3ème conférence Internationale des énergies renouvelables CIER-2015
Proceedings of Engineering and Technology - PET

Three-dimensional study of parallel shear flow with variable density around an obstacle

TOUMI MOHAMED
Unité de Métrologie et des Systèmes Énergétiques, ENIM
toumimohamed80@yahoo.fr

ABSTRACT

Environmental Assessment of the coastal fringe area degraded by pollution discharges and the wide use of obstacles in industrial and energy applications conducts us to study the hydrodynamic phenomena that are formed when a shear flow encounter a fixed obstacle at the bottom of the water channel or a wind tunnel. The purpose of this study is to show the change of the physical parameters of the flow in parallel with the change of the velocity inlet; and the influence of an obstacle placed at the bottom of a channel. The physical presentation of the problem and the adopted mathematical model were showed at the beginning of this work. The governing equations based on K-ε model are determined by the finite volume method with SIMPLEC algorithm giving the closest of experimental work results (Vinçont-2000) and digital work results (Rossi, 2009). This work conducts us to focus on the horizontal and vertical velocity fields, the Reynolds stress field, turbulence intensity and scalar concentration. The obtained results show that the influence of the presence of an obstacle is observed at 4h and 6h of the X-axis, the developed numerical code developed has allowed us to establish the dynamic characteristics of shear flows at zero velocity and superior values than the flow velocity which has an ordinate of 3h above the obstacle in a water channel and a wind tunnel.

Keywords: *shear flow, obstacle, water channel, wind tunnel, finite volume, K-ε model*

1. INTRODUCTION

Shear flow develops in most of the flows that govern our environment such as rivers, the ocean or atmosphere. It also reveals a large number of industrial flows (aeronautical, energy, water) as an important parameter. Therefore it is not surprising that many works were devoted to the study and prediction of shear flows. To study the properties of these flows, the ways of simulation and numerical modeling have been very successful. Numerical simulation appeared since the 1980s as a very effective methodology and increase in available computing resources has achieved numerical simulations of realistic shear flows. For several years, the understanding of the physical processes that alter the shear flow has been improved and many studies have been conducted to develop, from appropriate perceptions, numerical models of shear flows. There are generally two classes of models:

- Phase resolution models are based on equations that show, in an integral way, the instantaneous movements of the fluid whatever the depth (Eq From Boussinesq, 1872. Eq.from Korteweg & de Vries, 1895). The major constraint of this model is the importance of required computation time, which limited the number of study.

- The averaged phase models are usually used in oceanic and continental seas. Stochastic models solve the energy propagation equation and allow considering the generation phenomena, dissipation and energy transfer present in deep water (WAMDI Group, 1988; Komen et al. 1994). At the present time, these models are common in coastal area by taking into account the terms associated with shallow water propagation (bathymetric wave breaking, nonlinear interactions between frequency triplets). Then, understanding the shear flows is so important because it regulates the hydrodynamics of the site. The resolution of hydrodynamics is based on the three-dimensional Navier-Stokes meaning on the mass and momentum conservation equations. From these equations, some hydrodynamic mode can be classified according to the size of the studied phenomenon (Van Rijn, 1989). In fact, the achieved work during the past four years has as essential purposes to:

- Evaluate distinct numerical models of sheared and hydrodynamic flows by estimating the errors at every step.

- Improve the state of knowledge of the site by characterizing shear flows induced by the significant hydrodynamic effects.

The aim of our work is to study through a numerical simulation the dynamic profile of a shear flow around a cubic obstacle installed at the bottom of a water channel or an air tunnel. Essentially, we want to show the influence of inlet velocities on the perturbation of the flow in the channel. Practically, we want to follow numerically the change on the profile of the horizontal and vertical velocity, as well as the Reynolds stress field, turbulence intensity and scalar concentration.

2. ANALYSIS

This part aims to briefly explain the phenomenology and theoretical approaches of a shear flow, and also the various physical models, mathematical and numerical techniques adopted to simulate this kind of flow. We place ourselves in the context of incompressible flows for which the velocity field has a zero divergence. While it is impossible to characterize the shear flow with a strictly analytical point of view, it is possible to identify its principal properties of turbulent regime. So viscous incompressible flow of two continuous fluids, for example, air and water, are governed by mass conservation (continuity equation) and momentum conservation (Navier –Stokes equation) formulating classical conservation laws. To model the development of shear flows with variable density around an obstacle at the bottom of a channel, the finite volume method is adopted; but before that, you have to show the physical presentations of the problem surrounded by the boundary conditions for all the cases that we have.

.

2.1 Physical presentation problem

The physical presentation of the proposed problem is represented by the figures (1,2) .It describes a cuboids' channel with a length L = 256 cm, width l= 306 mm for the water and 500 mm for the air, height H = 170 mm for water and 500 mm for the air, and a square section of barrier installed at the bottom of a channel at h = 7mm for water and 10mm for the air. The distance between the channel inlet and the first surface of the barrier is 130h, the distance between the last surface of the barrier and the channel output is 234.7h. The geometrical dimensions are those used by Vinçont 2000 and Rossi 2009. The experimental measurements are carried out in the water channel with a Reynolds number Reh = 700 and 1500 to the air tunnel. The thickness of the boundary layer, without the obstacle positioned at the bottom of channel is about 5 cm, so that $\delta / h = 7$; This factor is the parameter that shows the length of the recirculation zone after the barrier where there is a large area of separation with the flow directed toward the opposite wall. The Reynolds number of the boundary layer was $R_\theta \equiv U_e\, \theta / \vartheta = 560$ for water and 980 for the air, where the thickness of momentum without obstacle θ = 5, 3 mm for water and 6, 3mm for the air, ϑ is the dynamic viscosity ϑ=1.005 $10^{-6}\ m^2 s^{-1}$ for water and 1.46 $10^{-5} m^2 s^{-1}$ for the air.

For the water flow, the little time interval Δt = 7.23 ms between the pair of lasers can be changed according to the average velocity of the flow in the region where the PIV pictures (Particle Image velocimetry) were obtained. The instantaneous velocity components U and V in the illuminated plane are determined by digitally cross-correlating the iriodine particle images in each subarea. The distance between the centers of each sub-area of the cross-correlation and the average of particles in the sub-region has been displaced in the time interval Δt Knowing these distances allows the calculation of the straight and upright velocities. The passive scalar which has infiltrated into the boundary layer of the water was rhodamine B, a fluorescent dye with a peak wavelength emission of 575 nm and a Schmidt number, Sc, about 2500. The physical principle on which is based the measurement of the scalar concentration is LIF (laser-induced fluorescence), which is based on the ability of the fluorescent dye to absorb the incident light at a wavelength relative to the concentration of the fluorescent dye at the measurement point (J Sakakibara et al 1999).

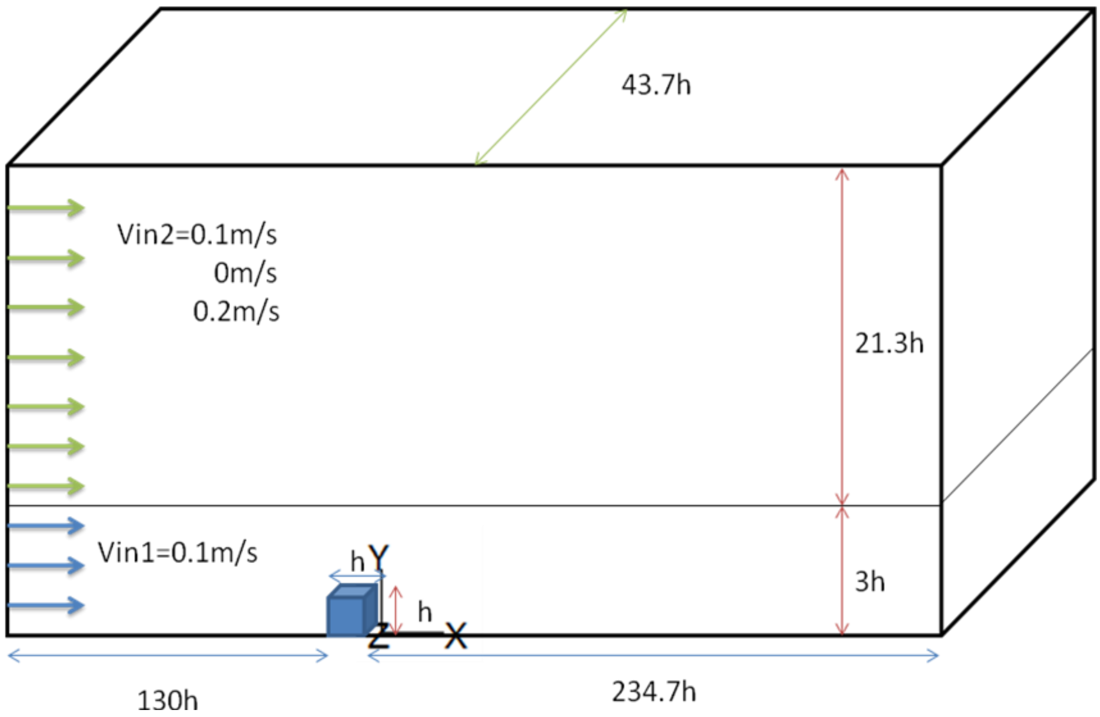

Figure 1: Physical model of the water channel

For the airflow, passive scalar that is infiltrated into the boundary layer was the smoke from the incense dry layer with a particle diameter about 0,1- 3μm, an average of

particle diameter is about 0,9 μm and a standard deviation of 0,5 μm (Chiang K.-F. 1998, personal communication). Smoke particles in this size similarly follow the flow; therefore the particles can be used for PIV measurements. The average of particle response time to a shift velocity is about 5 x 10^{-7}s; the peak frequency in this turbulent flow determined from the velocity spectra for a period about 10^{-3}s. The Schmidt number for this kind of smoke is very high in the order of 10^{6}. As the experience of the water flow, the second CCD camera, in front of the first one, has been extended so that the smoke particles could be viewed individually by PIV. The little time interval Δt between the pair of lasers has been changed to the air flow experiment from 0.4 ms close to the bottom to 0.225 ms in the flow area above the obstacle. The physical principle on which is based the measurement of scalar flow is MSD (Mie Scattering Diffusion) suggesting conditions where the size of diffusion particle is comparable to the wavelength of light (Charlson RJ et al 1999).
The simulated values are the horizontal velocity Ui, vertical velocity Vi, horizontal turbulence intensity Urms, vertical turbulence intensity Vrms, the Reynolds shear stress $\overline{uv}$ and mean concentration. For details about measurement techniques the reference is the work of Vinçont et al. (2000).

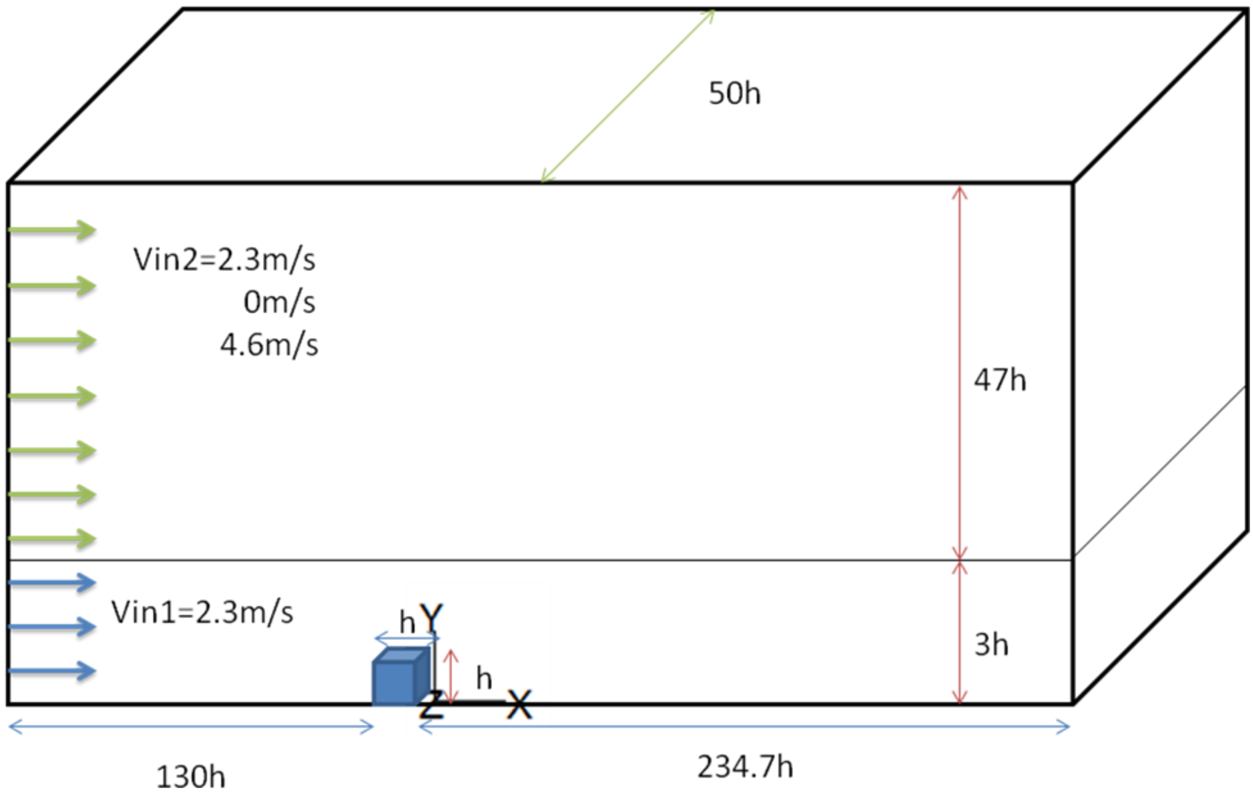

Figure 2: Physical model of the wind tunnel