J. Wind Eng. Ind. Aerodyn. 142 (2015) 173–187

Contents lists available at ScienceDirect

Journal of Wind Engineering and Industrial Aerodynamics

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

Computational simulation of the turbulent flow around a surface mounted rectangular prism

E.C. Joubert *, T.M. Harms, G. Venter

Department of Mechanical and Mechatronic Engineering, Stellenbosch University, South Africa

ARTICLE INFO

Article history:
Received 10 September 2014
Received in revised form
27 March 2015
Accepted 27 March 2015

Keywords:
Bluff body aerodynamics
Rectangular prism
Wind tunnel tests
Computational simulation
Detached eddy simulation
OpenFOAM

ABSTRACT

Engineering structures considered as bluff bodies are often associated with complex unsteady turbulent flow structures that are not yet fully understood and difficult to reproduce numerically. One of the main challenges is to find a numerical model that accurately account for turbulence in the flow without using too much computational resources. In this paper the Spalart–Allmaras improved delayed detached eddy simulation (IDDES) turbulence model with an all-y^+ wall treatment is used to numerically reproduce the flow features around a rectangular cross-sectioned beam in a wind tunnel at a Reynolds number of 7.6×10^4. The beam is orientated with the long edge of the cross-section parallel to the flow and can be characterised by the ratios of $L/D = 2.63$ and $H/L = 5$. The simulation results are compared to time-averaged experimental data which includes particle image velocimetry (PIV) and pressure. The comparison of the simulation results against experimental data shows that the Spalart–Allmaras IDDES model accurately reproduces the flow field around the beam with only minor discrepancies. Furthermore, the complex time-varying three-dimensional flow field is discussed in detail. Alternating vortex shedding occurs but this seems to be periodically interrupted. The Strouhal number over the height of the beam was found to vary between 0.053 and 0.059 which corresponds well with studies of similar L/D ratios in the literature. The maximum cyclic loading was found to occur at a height of $y = 0.5H$ and along the side wall at $0.8L$ from the upstream edge of the beam.

1. Introduction

Numerical modelling and measuring of bluff body aerodynamics has found significant interest in engineering sciences. The motivation for the vast interest in bluff body flows is due to its wide applicability in our daily life (for example, aerodynamics of bridges, buildings, antennas, vehicles and equipment) and the complexity of the fundamental physical phenomena related to these flows. Bluff body flows are especially challenging since they are highly three-dimensional and usually associated with a wide range of flow regimes such as stagnation, separation, recirculation, strong shear layers and unsteady vortex shedding. These flow features are difficult to capture with measurements techniques and even more challenging to model accurately.

A number of studies have investigated two and three-dimensional flows around infinite cylinders with circular, square and rectangular cross-sections (Williamson, 1996; Zdravkovich, 1997; Matsumoto, 1999; Wissink and Rodi, 2008; Lübcke et al., 2001; Wang et al., 2001; Lee, 1997; Bruno et al., 2010; Mannini et al., 2010) to better understand the related fluid dynamics and accuracy of various numerical modelling strategies. Although these studies shed light on the underlying causes of the flow phenomena as well as the strengths and limitations of available numerical models, finite height geometries (including wall mounted cubes), on the other hand, cannot be treated as infinite cylinders since they introduce additional three-dimensional flow phenomena due to the effects of the wall mounting and free tip which interacts with the wake dynamics typically understood to originate from the cross-section shape of infinite cylinders (Lim et al., 2009; Yakhot et al., 2006; Iaccarino et al., 2003; Rodi, 1997; Wang and Zhou, 2009; Wang et al., 2009; Becker et al., 2008; Uffinger et al., 2008, 2010, 2013; Kawai et al., 2009). One example is the study by Kawai et al. (2009), who conducted detailed PIV measurements of flow around a finite height building structure and elaborated on the complexity of the unsteady three-dimensional interaction of the side-wall vortex shedding and the arc vortex originating from the top of the cylinder.

Wang (2004) presented a conceptual schematic of the basic flow features that have been identified for finite height cylinders with square cross-sections (see Fig. 1). This highlights the three-dimensional effects that are involved and, although it is not the same

* Corresponding author.
E-mail addresses: ecjoubert@sun.ac.za (E.C. Joubert), tmh@sun.ac.za (T.M. Harms), gventer@sun.ac.za (G. Venter).

http://dx.doi.org/10.1016/j.jweia.2015.03.019
0167-6105/

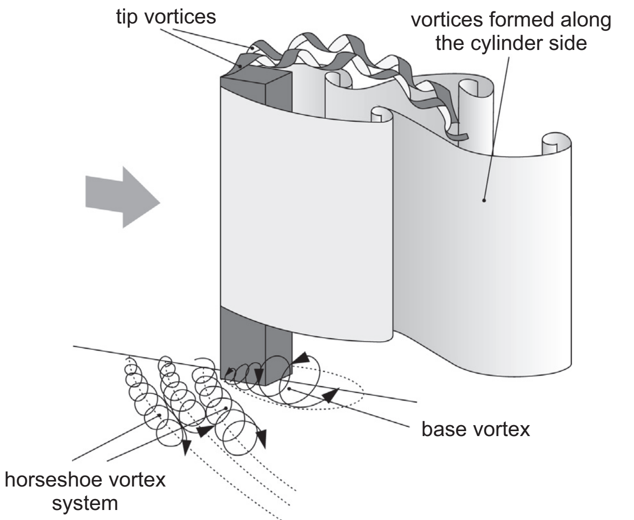

Fig. 1. Flow patterns characteristic of a finite height, square cross-section cylinder (Wang, 2004; Uffinger et al., 2013).

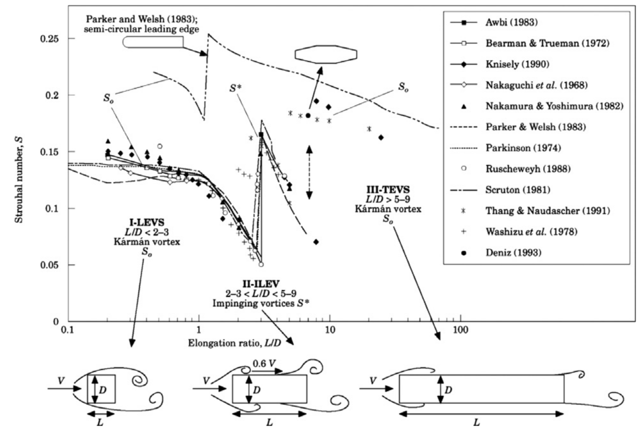

Fig. 2. Vortex shedding frequency for rectangular cross-sections with different L/D ratios (Morgenthal, 2000; Deniz and Staubli, 1997).

geometry as the beam in this study, it does provide a good overview of the general flow features to be expected and discussed. The flow features around the geometry include the horseshoe vortex upstream of the cylinder, the base vortex, side-wall vertical vortices and tip vortices.

The horseshoe vortex upstream of the flow seems to have little effect on the rest of the flow (Sau et al., 2012; Bourgeois et al., 2011). The base vortex originates from the upstream tip of the cylinder side edge at the base and is characterised by an upward flow at the cylinder side-wall. At mid-height, with little influence from the base and tip, vortices originate from the side-wall upstream edges. These side-wall vortices are closely related to the observed flow phenomena of infinite cylinders. The existence of tip vortices were proposed by a number of investigations (Etzold and Fiedler, 1976; Kawamura et al., 1984; Sumner et al., 2004) as well as the existence of a overarching vortex structure connecting the side-wall vortices (Sakamoto and Arie, 1983; Wang and Zhou, 2009). Investigations of the phase shift of vortices over the height of the cylinder further suggest that bent shape vortex structure (reaching further downstream at the mid-height than at the top and base) exists in the wake region.

When considering cylinders with rectangular cross-sections it is also important to consider the effect of a L/D ratio on the vortex shedding phenomena. Deniz and Staubli (1997) presented the findings of various authors that relate the effect of the L/D ratio of a rectangular cylinder to the vortex shedding frequency (represented by the Strouhal number). This relation is depicted in Fig. 2. For $L/D < 2$–3, such as for the widely studied square cylinder, the afterbody (length of geometry after separation point) is short and the upstream edge vortex shedding dominates without impinging on the side-wall. The next regime is for $2\text{–}3 < L/D < 5\text{–}9$ where upstream edge vortex shedding impinges on the side-wall but without reattachment taking place. Lastly, for $L/D > 5$–9 the upstream edge vortices attach to the side-wall and trailing edge vortex shedding dominates.

Numerical simulation methods, and especially turbulence modelling, for accurately reproducing the flow features around bluff bodies continue to be a challenge. This is due to the complexity of turbulence, the wide range of turbulence scales and the different effects that these scales have on the mean flow. This means that turbulence models are usually either very flow regime specific, resulting in inaccuracies where the model is not applicable, or computationally expensive, leading to long simulation times.

RANS models, although widely used for simplicity and robustness (Kim and Boysan, 1999; Paterson and Apelt, 1990), for example, are well suited for attached boundary layer flows but struggle in separated regions and stagnation zones (Kim and Boysan, 1999; Li et al., 1998; Rodi, 1997; Lee, 1997; Mannini et al., 2010; Huang et al., 2007) and are, therefore, not considered appropriate for this investigation. On the other hand, large eddy simulation (LES) models are better suited than RANS models for resolving complex turbulence induced flow fields but require large computational resources due to the fine mesh demands in the near-wall region where eddy structures resolved by the LES reduce significantly in size (Murakami et al., 1992; Lübcke et al., 2001; Rodi, 1997; Huang et al., 2007). For example, for flow around a circular cylinder Lübcke et al. (2001) reported that LES simulations took 20 times longer to solve compared to RANS models. Furthermore, Rodi (1997) reported that LES simulations took more than 36 times longer than RANS simulations for flow around a square cylinder and surface mounted cube.

In order to find a middle ground between the efficiency of RANS models and the accuracy of LES, detached eddy simulation (DES) was first introduced by Spalart et al. (1997) as a hybrid RANS/LES turbulence model for highly separated flows based on the Spalart–Allmaras RANS model (Spalart and Allmaras, 1992). DES combines the favourable aspects of RANS (efficient and accurate at calculating attached boundary layers) and LES (more accurate in highly separated flows). Some of the newer developments in DES include delayed DES (DDES) where the model is shielded against grid induced separation (Spalart et al., 2006) as well as improved DDES (IDDES) which incorporates wall modelled LES (WMLES) (Shur et al., 2008) making it applicable to a wide range of applications. IDDES can, therefore, provide the accuracy of LES for highly separated flow regions and computational efficiency of RANS in the near-wall region making it the turbulence model of choice for this application. For the interested reader a comprehensive background of DES is given by Mockett (2009).

The near-wall treatment in CFD usually requires special attention since the meshing of this region usually has a significant influence on the simulation time and the effect of wall shear stress strongly influences the bulk flow. Near the wall turbulence structures differ significantly from that of the far field. Turbulence eddies near the wall are small, isotropic, dissipating and characterised by dominating viscous stresses. In the far field, on the other hand, large eddies exist that are un-isotropic and