Computers & Fluids 88 (2013) 272–286

Contents lists available at ScienceDirect

Computers & Fluids

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

Numerical simulation of flow past an underwater energy storage balloon

A.R. Vasel-Be-Hagh *, R. Carriveau, D.S.-K. Ting

Turbulence and Energy Laboratory, Lumley Centre for Engineering Innovation, University of Windsor, Windsor, Ontario N9B 3P4, Canada

ARTICLE INFO

Article history:
Received 10 January 2013
Received in revised form 5 July 2013
Accepted 13 September 2013
Available online 25 September 2013

Keywords:
Underwater balloon
Droplet-shaped bluff body
Vortex tube
Swirling tube flow
RANS k–ω
LES Dyna-SM

ABSTRACT

A three-dimensional simulation was conducted to investigate water flow over the accumulator unit of an underwater compressed air energy storage system. The accumulator unit, which is a droplet shaped balloon, was installed close to the bed of deep water. The simulation was carried out at a free stream Reynolds number of 2.3×10^5 using URANS k–ω and LES Dyna-SM turbulence models. The URANS model failed to capture the turbulent nature of the flow; however, its predictions of the mean values were in reasonable agreement with those of the LES model. The time-averaged force coefficients of the balloon were found to be greater compared to the literature for spherical bluff bodies and smaller than those of circular cylinders. The structure of the flow was closely investigated using isosurfaces of the second invariant of the velocity gradient and three-dimensional path lines. Several shedding vortex tubes were identified downstream of the balloon. The dynamics of these vortex tubes was further illustrated through time series snapshots containing vorticity lines on two-dimensional planes perpendicular to the flow direction. The frequency of the shedding and the turbulent movements of the vortex tubes were studied through power spectrum analysis of the force coefficients.

1. Introduction

The energy industry continues to advance with clean electricity generation through the development of renewable energy technologies. However, it is still dealing with the challenge of non-constant supply from these characteristically intermittent generators. Great variation in the power generated by wind turbines from windy to calm days and variation in output from photovoltaics between sunny and cloudy days limits the potential for base load penetration of these generators. Energy storage promises great potential as a way of managing timely distribution of these resources in the most efficient way. Viable energy storage solutions could enable these resources to become dispatchable and suitable for base load service.

Such a potential solution is currently being studied by the authors, namely underwater compressed air energy storage (UW-CAES) [1]. In this system, flexible accumulators are installed close to the bed of a deep water source, i.e. a lake or ocean. Surplus electrical energy is used to compress air into underwater accumulators. Hence the stored air is under the hydrostatic pressure applied by the water, ready to be released to drive turbo expander generators to deliver power to the grid when desired. The Professional (PF) series of underwater lift balloon produced by SUBSALVE USA CORPORATION [2] have served as a suitable choice to perform as the accumulator unit of the UW-CAES system. The generally droplet-shaped series of different sized PF lift balloons are presented in Fig. 1.

There has been significant progress in the understanding of fluid structure interactions over the last couple of decades. The focus, however, has been limited to flow over circular cylinders [3–17], square cylinders [18–22], rectangular cylinders [23–26] and elliptical cylinders [27]; these are primarily two-dimensional studies. There are also several three-dimensional studies but they are mostly concerned with the flow over regular shapes such as spheres [28–33], circular cylinders [34–38] and square cylinders [39].

Our review of flow over bluff bodies indicated a dearth of published research that explored the cross-flow around a droplet-shaped bluff body like underwater balloons. Hence, the current numerical study was carried out to examine flow over an underwater balloon, specifically the PF20000 model. This research was driven by a need to provide insight into the potential hydrodynamic behavior of the submerged UW-CAES accumulators. Subsequently, the authors were asked to investigate the force characteristics and the structure of the flow. Due to the absence of any experimental evidence, the simulation was carried out using both URANS and LES turbulence models for the sake of results comparison. In addition, mesh and domain independence analyses were done for both of the turbulence models separately.

2. Computational details and boundary conditions

In the present paper, the characteristic diameter used to calculate the Reynolds number was defined as

* Corresponding author.
E-mail address: vaselb@uwindsor.ca (A.R. Vasel-Be-Hagh).

0045-7930/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.compfluid.2013.09.017

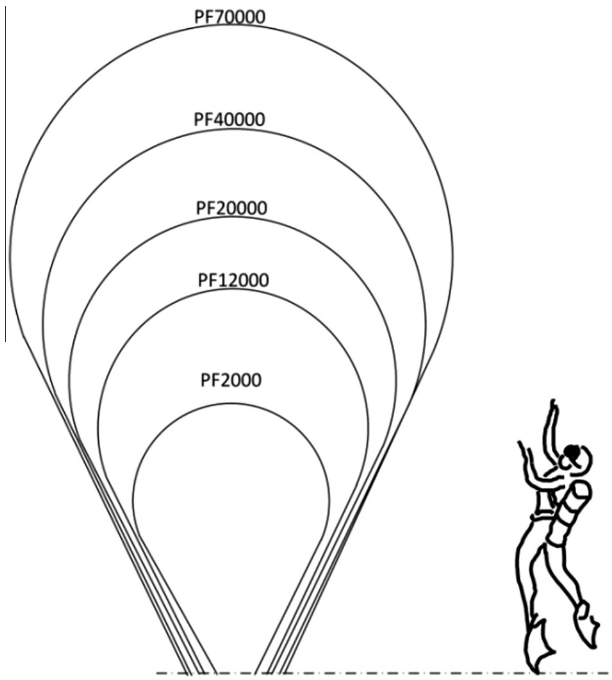

Fig. 1. General shape of various Professional (PF) series of lift bags produced by SUBSALVE USA CORP [2]. The PF20000 model was considered in the current study.

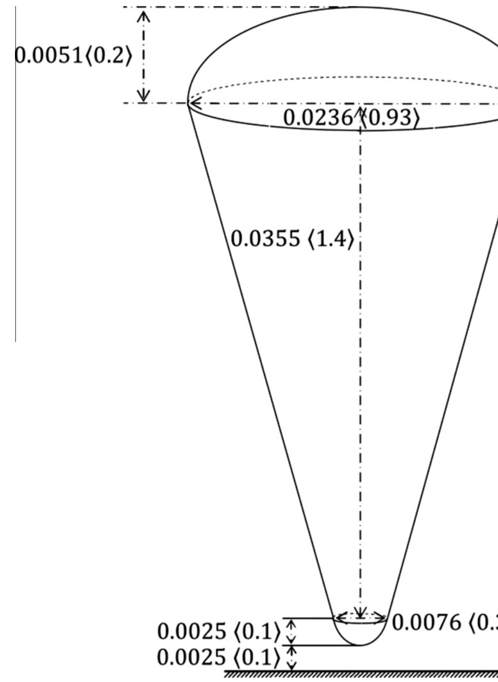

Fig. 2. Scaled PF20000 balloon. Values are in meters, and brackets ⟨⟩ are in inch.

$$D = 6 \times V/A \tag{1}$$

where V and A are volume and surface area respectively. According to Eq. (1), the characteristic diameter of the PF20000 balloon used as the accumulator unit of the UW-CAES is 2.31 m [40]. The Pilot Study of the UW-CAES was carried out in the Lake Ontario [1]. According to Centre for Operational Oceanographic Products and Services [41] the mean offshore current velocity of this lake is approximately $V_{mean} \approx 0.2–0.25$ Knot that is 0.1–0.13 m/s. By choosing $V = 0.115$ m/s, Reynolds number of the flow crossing the full scale balloon, $D = 2.31$ m, is $\mathrm{Re} = \rho VD/\mu \approx (998) \times (0.115) \times (2.31)/1.15 \times 10^{-3} \approx 2.3 \times 10^5$ where $\mu = 1.15 \times 10^{-3}$ is the dynamic viscosity of water at mean temperature of the lake water which, according to National Weather Service Forecast Office [42], was approximately 15 °C during 2011.

In order to decrease computational expenses, dimensions of the balloon were scaled down by a factor of 100. As the bluff body force coefficients are functions of the Reynolds number, the free stream velocity was scaled up 100 times to retain the same Reynolds number. Therefore, the free stream velocity was set to be $V = 100 \times 0.115$ m/s = 11.5 m/s. Dimensions of the scaled balloon are presented in Fig. 2. Original dimensions of the PF20000 balloon are reported in inches on the SUBSALVE USA CORPORATION web site [40]; therefore, in Fig. 2 the scaled dimensions are presented in both meters and inches (inches in brackets) to ensure the decimal accuracy.

The dimensions of the computational domain are given in Fig. 3 in terms of the characteristic diameter. Blockage ratio BR of the balloon is the most determinative parameter in choosing the height and the width of the computational domain. Prasanth et al. [43,44] demonstrated that the effects of the blockage ratio on the interactions between flow and bluff bodies are more significant at low Reynolds numbers so that the blockage ratio should be 1% or less, whereas at large Reynolds numbers it does not affect significantly. For instance, they showed that at Reynolds numbers larger than 100 the characteristics of the flow crossing a circular cylinder are very close for blockage ratios of 1% and 5%. At a Reynolds number in the order of what is studied in the present paper, i.e. $\sim 10^5$, the blockage ratio is mostly chosen to be in the range of 3–25% (see Table 1). However, in the present paper, to ensure the negligibility of the walls effect the height and the width of the computational domain were chosen to be 11D and 13.2D respectively to end in a blockage ratio of 0.88% which is too much smaller than what is normally chosen for the similar cases (Table 1). The blockage ratio was calculated as $\mathrm{BR} = a_c/A_C$, where $a_c \approx 1.06\ \mathrm{in}^2$ and $A_c \approx 120.24\ \mathrm{in}^2$ are cross sectional areas of the balloon and the computational domain respectively.

Although the dependency of the results on the upstream distance L_u (the distance between the inlet boundary and the middle of the balloon) and downstream distance L_d (the distance between the middle of the balloon and the outlet boundary) is more significant at low Reynolds numbers [32,52,53], the present simulation was repeated for three different domain sizes to ascertain the independency of the results from upstream and downstream distances. Effects of the upstream and downstream distances on the mean force coefficients are shown in Table 2.

It is observed that the drag coefficient is more sensitive to the domain size compared to the lift coefficient. According to Table 2, upstream and downstream distances larger than 12D and 56D do not significantly affect the simulation results, so, the present paper was conducted for L_u = 12D and L_u = 56D. There is a distance of 0.1D between its lowest point and the solid bed. It should be mentioned that the origin of the coordinate system used in this study is fixed right underneath the balloon on the bed surface; accordingly the bottom of the balloon is at $(0, 0.254 \times 10^{-2}\ \mathrm{m}\ \langle 0.1\rangle, 0)$.

The boundary conditions that are employed in the current simulation are also depicted in Fig. 3. A mass-flow-inlet condition with a mass flow rate of 888.5 kg/s is set at the inlet boundary, resulting in an inlet velocity of 11.5 m/s to retain the Reynolds number at 2.3×10^5. An outflow condition with flow rate weighting of 1 is used at the outlet boundary, as it is the only outlet of the computational domain. In the real application the accumulator unit is installed in deep water, therefore, to be in accordance with the real case a free surface condition should be applied on the top boundary of the computational domain. To define the free surface boundary condition a two-phase model must be added to the simulation