Physica A 492 (2018) 2019–2026

Contents lists available at ScienceDirect

Physica A

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

Partial obstruction of flow through a channel

A.D. Araújo [a,*], Izael A. Lima [a], M.P. Almeida [a], J.B. Grotberg [b], José S. Andrade Jr. [a]

[a] Departamento de Física, Universidade Federal do Ceará, Campus do Pici, 60451-970 Fortaleza, Ceará, Brazil
[b] Department of Biomedical Engineering, University of Michigan, Michigan, MI 48109-2099, United States

ARTICLE INFO

Article history:
Received 22 May 2017
Received in revised form 7 October 2017
Available online 21 November 2017

Keywords:
Fluid flow
Drag force
Numerical simulation
Reynolds number

ABSTRACT

We study the disturbance on two-dimensional flow generated by a circular obstacle of radius r placed downwind in front of a duct of width w at a distance λ between the center of the obstacle and the inlet position of the channel. Our results show that, at low Reynolds conditions, the flux ϕ at the duct exhibits distinct regimes for different λ intervals.

1. Introduction

The hydrodynamical interaction between a flowing fluid and suspended particles is an interesting problem with important applications in science and technology [1–3]. There are several situations where microscale suspended soft particles might clog the duct cross section disturbing or even interrupting the flow. When the fluid is viscous and the particles are deformable, they can readily undergo large deformations to accommodate the hydrodynamic forces. This process is observed in many situations and scales, from blood flow in microscopic vessels up to sewage flow in macroscopic pipes. In vasculatory systems, we find some extreme cases where the cross section of the capillary vessel and the size of the blood cells that flow through them have the same order of magnitude [4]. In this case, the cells must undergo a large deformation in order to travel along the vessels. In some blood diseases the deformability potential of the red cells is reduced [5,6] blocking the flow through microvascular vessels. In other cases the blood can coagulate to form large clusters that behave like a solid [7]. As a consequence of these processes, in the extreme limit, the fluid flow is eventually interrupted. It is therefore important to understand how the presence of an obstacle with the same dimension of the duct width disturbs the flow and changes the flux across this duct.

Due to the broad applicability of this problem, many approaches and techniques have been used to simulate the interaction between the fluid and particles. Some of them belong to a group of pseudo-particle methods that form a class of multi-scale simulation approaches in computational fluid mechanics. Among them, we have lattice-based cellular automata methods (lattice gas, lattice Boltzmann) [8] and off-lattice approaches (dissipative particle dynamics, direct simulation Monte Carlo, multiple particle collisions) [9–12]. Alternatively, one may solve the Navier–Stokes equation directly finding the pressure and velocity fields [13,14] and then introduces particles that can interact or not with the fluid.

The interaction of a particle with the fluid flow within a channel has several characteristic lengths which are related to the particle and channel geometrical dimensions [15]. A straightforward way to approach this problem, consists basically in solving the Navier–Stokes equation in the presence of particles for a set of boundary conditions, calculating the velocity and the pressure fields. Subsequently, the particles are moved by a very small distance, according to the drag forces exerted

* Corresponding author.
E-mail address: ascanio@fisica.ufc.br (A.D. Araújo).

https://doi.org/10.1016/j.physa.2017.11.117
0378-4371/

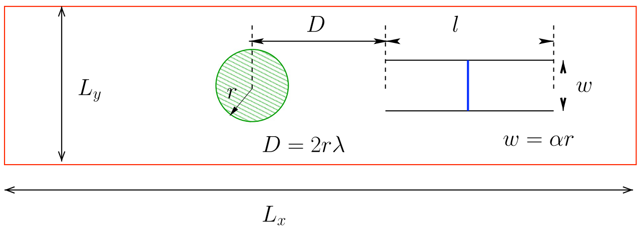

Fig. 1. Schematic view of the problem investigated here. Basically, the geometry consists of a box with length L_x and width L_y. The channel has a length l and a tunable width $w = \alpha r$, where α is a dimensionless parameter. The circle obstacle with radius r is located in front of the channel with its center distant $D = 2r\lambda$ from the entrance, where λ is a dimensionless parameter.

on them, the velocity and pressure fields are calculated again, and so on. Using this numerical calculation sequence, we can determine how each particle interferes in the flow around its surface and force acting on it. Furthermore, we can apply this idea to study a traveling particle transported by a fluid approaching to a narrow channel.

The main purpose of this paper is to study how the flow disturbance caused by a solid obstacle inside a duct influences the flux through a channel placed downwind the obstacle as a function of the distance between the obstacle and the channel entrance. We measure the flux ϕ through a channel placed a distance λ downwind the obstacle. For this, we numerically solve the Navier–Stokes equation to compute the flux ϕ and the drag force over the obstacle as a function of λ. Our results indicate the occurrence of distinct flow regimes for different values of λ, with the flux ϕ decaying as a power-law in the limit $\lambda \to 0$. Finally, the presence of a minimum in the drag force over the particle is also observed for a given λ value.

2. Model

We study the system shown in Fig. 1 that consists of a two-dimensional duct with length $L_x = 100$ m and width $L_y = 20$ m, inside which there is a smaller channel of length l and width w with its longitudinal axis aligned along the axis of the enclosing duct. A circular solid obstacle with radius $r = 1$ m is placed at a distance D upwind the entrance of the smaller channel with its center on the duct's axis. The various geometric configurations we use are described in terms of two non-dimensional parameters, namely, $\lambda = D/(2r)$ and $\alpha = w/r$, with $\alpha > 0.5$ for all simulations presented. In order to avoid border effect, the width of the duct L_y is at least one order of magnitude larger than the channel width w. We impose no-slip boundary conditions along the entire solid–fluid interface. At the inlet ($x = 0$), we impose the conditions, $u_x(0, y) = V$ and $u_y(0, y) = 0$, while the boundary condition at the outlet ($x = L_y$) is imposed to be $\nabla p = 0$. The Reynolds number is defined as $Re = \rho Vr/\mu$, where ρ and μ are, respectively, the density and the viscosity of the fluid, and V is the velocity at the inlet section. We take $\rho = 1$ kg/m^3 and $\mu = 1$ kg/(m s).

We use the CFD software Fluent (Ansys, Inc.) [16] to numerically compute the steady-state solution of the two-dimensional flow of an incompressible Newtonian fluid in terms of the Navier–Stokes and continuity equations,

$$\rho\vec{u} \cdot \nabla\vec{u} = -\nabla p + \mu\nabla^2\vec{u} \tag{1}$$

$$\nabla \cdot \vec{u} = 0 \tag{2}$$

where $\vec{u}$ is the velocity and p the pressure.

In Fig. 2 we show a typical velocity field obtained from numerical simulations in a regime of low Reynolds number, $Re = 0.356$, for three different values of λ and fixed width w. The colors ranging from blue to red correspond to low and high velocity magnitudes, respectively. The contour plot of the velocity magnitude clearly reveals that, as the obstacle is approaching the channel entrance, the wake region of the obstacle, shown as the dark region behind it, reaches the channel and the obstacle begin to play a more significant role on the flow towards the channel.

3. Results and discussion

In order to quantify the obstacle influence on the channel flux, we calculate the flux ϕ through the channel, as the integral of the velocity along the linear distance orthogonal to the flow times the length of this line. In our analysis we have normalized the flux ϕ by the total flux ϕ_0 calculated as the same of the flux ϕ but now considering the entrance of the duct. In that case, ϕ_0 is the total flux getting inside the duct while ϕ is the fraction of this flux that goes inside the inner channel. Fig. 3 shows a log-linear plot of the normalized flux (ϕ/ϕ_0) as a function of λ, for five different values of w. Three distinct regimes of ϕ/ϕ_0 can be clearly identified. In the limit of large λ ($\lambda \geq 10$), the flux ϕ tends to a saturation value ϕ_s which depends on the width w as a power law, $\phi_s \propto w^{2.66}$, as the collapse of all curves in this region indicates (see the inset of Fig. 3).