Study of Dynamic Field around a Vertical Circular Cylinder placed in an Open-Chanel Flow

M. Tchawe¹, B. Djeumako¹, C. Koueni-Toko², D. Tcheukam-Toko³* and A. Kuitche⁴

¹Department of Mechanical Engineering, University of Ngaoundere, ENSAI
P. O. Box 455 Ngaoundere, Cameroon

² Departments of Renewable Energy, ISS, University of Maroua
P. O. Box 72 Maroua, Cameroon,

³ Department of Energy Engineering, University of Ngaoundere, IUT
P. O. Box 455 Ngaoundere, Cameroon

⁴Department of Electrical, Energy and Automatism Control Engineering, University of Ngaoundere, ENSAI
P. O. Box 455 Ngaoundere, Cameroon

Abstract
Numerical analysis was performed for the two-dimensional open-channel flow around a vertical circular cylinder emerged. Special attention has been paid for the interaction of vortexes structures around a cylinder. The volume of fluid (VOF) model, coupled to the turbulent model has been applied a standard $\kappa-\varepsilon$ two equations model and the two-dimensional Reynolds Averaged Navier–Stokes (RANS) equations are discretized with the second order upwind scheme. The SIMPLE algorithm, which is developed using control volumes, is adopted as the numerical procedure. Calculations were performed for a wide variation of the Reynolds numbers (Re), corresponding to different flows. The results reveal that with increasing Reynolds number, the pressure increases and the velocity decreases at the upstream zone from the cylinder. This velocity becomes maximal at the cylinder plane $45° \leq \alpha \leq 90°$, and becomes zero at the plane $\alpha = 0°$. These results reveal the importance of a cylinder diameter on the diminution of scour hole around a cylinder and wake forces at the downstream from the cylinder. Comparison of numerical results with the experimental data available in the literature is satisfactory.

Keywords: open-channel flow, dynamic field, vertical cylinder, scour hole, horseshoe vortex, VOF, CFD.

1. Introduction

A one dimension flow which knocks itself on a vertical obstacle in a channel becomes suddenly a 3D flow. The flow model resulting around a cylinder becomes complex and it is difficult to evaluate the hydrodynamic. The rivers flows present a big variety of behaviors of their free surface. A flow is called “free surface flow” when part of the flow is opening on the atmosphere. The presence of another fluid called air, affects to its way the flow of water (frequently on surface). The variation of the bathymetry of water and the roughness of bed, that we meet so much in natural milieu (rubbles, alga, rock), as in hydraulic sewing (pile of bridge, canalization …), are often at the origin of this diversity.

The open-channel flow, in natural milieu or urban are produced in general with inhomogeneous boundaries conditions, because of the distribution of bed roughness and/or of important deformation of the free surface. When we place a vertical cylinder in these flows, hydraulics characteristics begin to vary poorly, then they emphasis up to generate superficial small vortex. This phenomenon becomes more and a big turbulence takes its maximal value either in surface, either near the bed: we then called « horseshoe vortex ». During the augmentation of the flow velocity, particles will be more and more detaching due to the formation of vortex which becomes more and stronger, forming a scour hole by increasing the height and deepness.

Many works have been done on the study of flow around a vertical cylinder, in order to understand and to well explain the mechanism of horseshoe vortex and the formation of scour hole around an obstacle. Lacey [1], has put in evidence a formula, but which was not taking into consideration very important parameters such as the height of water, the vertical cylinder, etc… Melville [2], observes the scour hole in sites of bridge in New Zealand, and in 1977, he studies the characteristics of flow in the scour hole under pillar. Nakagawa et al. [3], studies the structure of turbulence in a free surface flow. Dey et al. [4], proposed a model which permits to understand the scour
hole in clean water around a cylindrical pile. Yulistiyanto et al. [5], study flow around a cylinder placed in a channel at fixed bed, followed by the study of the flow around a cylinder placed in the channel of erodible bed, effectuated by Istiarto [6]. These study enumerated up are for all, the experimental studies. Many others numerical studies have been done at the instars of Rahman et al. [7], who have modified the formula of Lacey, while introducing certain parameters such as the lateral width of the obstacle and the height of water. Gao et al. [8], did a numerical simulation around a vertical cylinder to show the effects of the turbulence in this zone.

Despite this many studies, the horseshoe vortex remain a phenomenon difficult to explain and its rule in the process of scour hole is not well know. In this present work, we are going to do a numerical study of dynamic field generated by a turbulent flow around a vertical circular cylinder placed in an open-channel. This study will represent the normal conditions of the flow imposed by Istiarto [6], in his experimental dispositive. These phenomenons are produced with the flow dynamic around a vertical circular cylinder. It is interesting to determine the dynamic interactions of vortex around the cylinder.

The numerical model constitutes an essential equipment to determine these interactions. To lead well this study, we are going to present the mathematical formulation and the computation procedure employed for calculating the velocity and pressure fields for different Reynolds numbers, and by using a model of bi-dimensional turbulence, isotropic and stationary.

2. Mathematical Formulation and Computation Procedure

2.1 Assumption and Calculation domain

Our work of simulation is done through a rectangular canal with a length of L = 29 m and width of B = 2.45 m; a vertical circular cylinder is placed in the middle of width at 11 m from the canal entrance (figure 1). This cylinder has a configuration of diameter D = 0.15 m and a height of 0.20 m. The ratio B/D is 13.6 > 8, representing the minima ratio. The dimensions of canal and cylinder are chosen in conformity with the works of Istiarto [6]. The fluid is the water which is injected with a diameter hydraulic of d1 = 0.18 m.

2.2 Governing Equations

The monophasic turbulent flow in the open-channel flow is described by a set of non-linear partial differential equations expressing the physical laws of conservation between the velocity and pressure at each point of the flow: the Navier-Stokes equations. To these equations, we add the equation of turbulent kinetic energy and that of its dissipation rate like proposed by Launder and Spalding [9]. Solving these equations will reveal features such as pressure and dynamic fields.

\[
\frac{\partial (\rho \bar{u}_i)}{\partial x_j} = 0 \quad (1)
\]

\[
\frac{\partial (\rho \bar{u}_i \bar{u}_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial (\rho \bar{u}_i \bar{u}_j)}{\partial x_j} + F_i
\]

\[
\frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial \bar{u}_k}{\partial x_k} \right) \right] \quad (2)
\]

The relation (1) is a continuity equation, and (2) is the equation of the quantity of movement conservation in which:

- the left term represents the convective transport;
- the first term on the right represents the forces due to the pressure;
- the second and third terms on the right represents the forces generated by the turbulence.
the last term on the right represents the forces of viscosities;

The means equations lead to the appearance of the double correlation terms of the velocities fluctuations. They come from the non-linearity of conservation equations. These terms are called Reynolds stress \( \rho \overline{u_i u_j} \), and translate the effect of turbulence on the evolution of the means movement, giving the equations systems open by introducing supplementary unknowns terms. The closure problem is resolved through the hypothesis of Boussinesq:

\[
-\rho \overline{u_i' u_j'} = \mu_i \left( \frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i} \right) - \frac{2}{3} (\rho k) \delta_{ij}
\]

We are going to use for our resolution, the model of turbulence k-\( \varepsilon \) standard proposed by Fluent [10], which is a model enough used:

\[
u_j = C_\mu \frac{k^2}{\varepsilon} \tag{4}
\]

To determine \( \nu_j \), we have to calculate the two variables \( k \) and \( \varepsilon \). The equations of kinetic energy turbulent and its dissipation rate give us the following relations below:

For the kinetic energy turbulent:

\[
\frac{\partial}{\partial x_j} \left( \frac{\overline{u_j} k}{\varepsilon} \right) = C_\mu \left( \frac{k^2}{\varepsilon} \frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i} \right) \frac{\partial \overline{u_j}}{\partial x_j} + \frac{\partial}{\partial x_j} \left( \frac{C_\varepsilon k^2}{\sigma_k \varepsilon} \right) - \varepsilon \tag{5}
\]

- the term at the left represent the variation of the kinetic energy turbulent;
- the first term at the right represents the production of kinetic energy turbulent;
- the second term at the right represents the diffusion;
- the last term at the right represents the dissipation.

The dissipation energy equation is given by the following relation:

\[
\frac{\partial}{\partial x_j} \left( \frac{\overline{u_j} \varepsilon}{\sigma_k} \right) = C_{\varepsilon_2} \frac{k^2}{\varepsilon} \frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial}{\partial x_j} \left( \frac{C_\varepsilon k^2}{\sigma_k \varepsilon} \right) - \frac{C_{\varepsilon_2} \varepsilon^2}{k} \tag{6}
\]

Where, \( C_\mu, C_{\varepsilon_1} \) and \( C_{\varepsilon_2} \) are empirical constants; \( \sigma_k \) and \( \sigma_\epsilon \) are respectively the turbulent Prandtl numbers relative to \( k \) and \( \epsilon \). The values of these constants proposed by Jones and Launder [11], are represented on table 1 below, already used by Tcheukam-Toko et al [12].

<table>
<thead>
<tr>
<th>( C_\mu )</th>
<th>( C_{\varepsilon_1} )</th>
<th>( C_{\varepsilon_2} )</th>
<th>( \sigma_k )</th>
<th>( \sigma_\epsilon )</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.09</td>
<td>1.44</td>
<td>1.2</td>
<td>1.3</td>
<td>1.0</td>
</tr>
</tbody>
</table>

2.3 Computation Procedure

The transition from physical domain to the numerical domain begins with the generating mesh geometry by a preprocessor. Then import this into a computational code for the iterative solution of equations to determine the values of variables on each node of the mesh. The segregated solution method was chosen for the resolution of turbulence model and governing equations. Governing equations were discretized with the control volume technique. For the convective and the diffusive terms, a second order upwind method was used while the SIMPLE (Semi Implicit Method for Pressure Linked Equations) procedure was introduced for the velocity-pressure coupled to a multiphase model (VOF) (Patankar [13]). The convergence of the numerical calculation is checked by examining the evolution of relative residuals in each governing equation for a convergence criterion of 0.001%. The stability of the iterative process was carried out by relaxation coefficients associated with the velocity, pressure, \( \kappa \), \( \varepsilon \) and \( \mu_t \). The Standard Wall-Functions were used to take into account the effects of friction near the wall. Three mesh distributions have been tested to ensure that the calculated results are grid independent. The height of roughness has been introduced as a term \( k_s \) in relation with the Strickler coefficient \( K_s \), calculated by the formula (7) below, according to Sinniger and Hager, [14].

\[
K_k k_s^{1/6} = 8.2 \sqrt{g} \tag{7}
\]

3. Result and Discussion

3.1 Generating mesh geometry

The figure 2 below represents the computational domain meshed with the code GAMBIT. The grid distribution is a set of quadrilateral cells (uniformly structured mesh). The calculations will use the software Fluent [10], The figure 2
bellow represents the mesh near the cylinder. This mesh is very uniformly fine near the obstacle where the velocity gradient is large. The grid distribution impacts the computation time and the number of iterations required for the solution converge. The choice of the mesh size of 95,464 cells is a good compromise and the results that will be presented later are those of this mesh size. The no-dimensional variables are:

\[ X = \frac{x}{d_1} \quad (8) \]
\[ Y = \frac{y}{d_1} \quad (9) \]
\[ U_x = \frac{u_x}{U_{\text{max}}} \quad (10) \]
\[ P = \frac{p}{\rho g H} \quad (11) \]

3.2 Velocity field

The figure 3 below represents the velocity field for different Reynolds numbers. We observe that the fluid viscous friction with the cylinder wall generates the velocity decreasing in the downstream from the cylinder when the Reynolds number is increasing. Near the lateral side of the cylinder, we observe a strong oscillatory flow representing two vortexes in each side. Despite the variation of Reynolds numbers, the velocity contours are almost similar. This shows that for any Reynolds numbers, the horseshoe vortex distance remind too small because the cylinder diameter is very smaller than the canal width. The velocity value becomes maxima in the lateral plane of cylinder (\( \alpha = 90^\circ \)). This value decreases when the flow go up to the downstream. In this zone, we observe a flow reverse at the free surface, which create the wake vortex generating the formation of scour hole. These results are similar to the experimental results of Istiarto [6] and Yulistiyanto et al. [5].

3.3 Velocity profiles

The figure 4 below represents the velocity profiles for different Reynolds numbers, at the position \( x = +5 \) m in downstream from the vertical cylinder. These profiles show that the longitudinal velocity gradient is increasing with the increasing of Reynolds number. This strong velocity gradient generates the instabilities which could allow the formation of scour holes around a vertical cylinder.

3.4 Pressure field

The figure 5 below represents the pressure field for different Reynolds numbers. The pressure in upstream of the cylinder is increasing with the Reynolds number while the velocity is decreasing. This pressure is very high near the bed and there is vortex formation generating by the flow obstruction created by the vertical cylinder. Around a cylinder, a negative pressure gradient generates a flow reverse showing an oscillatory flow heading for the downstream. This oscillatory flow is called “horseshoe vortex”. The studies of Shen et al. [15], and Ahmed & Rajaratnam [16], show the similar results.

3.5 Pressure profiles

The figure 6 bellow represents the difference of pressure profiles at the upstream (position \( x = -5 \) m), and downstream (position \( x = +5 \) m), from the cylinder. At the position (\( x = -5 \) cm), we observe that the pressure increases with the Reynolds number. This relates the stagnation effect due to the presence of obstacle. At the position (\( x = +5 \) cm), the pressure gradient is negative showing a depression due to the flow behavior. We observe that the pressure decreases with the augmentation of the Reynolds number. The figure 7 bellow represents the difference of pressure profiles for many others position in the channel for \( Re = 81,000 \) in upstream and downstream from the cylinder. In upstream, the pressure increases up to his maximum value at the cylinder position. In downstream, the pressure shows a periodic profile justifying that it’s a wake zone.
Figure 3: Velocity field for different Reynolds numbers
Figure 4: Velocity profiles at 5 m in downstream from the vertical cylinder

a): Re = 18,000
b): Re = 54,400
c): Re = 81,000
d): Re = 151,200
Figure 5: Pressure field for different Reynolds numbers

Figure 6: Difference of pressure profiles; a): in the upstream from the cylinder (x = -5 m); b) in the downstream from the cylinder (x = +5 m)

Figure 7: Difference of pressure profiles for many others positions for Re = 81,000. a): at upstream from the cylinder positions; b): at downstream from the cylinder positions
4. Comparisons with Experimental Results

The figure 8 below represents the mean velocity profile around the circular cylinder compared to the experimental profile of Gao et al. [8]. The figures 9 below represent the velocity fields around the cylinder of this present study (b), compared to the experimental results (a), of Isiarto [6]. We observe that our results are in good concordance with the experimental results.

5. Conclusions

The study of velocity field has permitted to show around the cylinder a zone of recirculation amplified by an augmentation of Reynolds numbers. In these zones, there is a separation of the boundary layer and apparition of vortexes structures in the downstream from the cylinder. The velocity profiles show a negative gradient which confirm this hypothesis. The pressure field shows a negative gradient which created the depression at the sides and in the downstream from the cylinder. We have noticed that this depression generates a descending flow around of cylinder and an oscillatory flow in downstream. Our results are in good concordance with the experimental works available in the literature. In general rule, the good choice of the diameter of the cylinder is primordial for the minimization of the local scour hole around this.

Figure 8 Mean velocity profiles experiments (blue color), and numerical (Red color)
Figure 9: Velocity fields for Re = 81,000 and Fr = 0.34; a): Isiarto [6]; b): present study

Nomenclature

- $d_1$: hydraulic diameter (m)
- $D$: cylinder diameter (m)
- $x$: axial coordinate (m)
- $y$: vertical coordinate (m)
- $L$: channel length (m)
- $v$: vertical velocity (m/s)
- $u$: longitudinal velocity (m/s)
- $\bar{u}$: mean of longitudinal velocity (m/s)
- $u'$: longitudinal velocity fluct. (m/s)
- $g$: gravitational acceleration (m. s$^{-2}$)
- $h_1, h_2$: height of hydraulic jump (m)
- $p$: pressure (N/m$^2$)
- $Fr$: Froude number
- $Re$: Reynolds number = $V d_1/\nu$

Greek Symbols

- $\nu$: kinematic viscosity (m. Kg$^{-1}$. s$^{-1}$)
- $\rho$: density (kg. m$^{-3}$)
- $\mu$: dynamic viscosity (m. Kg$^{-1}$. s$^{-1}$)
- $\kappa$: turbulent kinetic energy (m$^2$. s$^{-2}$)
- $\varepsilon$: dissipation rate (m$^3$. s$^{-3}$)
- $\delta_{ij}$: kronecker symbol
- $\nu_t$: turbulent viscosity (m$^2$/s)
- $\tau$: shear stress

References


