Journal of Fluids and Structures 63 (2016) 103–119

Contents lists available at ScienceDirect

Journal of Fluids and Structures

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

Detached eddy simulations of flow induced vibrations of circular cylinders at high Reynolds numbers

Vinh-Tan Nguyen *, Hoang Huy Nguyen

*Institute of High Performance Computing, Agency for Science, Technology and Research, Singapore (A*STAR), 1 Fusionopolis Way, #16-16 Connexis, Singapore 138632, Singapore*

ARTICLE INFO

Article history:
Received 15 August 2015
Received in revised form
7 January 2016
Accepted 16 February 2016

Keywords:
Flow induced vibrations
Vortex induced vibrations
Computational fluid dynamics
Detached eddy simulations
High Reynolds number flows
Fluid–structure interactions

ABSTRACT

This work presents a development of computational fluid dynamic model for numerical simulations of flow induced vibrations, important physical phenomena characterizing operations of various engineering structures, especially offshore installations including risers and platforms. We propose an application of detached eddy simulation (DES) approach for simulations of incompressible Navier–Stokes flows over cylindrical structures coupled with its rigid body motions. In this approach, a hybrid turbulence model based on Reynolds average Navier–Stokes (RANS) and large eddy simulations (LES) is implemented for modelling of high Reynolds number flows over free oscillating cylinders where near wall flow features are modelled with RANS and LES is employed for resolving wake dynamics. The proposed approach is validated against several experimental set ups of fixed cylinder as well as cylinders under vortex induced vibrations. A straightforward implementation of the proposed DES approach provides more reliable and accurate numerical results than traditional URANS for high Reynolds number flows in predictions of forces acting on cylinders and their responses. Numerical experiment using the presented approach for cylinder with single degree of freedom (cross flow oscillations) and two degree of freedoms (in-plane oscillations) shows that it is able to capture VIV fundamental characteristics and predict accurately responses of cylinders at various reduced velocities and mass ratios. Numerical prediction is in good agreement with earlier experimental studies for presented benchmark test cases. It shows a great potential of the proposed approach for design and analysis of structures under flow induced vibrations.

1. Introduction

Flow induced vibrations are well-known hydrodynamic phenomena in which strong vortices are shed in the wake of flows passing through bluff body structures such as columns, risers and mooring lines. These vortices change hydrodynamic loading on the structures, resulting in vibration of the structures which, in turn, affects the shedding of the vortices. This interaction between flows and structures is highly nonlinear and poses a great challenge for modelling and simulations. The flow induced vibrations of risers, in particular, are a topic of industrial importance, since these vibrations can destroy the riser due to strong shear forces or dramatically shorten the riser life span due to quicker onset of fatigue. Current design practices still rely heavily on model testing and empirical data. Also, design criteria for single risers are usually applied for riser array designs.

* Corresponding author.
E-mail address: nguyenvt@ihpc.a-star.edu.sg (V.-T. Nguyen).

http://dx.doi.org/10.1016/j.jfluidstructs.2016.02.004
0889-9746/

Recently, numerical simulations, in particular computational fluid dynamics (CFD), have emerged as an efficient tool with high accuracy for studying flow induced vibrations. There has been active development and good progress of CFD tools for simulations of vortex induced vibrations (VIV) for risers. However, the predictive capabilities of CFD simulations are still limited due to the complex nature of flow dynamics and nonlinear interactions between long risers and currents. Due to these limitations, empirical and analytical tools are still extensively employed together with experimental measurements in the design and deployment of risers. Yet, full scale measurements can only be done in the last stage of design, and the number of configurations is limited by the high cost and expensive setup involved. Improving accuracy and reliability of numerical simulations for predictions of flow induced vibrations is thus essential in establishing CFD as a complementary tool to physical experiment in practical design and analysis of offshore structures under VIV effects.

Most of the numerical work on VIV over the past few decades focuses on low Reynolds number flows $Re = 10–10^3$ in which fundamental fluid dynamics phenomena were illustrated and explored with greater level of details due to high fidelity, high resolution CFD simulations. However numerical simulations still require guidance and inspiration from physical experiment and measurements. Furthermore CFD is still much lacking behind in comparison with advances in measurements in terms of modelling real conditions of practical VIV in operations such as effects of high Reynolds number flow conditions, long riser configurations, surface roughness, suppression mechanisms. Limitations of numerical simulations were highlighted in earlier comprehensive reviews by Sarpkaya (2004) and Bearman (2011) with more information about earlier work as well as outstanding challenges. One of the main challenges of numerically simulating VIV phenomena is the resolution of flow features at high Reynolds numbers in the range of $10^4–10^6$, at which flows exhibit transitions to turbulent or fully turbulent behaviors. In this range of flow conditions, direct numerical simulations (DNS) become inhibited due to unrealistic computational demands even with the powerful computing infrastructure currently available. The importance of resolving flows past cylinders at high Reynolds numbers has led to the exploration of various advanced turbulent modelling techniques, from traditional industrial unsteady Reynolds-averaged Navier–Stokes (URANS) models (Rosetti et al., 2012; Stringer et al., 2014) to large eddy simulations (LES) (Koobus and Farhat, 2004; Feymark et al., 2012) and hybrid detached eddy simulations (DES) (Travin et al., 1999). One can refer to the works listed above, as well as references cited therein, for an extensive overview of numerical work in this area.

For a more affordable computational cost, these turbulent modelling approaches (RANS, LES) attempt to generate turbulence resolving simulations in which the turbulent scales are larger than the ones given by DNS. In large eddy simulations, larger turbulent scales are resolved while small scales called subgrid scales (SGS) are modelled instead. A spatial filter is then employed to separate the resolved turbulence from its modelled counterpart. For accurate LES predictions, a suitable filtering mechanism is required to separate large wave components from smaller ones at some cut-off frequency. Alternatively implicit LES based on intrinsic filtering mechanism of finite volume discretization has been shown to be capable of providing accurate prediction for flows over fixed and oscillating cylinders (Feymark et al., 2012). In both implicit and explicit LES implementation, there are restrictions on time step and grid sizes, especially for wall bounded flows where wall modelling functions have a strong effect on numerical predictions. In Koobus and Farhat (2004) variational multiscale LES was introduced with certain advantages in hierarchical implementation on unstructured grids and reduction of mesh resolution. Results of forces and frequency for simulations of flows over cylinder at $Re = 22\,000$ showed more accurate prediction than traditional LES approaches. While LES showed excellent results in capturing unsteady flow features and good agreement with experimental data in Feymark et al. (2012), its main shortcoming is to extend the development for engineering applications beyond canonical test cases where the computational cost is still much higher than industrial widely-adopted unsteady RANS models.

In RANS, all turbulence is modelled, and it is attempted to solve for ensemble-averaged quantities. By separating flows into mean and fluctuating components, RANS models normally require some empirical approximations to model turbulent fluctuations. Unsteady RANS (URANS) is capable of predicting low frequency unsteadiness, but it fails to capture high frequencies. These high frequency fluctuations are important in many cases, including wake interference exemplified by flows around cylinders in tandem. As shown in an earlier work (Rosetti et al., 2012) through systematic verification and validation exercises, it was observed that traditional RANS models do not provide sufficient resolution for flows over cylinders especially at high Reynolds number due to their intrinsic properties of isotropic eddy viscosities, homogeneous Reynolds stresses and failure to model all eddy scales. In recent work (Stringer et al., 2014), flows over cylinders at a range of Reynolds number were simulated using state-of-the-art unsteady RANS models from different commercial CFD packages and an opensource code (OpenFOAM). Through a number of presented test cases, it was shown that RANS was consistently capable of predicting flow characteristics well for low Reynolds number flows ($Re \leq 10^3$). The performance of RANS for high Reynolds number flows was scattering between different implementations and showed large deviations from experimental data for lift fluctuations as well as drag coefficients in critical regions. The two-dimensional study in the earlier work may contribute significantly to the poor performance of RANS models at high Reynolds number flows; nevertheless it clearly demonstrated shortcoming of RANS approaches in resolving flows over cylinders in those conditions.

It was suggested in Rosetti et al. (2012) that hybrid approaches based on URANS models are able to improve the predictability of simulations for high Reynolds number flows. To strike a balance between resolving and modelling of turbulence quantities, DES was introduced and successfully applied in practice for simulations of flows over circular cylinders Travin et al. (1999) as a hybrid RANS/LES approach. In combining RANS and LES, the approach functions as a LES sub-grid scale model in areas with sufficiently fine grid resolutions while it relies on Reynolds average models in areas with lower grid resolution. Since its first application for flows over cylinder, DES has attracted great attention from community in