U.P.B. Sci. Bull., Series D, Vol. 76, Iss. 2, 2014 ISSN 1454-2358

FLOW AROUND AN IMMERSED CYLINDER IN THE PRESENCE OF FREE SURFACE

Nicoleta Octavia TĂNASE[1], Diana BROBOANĂ[2], Corneliu BĂLAN[3]

The paper is dedicated to the experimental and numerical studies of the flow around an immersed cylinder, in the presence of free surface. The experiments are performed in a 2D channel, the flow rate being controlled by a weir. Qualitative direct visualizations of the flow patterns and quantitative PIV measurements of the velocity magnitude have been performed. Numerical simulations are performed with turbulent solvers implemented in FLUENT, using the VOF code for the calculation of the free surface geometry. The final goal of the work is to establish the interaction between the free surface and the flow kinematics around the cylinder.

Keywords: free surface, immersed cylinder, experimental investigations, CFD

1. Introduction

The flow around a cylinder represents a benchmark problem of fluid mechanics, with many practical applications in aerodynamic, hydrology, hydraulic, hydraulic turbines, weirs, marine platforms, inflatable dams, [1-3].

One of the actual research subjects related to this topic is the modeling of the flow in vicinity of an immersed cylinder and the determination of the free surface influence on the flow spectrum downstream the cylinder, [4-8].

Theoretically, numerical and experimental investigations of this flow are characterized as a function of two non-dimensional parameters: Reynolds number and Froude number, [9-11], respectively:

$$Re = \frac{\rho \, V_0 \, D}{\eta}, \; Fr = \frac{V_0}{\sqrt{gD}}, \tag{1}$$

where V_0 is the average velocity upstream of the cylinder, D is the diameter of the cylinder, ρ is the density, η is the viscosity and g is the gravitational acceleration.

The main goal of this work is to establish a numerical procedure for computation of the free surface geometry developed in vicinity of an immersed cylinder, at given Reynolds and Froude numbers.

[1] Assistant Professor, REOROM Laboratory, Power Engineering Faculty, University POLITEHNICA of Bucharest, Romania; e-mail: notanase@yahoo.com, corresponding author
[2] Associate Professor, REOROM Laboratory, Power Engineering Faculty, University POLITEHNICA of Bucharest, Romania
[3] Professor, PhD, Power Engineering Faculty, University POLITEHNICA of Bucharest, Romania

2. Experimental set-up

The experiments are performed in a free surface water channel which allows the direct vizualisation of the flow, whose cross-section is a rectangle with maximum height $H_{\max} = 150$ mm and width $B = 15$ mm. The cylinder with diameter $D = 50$ mm is mounted on the channel wall at the distance mm from the entrance section. The total length of the channel is $L = 1232$ mm, see figure 1. In the middle of the channel is fixed a weir of thickness $b_{dev} = 6$ mm that controls the flow rate and the free surface level, see also figure 4.

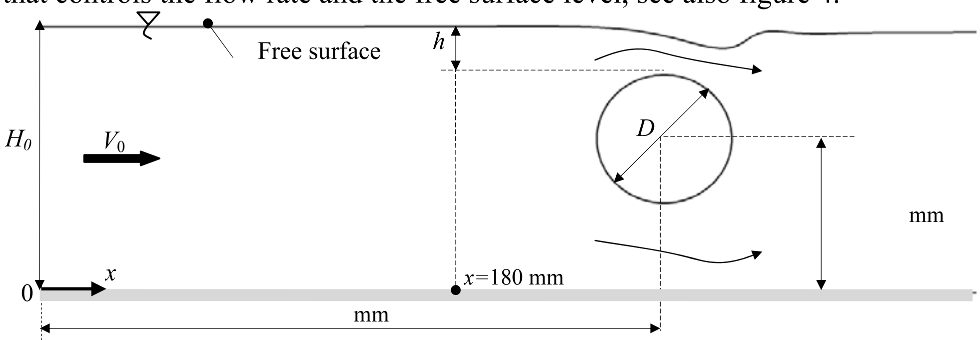

Fig. 1. A sketch of the flow configuration and geometry. Here V_0 is the average velocity in the channel, Q_1 and Q_2 being the flow rates transported up and down the immersed cylinder (the values of H_0 and V_0 are maintained constant for one measurement).

The tracer to mark the flow pattern is a colored blue dye introduced upstream the cylinder. The visualization of the wake and the location of separation points on cylinder surface are recorded for the tested flow regimes, [12].

During the measurements, the level was varied in the interval mm, keeping the height of the weir constant, mm. The free surface geometry (FS lines), the wake and the position of separation points (D_1 and D_2, respectively) obtained from direct visualization of flow are shown in the figure 2.

The flow characteristics and the values of Reynolds and Froude numbers are given in table 1. The free surface line was recorded using images taken with a Sonny digital camera at a frequency of 12 frames per second, see figure 3. Experimentally, it is noticed that flow is stationary upstream the cylinder, but downstream the cylinder the oscillations of the free surface are observed.

The experiments are performed in sub-critic ($Fr < 1$) transitory regime, $4000 < Re < 12000$, which is normally characterized by weak turbulence, [4-5], [12]. Under this working conditions, the “chaotic fluctuations” of free surface are expected, especially (for this particular flow configuration) downstream the detachment of boundary layer from the cylinder surface (marked by the location of critical point D_1, see figure 2).

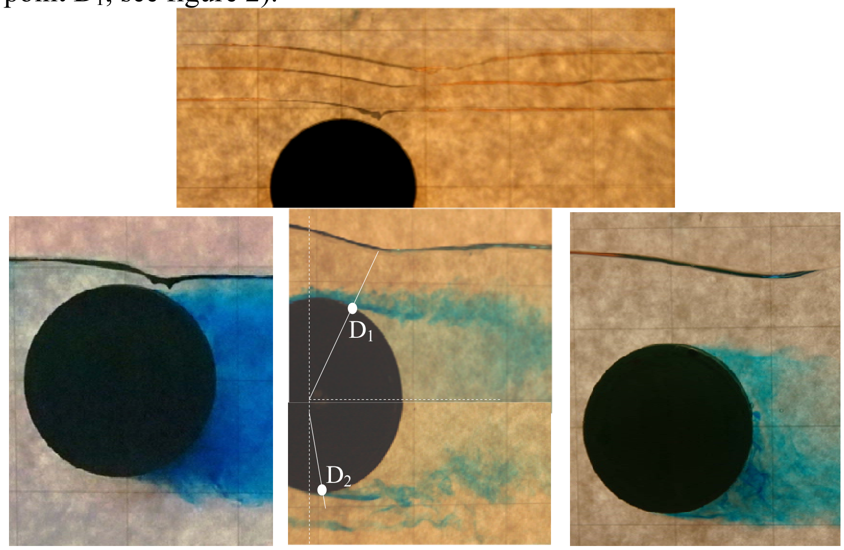

Fig. 2. Direct visualization of the steady free surface and the downstream wake: FS1 - H_0 = 92 mm, FS2 - H_0 = 105 mm, FS3 – H_0 = 115 mm, see Fig. 1.

Table 1

Flow characteristics and the values of Reynolds and Froude numbers

[mm]	[mm]	[m^3/s]	[m/s]	Re [-]	Fr [-]
92	7	0.1457	0.1	5 000	0.143
105	20	0.2415	0.15	7 500	0.214
115	30	0.3289	0.2	10 000	0.285

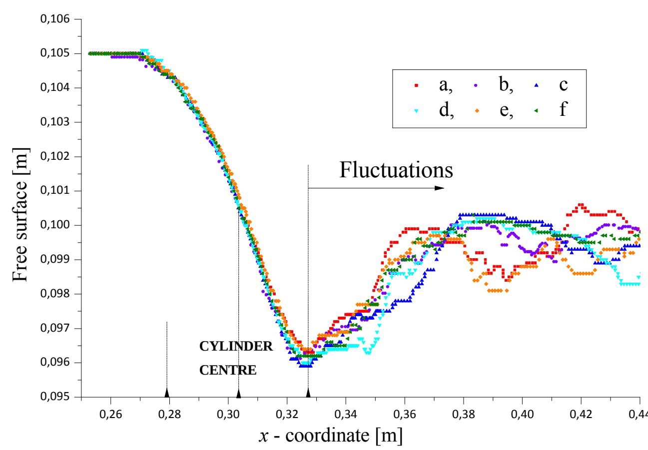

Fig. 3. Free surface geometry recorded at six moments in 1 second; the fluctuations are observed downstream the cylinder for steady imposed flow regime upstream the immersed body.

3. Numerical simulations

The main aim of the numerical study is to establish the best computed shape of the free surface. We are also looking to investigate the flow kinematics around the cylinder, in particular to determine: (i) the flow spectrum downstream the cylinder (e.g. velocity and turbulent intensity fields developed in the wake) and (ii) the drag force acting on the cylinder.

Numerical simulations have been performed with different turbulent solver implemented in FLUENT code. The best results are obtained using the k-ε RNG model, associated with the VOF code for the calculation of the free surface geometry, [13-14].

The boundary conditions shown here are the following: (i) entrance water ( ): the height mm kept constant, the linear pressure distribution imposed, , ; (ii) entrance – air; (iii) exit: constant atmospheric pressure, ; (iv) weir and wall: (adherence condition); for details see figure 4.

The numerical simulations are performed in 2D configuration with a structural mesh characterized by 638436 cells, 1270606 faces and 632170 nodes.

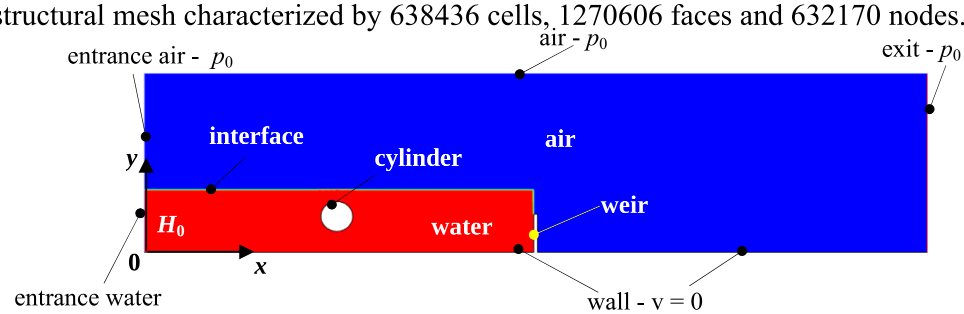

Fig. 4. Numerical domain and the boundary conditions implemented in FLUENT code.

The influence of the surface tension on the flow structure downstream the cylinder was also analyzed. In figure 5, the flow spectrum downstream the cylinder with and without the influence of the surface tension is presented. The presence of the surface tension does not bring major qualitatively changes on the boundary layer separation from the cylinder and on the flow pattern of the downstream wake.

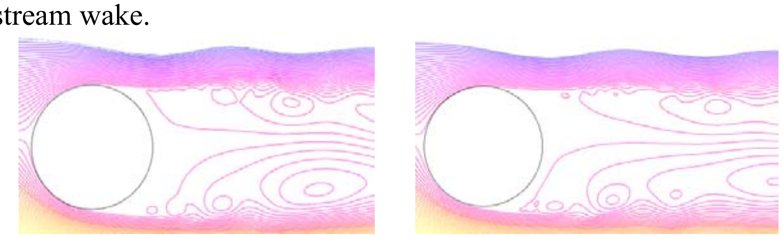
a) b)

Fig. 5. Numerical flow patterns (streamlines) at H_0 = 105 mm: a) presence of the surface tension (C1), b) no surface tension influence (C2)

A comparison between the experimental free surface line and the computed free surfaces (with and without the surface tension) is disclosed in figure 6. It can be noticed that oscillations of the free surface downstream the cylinder were fairly reproduced, even if the turbulent solver is steady (of course, in this case, the fluctuations are not obtained).

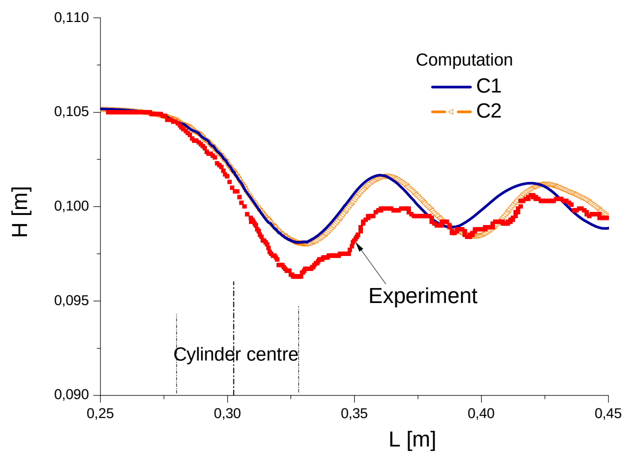

Fig. 6. Experimental free surface *vs*. computed free surface (case C1 and C2), see Fig. 5.

There are several differences recorded between the numerical reconstruction of free surface and the experimental pictures, mainly due to the 3D curvature of the free surface observed in vicinity of the immersed cylinder and the capillarity phenomena present at the walls of the channel. As consequence, the visualized free surface is a band of maximum thickness of 1.5 mm. Since the free surface line was considered as the lower limit of this band, the results from figure 6 are expected (respectively, the experimental free surface is always the lower limit of the computed free surface).

In figure 7 (a, b) the turbulent intensity and the velocity vector distribution downstream the cylinder are disclosed. The turbulent intensity is maximum in the vicinity of free surface and minimum upstream the cylinder, within the recirculation area behind the cylinder.

The distribution of the velocity vector around the cylinder highlights the wake region and the presence of vortices on the flow separation line. This flow pattern is associated to the Kelvin-Helmholtz instability, [12].

Experimentally, the wake is evidenced by direct visualization in figure 7c.

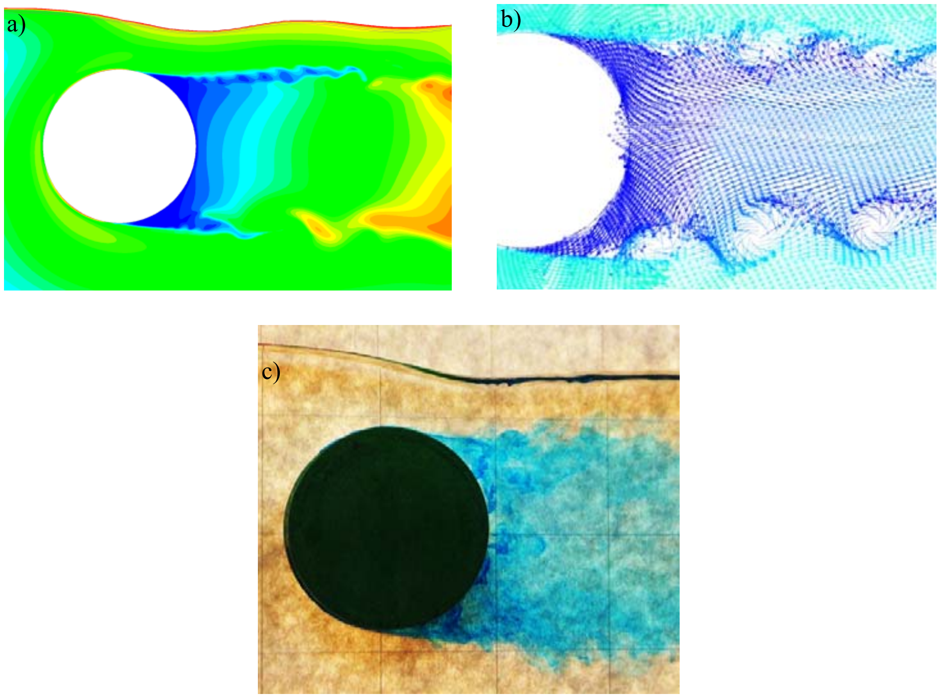

Fig. 7. Case C2: a) turbulent intensity (red - maximum value on the free surface),
b) velocity vectors downstream the cylinder (numerical computation),
c) the wake downstream the cylinder (direct visualization)

The benefit of numerical simulations is not only the reproducibility of the flow patterns, in particular of the free surface, but mainly the possibility to compute relevant hydrodynamics quantities for the designed applications. In Table 2 are shown the computed drag and lift forces/coefficients for the simulations associated with the investigated cases of study. Unfortunately, we cannot compare these figures with other data, since in the scientific literature similar computations for the immersed cylinders do not exist (the published studies in the domain are limited to the analysis of flow patterns and their simulations, see [4-11]).

Table 2

Hydrodynamic characteristics computed from the numerical simulations

[mm]	[m/s]	[-]	[-]	[N]	[-]	[N]	[-]
92	0.108	5400	0.154	1.545	5.3	17.91	61.3
105	0.146	7300	0.208	2.112	3.938	18.832	35.104
115	0.187	9350	0.267	2.76	3.173	18.9	21.73

4. Conclusions

In this paper, numerical and experimental investigations of the flow around an immersed cylinder, in the presence of free surface were conducted.

The numerical simulation of this particular hydrodynamics is used to characterize the motion in vicinity of the cylinder and to trace the free surface geometry. The numerical procedure to compute the free surface was validated by the experimental visualizations. Free surface computations in turbulent flows are important for many applications, but also to better understand the interaction between the wake developed behind the immersed bodies and the local geometry of the free surface. As results of computations, one can obtain valuable information about the magnitude of the immersed cylinder drag and lift coefficients for different associated flow regimes.

Comparisons between numerical and experimental results of the investigated free surface flow were made mainly qualitatively. To validate the numerical procedure we also have to obtain consistent quantitative results. This work is at the beginning. The first results are promising; they are based on the PIV measurements of velocity field in the domain between the cylinder and the free surface, see figure 8.

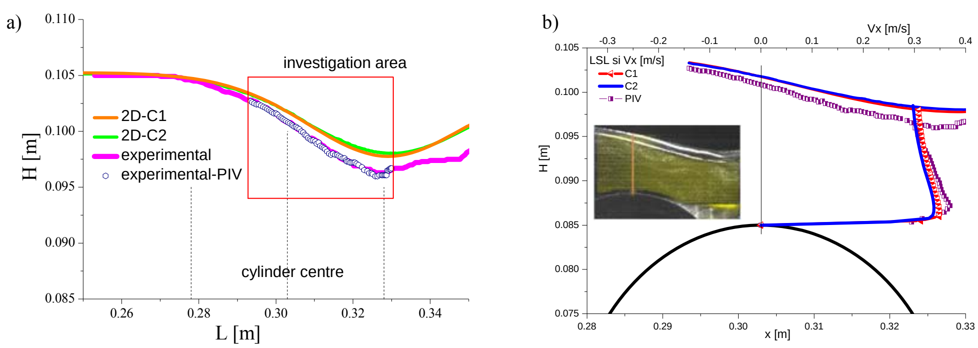

Fig. 8. Comparison between computed free surface and velocity distribution with PIV experiments

In conclusion, the present study discloses a good qualitative and quantitative agreement between numerical and experimental investigations of the free surface flows. The 2D numerical model of the channel flow gives fair results with relatively small computer resources. For the range of weak turbulence developed in vicinity of immersed bodies, the $k-\varepsilon$ RNG turbulence model (the steady solver) associated with the VOF code offer the best results and a good approximation of the free surface geometry.

Further experimental studies will be focused on two directions: (i) quantitative validations of the numerical simulations at smaller Reynolds numbers; (ii) the influence of patterned cylinder surface on the local geometry of free surface, especially for small immersed depths.

Acknowledgement

This work was supported by grant of the Ministry of National Education, CNCS – UEFISCDI, project number PN-II-ID-PCE-2012-4-0245; Nicoleta Octavia Tanase acknowledges also the financial contribution from the post-doctoral project InooRESEARCH POSDRU/159/1.5/S/132395.

R E F E R E N C E S

[1]. *L.E. Myers, A.S. Bahaj,* "Experimental analysis of the flow field around horizontal axis tidal turbines by use of scale mesh disk rotor simulator" in Ocean Engineering, **vol. 37**, 2010, pp. 218–227.

[2]. *L. P Chamorro, C.Hill, S.Morton, R. E. A. Arndt, C. Ellis, F. Sotiropoulos,* "On the interaction between a turbulent open channel flow and an axial-flow turbine", in J. Fluid Mech., **vol. 716**, 2013, pp. 658-670.

[3]. *M.C. Ong,* Applications of a standard high Reynolds number k-ε model and a stochastic scour prediction model for marine structures*,* PhD. Thesis. Trondheim, Norwegian University of Science and Technology, 2009.

[4]. *J. Sheridan, J-C. Lin, D. Rockwell, "*Flow past a cylinder close to a free surface", in J. Fluid Mech., **vol. 330**, pp. 1-30, 1997.

[5]. *P. Reichl, K. Hourigan, M. C. Thompson,* "Flow past a cylinder close to a free surface", in J. Fluid Mech., **vol. 533**, pp. 269–296, 2005.

[6]. *S.J. Lee, Daichin,* "Flow past a circular cylinder over a free surface: Interaction between the near wake and the free surface deformation", in J. Fluids and Structures, **vol. 19**, pp. 1049–1059, 2004.

[7]. *P. Oshkai, D. Rockwell,* "Free surface wave interaction with a horizontal cylinder", in J. Fluids and Structures, **vol. 13**, pp. 935-954, 1999.

[8]. *J. Tian, V. Roussinova, R. Balachandar,* " Characteristics of a jet in the vicinity of a free surface", in J. Fluids Eng., **vol. 134**, pp. 1-12, 2012.

[9]. *J. W. Hoyt, R. H. J. Sellin,* "A comparison of tracer and PIV results in visualizing water flow around a cylinder close to the free surface", in Exp. Fluids, **vol. 28**, pp. 261-265, 2000.

[10]. *S. Dong, G. E. Karniadakis, A. Ekmekci, D. Rockwell,* "A combined direct numerical simulation –particle image velocimetry study of the turbulent near wake", in J. Fluid Mech., **vol. 569**, pp. 185–207, 2006.

[11]. *J. H. Seo, J. H. Jung, H. S. Yoon, D-W. Park, H-H. Chun,* "Laminar flow structures near a circular cylinder in between a free-surface and a moving wall", in J. Society of Naval Architects of Korea, **vol. 49(3)**, pp. 213-221, 2012.

[12]. *N. O. Tănase*, Modelarea curgerilor tranzitorii în jurul corpurilor aflate în vecinătatea suprafeței libere în regimul subcritic, Ph.D. Thesis, Power Engineering Faculty, University "Politehnica" of Bucharest, Romania, 2013 (in Romanian).

[13]. *B. E. Launder, D. B. Spalding*, Mathematical models of turbulence, Academic Press, London, 1972.

[14]. ***Fluent Inc., Fluent 6.3 User's Manual, 2008.