European Journal of Mechanics B/Fluids 59 (2016) 7–17

Contents lists available at ScienceDirect

European Journal of Mechanics B/Fluids

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

Computational study of 3D rarefied gas flow in microchannel with 90 bend

O.I. Rovenskaya

Dorodnicyn Computing Centre, Federal Research Centre "Computer Science and Control" of Russian Academy of Sciences, Vavilova st. 40, 119 333 Moscow, Russia

ARTICLE INFO

Article history:
Received 6 February 2016
Received in revised form
17 April 2016
Accepted 19 April 2016
Available online 29 April 2016

Keywords:
Rarefied gas
90° microbend
Compressibility effect
Slip flow
Navier–Stokes equations

ABSTRACT

The pressure driven compressible gas flow through three-dimensional microchannel with 90° bend is numerically investigated using compressible viscous Navier–Stokes equations coupled with the first-order slip and Smoluchowski temperature jump boundary conditions. The results are compared to those from equivalent straight channel geometry in terms of mass flow rate and Poiseuille number. The competition between geometry, rarefaction and compressibility effects is discussed. The effect of slip boundary condition by comparison with no-slip results is estimated. The flow structure shows that 90° bend has in general a local effect and global performances are quite similar to those of a straight channel, allowing a certain freedom in the design of microdevices. However, it is found that the inclusion of a bend in a micro-channel produces a mass flow rate enhancement. Moreover, the pressure in bend is independent on an aspect ratio and behaves somewhat similar to two-dimensional microchannels, even for the square cross section. The velocity profile near the centerline is relatively similar along the width and depth for an aspect ratio of more than four, suggesting that the bent microduct can be modeled as a two-dimensional one, although, the velocity component along the depth is never identically zero, implying that the flow is not truly two dimensional.

1. Introduction

The continuous expansion MEMS devices applications, including e.g. micro power plants, microengines, microsensors etc., leads to an increase on activities in gaseous flow modeling at microscale, with a special regard to internal flows in microdevices. However, most papers in the open literature deal with studies in straight rectangular ducts or tubes, e.g. [1,2], despite the obvious fact that channel bends or curves, expansions or contractions, constrictions, and various shapes of inlets and outlets are very common in practice due to the design and technological constraints or limitations. In microdevices fabrication, e.g. micro heat exchangers, a serpentine channel design is generally used helping to keep the device compact and portable by fully utilizing the available surface area on the device.

In order to entirely characterize the performance of real microdevices, and possibly to exploit interesting features, it is essential to understand the effect of isolated corner element on the fluid flow. This characterization will assist in the design of efficient microfluidic systems by the selection of an appropriate driving force for a flow and/or chose a proper channel shape. Once the behavior of the bend is fully understood, it is possible to enhance its effect optimizing either pressure losses and/or heat transfer, combining several bends to create complex, or even fractal structures with high performances.

On the other hand, the detailed experimental measurements of fluid flows in arbitrary shaped microdevices are quite difficult and expensive. Some attempt has been done in [3] for miter, curved and double-bend configurations and demonstrated a decrease of the mass flow rate in all devices with respect to a straight microchannel. In particular, the mass flow rate through the miter bend was around 70% of the straight channel. Moreover, pressure measurements suggested a flow separation in a miter-bend microchannel. Authors implied, though could not demonstrate due to the complexity of local measurements, the onset of recirculation at very low Reynolds number *Re*. In [4], the pressure loss and heat transfer in elements like *L*-bends and T-joints and branching microchannels for *Re* ranging from 10 to 3000 were studied. An observed enhancement of heat transfer was caused by redirecting and splitting the flow, which at the same time increased the pressure loss. The characteristics of a pressure-driven water flow, including flow micro-structures and pressure drops, were investigated in serpentine micro-channels with miter bends using

---

E-mail address: olga_rovenskaya@mail.ru.

http://dx.doi.org/10.1016/j.euromechflu.2016.04.008
0997-7546/

a micron-resolution particle image velocimetry system (micro-PIV) in [5]. The experimental results showed that vortices around outer and inner walls of the bend appeared when $Re > 100$. Such vortices enlarged with increasing Reynolds number up to 1000, while for larger values their shape and size almost remain constant. The additional pressure drop due to the bend was commensurate with the strength of the appeared vortices. The growth of vortices strength with Re number caused the sharp increase of bend pressure drop. The experimental results also showed the bend pressure drop increased with decreasing hydraulic diameter.

Despite the study of geometrical shape of microchannel being a very important area in the design of MEMS there is a limited number of numerical works considering bent microchannels in the open literature. In particular, direct simulation Monte-Carlo (DSMC) method was applied for two-dimensional flow analysis in a rectangular corner in [6,7]. In [6] the compressibility effect on the gas flow through the bend was studied by ranging inlet pressure from $2\ 10^5$ to $3\ 10^5$ Pa, while Knudsen number was constant. The rarefaction effect was investigated in [7] varying inlet Knudsen number from 0.011 to 0.3 and fixing pressure ratio around 3. Both papers demonstrated that pressure and Mach number profiles along the bent channel are similar to that along a straight one, except for some peak value at the corner, indicating that no significant losses were introduced by adding a bend to micro-channel geometry. However, in [7], it was observed that in the range of inlet Knudsen number from 0.02 to 0.08 the mass flow rate along a microchannel with bend was slightly higher than the mass flow through a straight microchannel of the same length and subject to the same pressure difference. Although, further increase of the Knudsen number resulted in a bend mass flow decrease with respect to a straight channel one. The mass flow rate enhancement due to the bend was reported in [8] as well, using lattice Boltzmann method, but for the exit Knudsen numbers from 0.1 to 0.5, indicating that the rarefaction effect in gaseous flows may significantly affect the component performances.

Numerical investigations of a gas flow along complex geometry microchannel using less computationally expensive hydrodynamic approach have been done in [9–12]. In [9] a gas flow through microchannels with a 90° bend was studied using compressible viscous Navier–Stokes equations with the first-order slip boundary and temperature jump conditions. Only localized corner effect on flow performances, similar to those obtained by kinetic approach [6–8], was observed. However, the flow asymmetry in the corner led to high slip velocities and high velocity gradients on its inner side. The presence of a recirculation was detected on both the inner and outer walls of the corner for large Reynolds number, although the rarefaction delayed the onset of recirculation. The mass flux enhancement with respect to an equivalent straight microchannel one was detected, in agreement with kinetic approach [6–8]. A substantial reduction in mass flow rate through a micro-channel with two 90-degree bends at the outlet Knudsen number equal to 0.0585 was obtained in [10] using the compressible Navier–Stokes equations. The twisted geometry reduces the mass flow rate by approximately 160% than that for an equivalent straight microchannel. The effect of the fillet radius on flow characteristics using incompressible and compressible Navier–Stokes equations has been numerically analyzed in [11,12]. Results indicated that with increase of fillet radius the flow characteristics tend to equivalent straight channel ones.

It can be noticed that above mentioned papers are limited to two-dimensional geometries, however, as indicated in [13,14] three-dimensional effects in gas flows through a microchannel cannot be fully ignored. Only when a cross-section aspect ratio is larger than three, the flow pattern and heat transfer characteristics tend to those of two-dimensional results. However, the velocity component along the channel depth is never identically zero, implying that the flow is not exactly two dimensional.

Author's current understanding is that available literature does not fully explore pressure-driven flows through complex geometry microchannels considering combined effects of various factors such as rarefaction, compressibility, channel geometric arrangement and is generally limited by two-dimensional studies. Thus, the aim of the present paper is to partially fill this gap by examining a more realistic three-dimensional complex geometry under different flow conditions. The present work is devoted to numerical analysis of the flow characteristics in three-dimensional microchannels with 90° bend. Navier–Stokes equations coupled with first order Maxwell and Smoluchowski temperature jump boundary conditions are applied. Results are computed for a wide range of pressure ratio, exit Knudsen numbers and aspect ratios offering a map of competing effects such as geometry, rarefaction and compressibility, under different fluid dynamic conditions. The slip boundary condition role is estimated as well. The details of the three-dimensional flow structures, not easily accessible in experiments, are discussed in detail.

2. Statement of the problem

The three-dimensional pressure-driven nitrogen gas flow through a micro channel with 90° bend shown in Fig. 1 is considered. The geometry is a bent channel of width W and depth of $H = W/AR$, each channel's leg of length $L = 5 \times W$. Assuming an aspect ratio AR infinitely large we get the limited two-dimensional case. Inlet/outlet boundary conditions are standard and specify the inlet total temperature $T_{0i} = 300$ K, the total pressure p_{0i} and the flow direction. At the outlet the static pressure p_e is set equal to the ambient pressure, 10^5 Pa and $p_{0i} > p_e$. For easiness of boundary conditions implementation, the inlet free flow section of length $0.6W$ is introduced. The wall temperature T_w is equal to T_{0i}. The compressibility effect of the gas is monitored via the local value of Mach number Ma and the isentropic exit Mach number Ma_{is} (i.e., Ma that would arise from an isentropic flow with the same pressure ratio as the real one):

$$\frac{p_{0i}}{p_e} = \left(1 + \frac{\gamma - 1}{2} Ma_{is}^2\right)^{\frac{\gamma}{\gamma-1}}, \tag{1}$$

$$\frac{T_{0i}}{T_0} = \left(1 + \frac{\gamma - 1}{2} Ma_{is}^2\right). \tag{2}$$

$$\rho_0 = p_0/RT_0. \tag{3}$$

The gas flow in bent channel is driven by a pressure gradient. The static inlet pressure p_i is the result of computation. Although, for near incompressible flow, due to the low inlet velocities, p_i is close to the inlet total pressure p_{i0}.

The rarefaction of gas flow is characterized by the Knudsen number Kn based on the channel hydraulic diameter D_h:

$$Kn = \frac{\lambda}{D_h}, \tag{4}$$

$$D_h = \frac{2HW}{(H + W)}, \tag{5}$$

where λ is the molecule local mean free path. Reynolds number is computed using the channel hydraulic diameter D_h and a gas flow viscosity μ and velocity u:

$$\mathrm{Re} = \frac{\rho u D_h}{\mu} \tag{6}$$

and related to the Knudsen and Mach numbers by the following relationship:

$$Kn \approx \sqrt{\frac{\gamma\pi}{2}}\frac{Ma}{\mathrm{Re}}. \tag{7}$$