Flow investigation around cylinders assembled in a confined straight channel geometry

Nicoleta - Octavia Tanase¹, Diana Broboana¹, and Corneliu Balan¹

¹University Politehnica Bucharest, HFMEE Department, 060042 Splaiul Independentei 313, Romania
octavia.tanase@upb.ro

Abstract. The paper is dedicated to experimental and numerical investigations of the dynamics of 3D-flow around a cylinder with 10 mm diameter. The cylinder is positioned in a confined geometry represented by a long straight channel with 50 mm width and 10 mm height. The study analyses the dependence on the Re-number of the path lines distributions in the vicinity of the immersed cylinder. The visualized flow patterns are fairly reproduced by the numerical solutions of the Navier-Stokes equations up to Re = 1000. The velocity and wall stresses are computed in the vicinity of the cylinder. One main goal of the present work is established by the correlation between the flow pattern, wake boundary and wall shear stress distributions downstream the cylinder.

1. Introduction

The incompressible flow of viscous fluids past bluff bodies as circular cylinders and spheres represents a classical/benchmark problem in fluid mechanics, due to numerous engineering applications of flow dynamics around cylindrical configurations [1-4].

Jeong and Yoon [3] investigated the two-dimensional Stokes flow around a circular cylinder in a microchannel. This problem is important in chemical technology and bio-MEMS technology related to the transportation of particles in microchannels.

The three-dimensional flow structure and time evolution of the vorticity pattern in cylinder wakes was analyzed by Scarano F. and Poelma C. [4]. The experiments were performed in a water tunnel with time-resolved tomographic PIV, at Reynolds number ranging from 180 to 5540.

Haward S.J. et al. [5] investigated the flow of a well-characterized dilute viscoelastic polymer solution, and its viscous Newtonian solvent, around a single cylinder with radius \( r = 20 \mu m \) and also around two identical cylinders aligned on the flow axis and separated by a significant distance \( L = 50r \). The flow field is characterized in significant detail using micro-particle image velocimetry (\( \mu \)PIV) and we also perform flow birefringence measurements to provide complementary information about the regions of high polymer stretching and elongational stress.

Hale R. S. et al [6] presented the importance of the micropillar arrays for controlling capillary flow in microfluidic devices, and array permeability is a key parameter in determining fluid flow rate. Numerical simulations of pillar permeability are the most desirable due to their accuracy. For pillars arranged in a square pattern, the 2D analytical solution proposed in this study performs well at short pillar heights while the Brinkman equation is more accurate at tall pillar heights. These findings are applied to capillary fluid flow in heat pipes to explore the effects of pillar spacing, diameter, and height on the maximum fluid flow rate through the wick.
The flow field over a low aspect ratio (AR) circular pillar ($L/D = 1.5$) in a microchannel was studied experimentally by Jung J. et al. [7]. Microparticle image velocimetry (µPIV) was employed to quantify flow parameters such as flow field, spanwise vorticity, and turbulent kinetic energy (TKE) in the microchannel. Flow regimes of cylinder-diameter-based Reynolds number at $100 < Re < 6700$ were elucidated at the microscale.

Lee S. J. et al. [8] investigated numerical the flow around a pair of parallel rectangular cylinders placed perpendicular to the direction of the flow using the immersed boundary method at a fixed Reynolds number of 100. The two cylinders are arrayed in inline and a staggered arrangement. The pattern of the wake of the two cylinders depends on their arrangements and spacing. The flow characteristics depend on the flow regime, including the flow structure, drag force, lift force, and frequency. We analysed the flow characteristics by comparing the flow regimes, vortex shapes, drag and lift coefficients, and Strouhal numbers, which depended on the arrangement.

The aim of the paper is to investigate the evolution of the downstream cylinder’s wake pattern, in relation the positions of the detachment points and the value of the Re-number. The influence of the Re-number on the flow spectrum around the cylinder is analysed based on the correlation between direct flow visualizations and numerical simulations performed with the ANSYS Fluent code.

The expression of the Re-number is given by:

$$Re = \frac{\rho v_0 D}{\eta},$$

where $D$ is the cylinder’s diameter, $v_0$ is the average upstream velocity. The working fluid is water with $\rho = 1000$ kg/m$^3$ mass density and viscosity $\eta = 1$ mPas.

2. Experimental set-up

The flow in the experimental set-up (figure 1) is produced by a centrifugal pump that supplies a constant level of water in the tank. The transparent channel, where the cylinder is positioned, is connected to the tank. To obtain a stable velocity distribution in the channel a diffuser and a contraction are used as the entrance and exit sections of the channel, respectively. The transported flow rate through the channel during the experiments is measured using volumetric technique.

The transparent plexiglass straight vertical channel has the length $L = 400$ mm, the width $w = 50$ mm and the height $h = 10$ mm. The cylinder with the diameter $D = 10$ mm and the height $h$ is positioned in the middle of the microchannel, at the distance $a = 250$ mm from the entrance section (figure 2).

The flow domain and configuration (same dimensions with the experimental channel), with the details of the cylinder and the corresponding mesh used in numerical simulations are shown in figure 2.

A coloured ink is introduced in the flow field through needles located in entrance section of a channel, far upstream the cylinder, using a syringe pump. The seeds particles are aligned with the flow streamlines and their visualizations put in evidence the location of the separation points on the cylinder’s surface and the downstream flow patterns and wakes.

In figure 3 are presented the direct visualizations for the flow spectrum around a cylinder using a Sony SLT high resolution digital camera. The separation line between the main flow and the wake region and the positions of the detachment point with Reynolds number are observed, respectively.
3. Numerical simulations

The 3D numerical solutions were performed with laminar incompressible solver implemented in the numerical code ANSYS Fluent [12]. The geometry and mesh are presented in figure 2. The mesh used in numerical simulations consists of 1,740,580 hexahedral cells and 1,849,302 nodes.

The boundary conditions are: constant velocity at the inlet, $v = v_0$, constant pressure, $p = p_0$ at the outlet of the channel, adherence conditions on the walls (upper, lower and lateral walls) and on the surface of the cylinder. The steady Navier-Stokes solutions are obtained at a convergence of $10^{-6}$ residuals for the velocities and the continuity equation.

In figure 4 is presented a qualitatively comparison between numerical and experimental results of the flow pattern around a cylinder. The numerical flow spectrum represented by the path lines distribution around a cylinder fit well the experimental flow visualizations from figure 3.
Figure 4. Comparison between experimental and numerical flow spectrum around a cylinder at different Reynolds numbers. The detachment points and the wake region are marked (see figure 3).

Using the numerical solutions, the vortical structures downstream the cylinders and the corresponding velocity magnitude distributions are analyzed.

In figure 5 are plotted the distributions of the path lines colored by velocity magnitude. The length of the vortices downstream of the cylinder increases linear with the magnitude of the Re-number.

Figure 5. The flow spectrum in the median plane with (the trajectories colored by velocity magnitude). Recirculation region is observed. The median plane is positioned at $h/2$ (where $h$ is the channel height).

The comparison of the velocity and pressure distributions on planes upstream and downstream the cylinder is shown in figure 6 for three Re-numbers.

Figure 6. The pressure distributions on normal planes to the flow, upstream and downstream the cylinder at: a) $Re = 40$, b) $Re = 150$, c) $Re = 350$. 


The distributions of the velocity magnitude on the lines L1, L2, L3 and L4 (figure 7) for $Re = 40, 150, 350$ are shown in figure 8. The area of interest around the cylinder is observed in the graphical representations, where the flow is disturbed by the presence of cylinder.

**Figure 7.** The geometry of the lines positions in the median plane of the microchannel corresponding to different $y$ coordinates from the inlet ($y = 255$ (L1); $265$ (L2); $275$ (L3), $285$ (L4), the units are millimeters); b) extrados and intrados of the circle and critical points positions in median plane $z$.

**Figure 8.** The velocity magnitude distribution on perpendicular lines to the flow direction (represented in the figure 7).
Figure 9. The components of wall shear stress distribution on extrados circle in median plan \( z \) and the locations of critical points (A, D1 and F) at three Reynolds numbers (the geometry of circle is represented in the figure 7).

Numerical solutions give the insight of all flow quantities, one of the most important being the WSS distribution on the cylinder, see figure 9.

The separation point is the point where the wall shear stress is equal to zero and the flow (path lines) is detached from the cylinder wall [9÷11].

Plotting the components of WSS as function of the length the circle in the \( y \)-direction, one can determine the location of critical points (impact and separation points) on the immersed cylinder.

4. Conclusions

The study was focused to the experimental and numerical investigations of the flow around a cylinder assembled in a confined straight channel geometry.

Experimental investigations were limited up to now only to the direct visualization of the flow spectrum around a cylinder.

The 3D numerical simulations were performed with laminar solver implemented in the code ANSYS Fluent.

The numerical solutions for flow around a cylinder was confirmed by direct visualizations (figure 4). The shape and characteristics of vortices that develop downstream the cylinder were numerically investigated.
Figure 10. The trajectories colored by velocity magnitude and the wall shear stress distribution on the cylinders. It is observed that the path lines are detached from the cylinder wall in the points where the wall shear stress is equal to zero (see figure 9).

References

[1] Myers LE and Bahaj AS 2010 Ocean Eng. 37 218
[2] Chamorro L P, Hill C, Morton S, Arndt R E A, Ellis C and Sotiropoulos F 2013 J Fluid Mech 716 658
[3] Jeong J-T and Yoon S-H 2014 JMST 28(2) 573
[4] Scarano F and Poelma C 2009 Exp Fluids 47(1) 69
[5] Haward S J, Toda-Peters K and Shen A Q 2018 J. Non-Newtonian Fluid Mech. 254 23
[6] Hale RS, Bonnecaze RT and Hidrovo CH 2014 Int J Multiphase Flow 58 39
[7] Jung J, Kuo C-J, Peles Y and Amitay M 2012 Int J Heat Fluid Fl 36 118
[8] Lee S J, Mun G S, Park Y G and Ha M Y 2019 J Mech Sci Technol 33(7) 3289
[9] Tănase N O, Broboană D and Bălan C 2014 Scientific Bulletin UPB S D 76(2) 259
[10] Tănase N O, Broboană D and Bălan C 2014 Proc. Romanian Acad. Series A 15 371
[11] Broboană D, Muntean T and Bălan C 2007 Proc. Romanian Acad. Series A 8 219
[12] ***ANSYS Fluent Theory Guide, ANSYS, Inc 2013.