:

$$Nu = \frac{\alpha \cdot D_h}{\lambda}$$

where λ – fluid thermal conductivity, equaled 1.2923 in this series of calculation (Pr = 3.25).

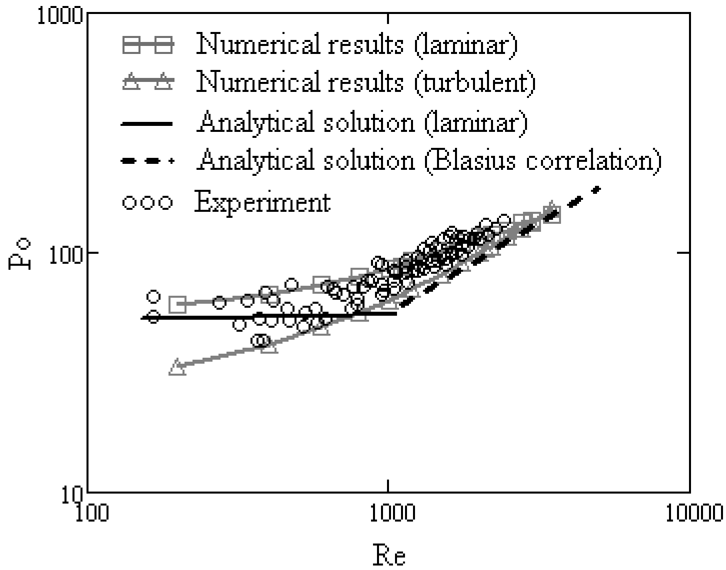

Figure 6 – Comparison of numerical modelling results with the experimental and analytical data

Also in the given microchannel block research of Nusselt number from Reynolds number dependence has been lead. In each microchannel on bottom and side walls the constant density of a thermal stream equal of 10 kW/m^2 was set. The top wall is adiabatic. In figure 7 comparison of results of numerical modelling and the experimental data is shown.

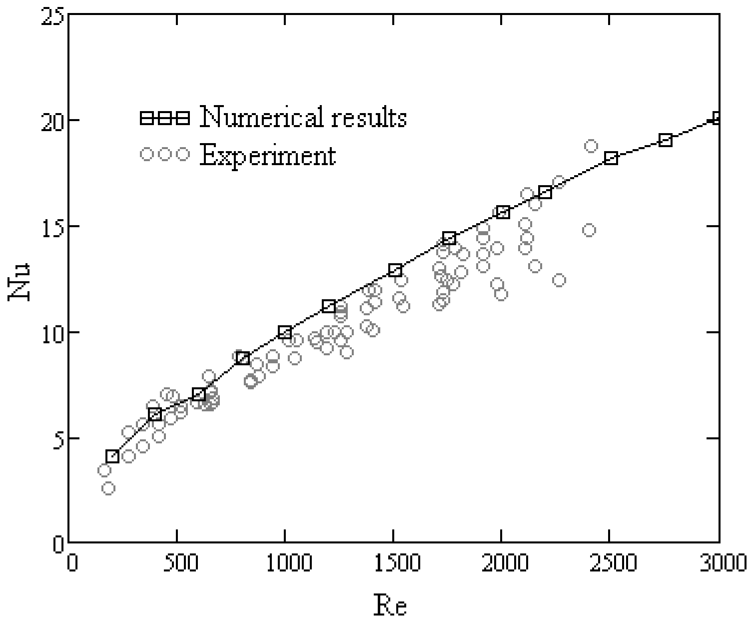

Figure 6 – Comparison of numerical modelling results with the experimental data

Apparently from figures 6-7 the received results have shown very good coordination between numerical modelling and the experimental and analytical data. The analysis of comparison in all cases has shown very good coordination of the data, hence, CFD package *SigmaFlow* can be applied to solve hydrodynamics and heat exchange problems.