J. Wind Eng. Ind. Aerodyn. 121 (2013) 16–28

Contents lists available at ScienceDirect

Journal of Wind Engineering and Industrial Aerodynamics

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

Suppression of vortex shedding on a bluff body

Alex Mendonça Bimbato [a,*], Luiz Antonio Alcântara Pereira [a], Miguel Hiroo Hirata [b]

[a] Institute of Mechanical Engineering, Federal University of Itajuba, Itajuba, CP 50, Minas Gerais, Brazil
[b] State University of Rio de Janeiro, FAT-UERJ, Resende, Rio de Janeiro, Brazil

ARTICLE INFO

Article history:
Received 13 January 2013
Received in revised form
27 June 2013
Accepted 30 June 2013

Keywords:
Vorticity transport
Strouhal number
Blockage effect
Vortex shedding suppression
Lagrangian Vortex Method with LES modeling

ABSTRACT

Previous experimental observations suggest that a moving ground can be numerically simulated avoiding vorticity generation on it. In this research line the current paper presents a two-dimensional Lagrangian Vortex Method, which is able to analyze the vortex shedding suppression on a circular cylinder near a moving ground in a high Reynolds number flow of $Re=1.0\times10^5$. The main purpose is to utilize numerical results to construct the instantaneous velocity and pressure fields for this kind of unsteady viscous flow. The analyses demonstrate that the physical mechanisms involved are associated with an additional circulation caused by the fluid viscosity and with the Venturi effect. Both of these effects are responsible for the increase in lift force as well as for the decrease in drag force acting on body surface.

1. Introduction

Bluff bodies are present in many engineering problems in all range of applications, starting from the smallest to the biggest scales. The cooling of electronic components is a good example of the small scale application. In the intermediate scales it can be mentioned that the aerodynamic loads acting on vehicles, cables and towers of power transmission lines and the flow over heat exchanger tubes. And, as examples of large scale applications, there are a lot of flows around offshore structures and large buildings.

The flow around bluff bodies is associated to interesting fluid dynamics phenomena such as separation, vortex shedding and turbulence transition; these phenomena arouse many questions of scientific interest, have great impact on engineering applications and are the starting point for the development of instabilities, which arise due to the interaction among three regions of the flow: the boundary layer, the two shear layers created at the separation points and the wake that develops downstream of the body. The rate at which vortices are cyclically shed from the body depends essentially on the interaction between the two shear layers, before mentioned, as function of the oncoming velocity, U, and the body diameter, d. The non-dimensional Strouhal number is the parameter used to measure the vortex shedding frequency and it is defined as

$$St=\frac{fd}{U}, \qquad (1)$$

where f is the vortex structures shedding frequency. Bearman (1984) stated that it is the presence of two shear layers, rather than the bluff body itself, that is primarily responsible for vortex shedding frequency.

A particular interesting situation occurs when a circular cylinder is close to a plane boundary; it can be observed that as the cylinder comes close to the ground for gap ratios (gap, h, between the bottom of the cylinder and the wall to the diameter, d, of the cylinder) around $h/d\leq0.2$ the vortex shedding frequency decreases until its suppression. According to Zdravkovich (1981) "the complexity of the phenomena involved is reflected in a distinct variation and almost limitless modification of the flow pattern is affected by the Reynolds number, turbulence level, surface roughness, three-dimensionality, and elastic response".

The present study is motivated not only by academic purpose but because this phenomenon is desired in many practical situations. Some particular motivating applications, where are identified the ground effect mechanisms, are aerodynamic designs of Wing-In-Ground effect Vehicles (WIGV), bluff bodies in close proximity to the ground and submarine cables of offshore industry subject to Vortex Induced Vibrations (Wooton et al., 1972).

Taneda (1965) studied the influence of the distance between the cylinder and the moving ground at $Re=170$ using a water tunnel. In these tests the ground had the same speed as the water flow; a decrease in the vortex shedding frequency was observed when the gap ratio was less than 0.1.

Bearman and Zdravkovich (1978) performed velocity measurements to study the vortex shedding frequency in a high Reynolds number flow of $Re=4.8\times10^4$when the cylinder is close to a fixed ground. The authors verified a constant value for the Strouhal number ($St\cong0.2$) for any value of the gap ratio $h/d<0.3$.

* Corresponding author. Tel.: +5535 3629 1299; fax: +5535 3629 1148.
E-mail address: alexbimbato@yahoo.com.br (A. Mendonça Bimbato).

0167-6105/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.jweia.2013.06.013

Unlike the results of Bearman and Zdravkovich (1978), Buresti and Lanciotti (1979) measured the Strouhal number when the cylinder is near to a fixed ground at $Re=1.9\times10^5$ and found a critical gap ratio of $h/d_c=0.4$ and a Strouhal number about 0.2 when $h/d>0.4$. It was concluded that the critical gap ratio and the Strouhal number depend on the flow regime and it seems to be impossible to define exact values for these variables. However, it can be stated that for high Reynolds number flows the Strouhal number decreases as the gap ratio decreases.

As shown by Lei et al. (1999), it is difficult to accurately determine the critical gap ratio, because experiments and numerical simulations are carried out using discrete gap ratios and the vortex shedding suppression occurs as the gap ratio is gradually reduced.

Zdravkovich (2003) reported the drag behavior for a cylinder placed near a moving ground running at the same speed as the freestream for higher Reynolds number of 2.5×10^5. The experiments showed some differences to all the previous studies. First, practically no boundary layer was developed on the ground. Second, the decrease in drag due to the decrease in h/d did not occur in the measurements. The differences encountered were attributed to the non-existence of the wall boundary layer or the flow regime at higher Reynolds number.

Experiments from Nishino (2007) showed drag and lift coefficients behavior acting on a cylinder surface placed near a moving ground at high Reynolds number flows of $Re=4.0\times10^4$ and $Re=1.0\times10^5$, respectively. The ground moved at the same speed as the air flow and it was observed that practically no boundary layer was developed on the ground surface; besides, three distinct ranges were found for the gap ratio: (a) if $h/d>0.50$, larger vortex structures are generated at the rear part of the cylinder; (b) if $0.35<h/d<0.50$ the vortex shedding is intermittent; (c) if $h/d<0.35$ the vortex shedding is suppressed. Nishino (2007) also studied the end effects on the aerodynamic loads behavior using end plates which is very important to compare two-dimensional and three-dimensional flow regimes.

It can be observed in the literature a lack of numerical works that deals with vortex shedding suppression. A few studies can be found and they practically are restricted to low Reynolds number flows using Eulerian methods in ranges of $80\le Re\le1000$ and $200\le Re\le600$ (e.g., Lei et al. (2000) and Huang and Sung (2007)).

Recently, Bimbato et al. (2011) presented numerical evaluations of the complex effects of the two-dimensional viscous flow around a cylinder in the vicinity of a moving ground. The numerical strategy involved the combination of Lagrangian Vortex Method with source Panels Method to calculate global as well as local quantities in a high Reynolds number flow of $Re=1.0\times10^5$. The main contribution of authors was to use a moving ground to present physical issues related to the flow. The purpose of using a moving ground is to eliminate the influence of the boundary layer developed on the ground, which is a crucial factor in the analysis. The numerical results were in reasonable agreement with limited amount of experimental data. However, physical explanations concerning the destruction of the vortex street and drag force reduction, especially for small gap ratios were not present. These phenomena are very difficult to capture using experimental measurements.

The present paper, therefore, contributes by using a Lagrangian Vortex Method in order to produce physical explanations concerning to the suppression of vortex shedding in moving ground effect for very small gap ratios. The numerical simulations consider two-dimensional high Reynolds number flows and include turbulence modeling. With the present methodology it is still possible to identify important aspects of ground effect and the results are useful for the accumulation of knowledge about moving ground mechanisms and vortex shedding control.

Discrete Vortex Method is one of the numerical methods used for computer simulation of turbulent fluid flows. For 25 years Discrete Vortex Method has been successfully used on wind engineering and industrial aerodynamic problems (e.g., Bienkiewicz and Kutz (1990); Turkiyyah et al. (1995); Alcântara Pereira et al. (2004); Rasmussen et al. (2010)).

In unsteady flows, the vorticity-containing regions move around and deform with time, and it is much more difficult to resolve numerically the ever-changing vorticity field using a fixed grid. On the other hand, this work discretizes the vorticity field present in the fluid domain using discrete vortices, which are followed individually throughout the numerical simulation. Failure to resolve vorticity with Eulerian methods gives rise to significant numerical dissipation, which results primarily from truncation errors associated with discretization of the nonlinear convective term. Lagrangian Vortex Method is efficient and highly self-adaptive, since discrete vortices are convected with the same speed induced on fluid. As consequence, the governing equations are solved only where vorticity is present; thus the far away boundary condition is automatically satisfied. These facts make the Lagrangian Vortex Method well suited for the analysis of complex, unsteady and vortical flows; some of the works using Discrete Vortex Method are Chorin (1973), Leonard (1980), Sarpkaya (1989), Lewis (1991), Kamemoto (2004) and Kathir and Lucey (2012).

2. Problem statement

Consider the two-dimensional, incompressible and unsteady viscous flow around a circular cylinder placed near a ground plane whose movement is represented by the absence of vorticity generation (Bimbato et al., 2011). Fig. 1 schematically represents this situation, U being the incident flow velocity, d the cylinder diameter, h the distance between the bottom of the cylinder and the ground plane and Ω represents the fluid domain defined by the surface $S=S_1\cup S_2\cup S_3$ such that S_1 is the body surface, S_2 is the ground surface and S_3 is the far away boundary.

This flow, depicted in Fig. 1, is governed by the continuity and Navier-Stokes equations, which can be written in the form

$$\frac{\partial u_i}{\partial x_i}=0, \tag{2}$$

$$\frac{\partial u_i}{\partial t}+\frac{\partial}{\partial x_j}(u_iu_j)=-\frac{1}{\rho}\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\nu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\right], \tag{3}$$

where u_i is the i–th velocity component, ρ is the density, p is the pressure field and ν is the fluid kinematic viscosity.

A LES modeling is used to separate the large eddies (to be numerically solved) from the small eddies (which require a turbulence model to be analyzed) by filtering the governing Eqs. (2) and (3). Consequently, these equations become

$$\frac{\partial \bar{u}_i}{\partial x_i}=0, \tag{4}$$

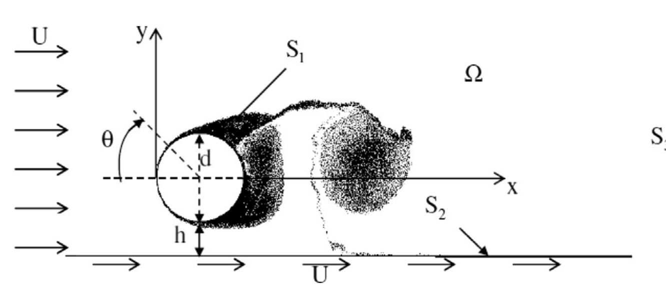

Fig. 1. Definition of fluid domain.