Marine Structures 22 (2009) 142–153

Contents lists available at ScienceDirect

Marine Structures

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

Numerical simulation of flow around a smooth circular cylinder at very high Reynolds numbers

Muk Chen Ong [a,*], Torbjørn Utnes [b], Lars Erik Holmedal [a], Dag Myrhaug [a], Bjørnar Pettersen [a]

[a] Department of Marine Technology, Norwegian University of Science and Technology, NO-7491 Trondheim, Norway
[b] SINTEF IKT Applied Mathematics, NO-7465 Trondheim, Norway

ARTICLE INFO

Article history:
Received 14 March 2008
Received in revised form 4 July 2008
Accepted 8 September 2008

Keywords:
Numerical models
Cylinder
Turbulent flow
High Reynolds number

ABSTRACT

High Reynolds number flows ($Re = 1 \times 10^6$, 2×10^6 and 3.6×10^6, based on the free stream velocity and cylinder diameter) covering the supercritical to upper-transition flow regimes around a two-dimensional (2D) smooth circular cylinder, have been investigated numerically using 2D Unsteady Reynolds-Averaged Navier–Stokes (URANS) equations with a standard high Reynolds number $k - \varepsilon$ turbulence model. The objective of the present study is to evaluate whether the model is applicable for engineering design within these flow regimes. The results are compared with published experimental data and numerical results. Although the $k - \varepsilon$ model is known to yield less accurate predictions of flows with strong anisotropic turbulence, satisfactory results for engineering design purposes are obtained for high Reynolds number flows around a smooth circular cylinder in the supercritical and upper-transition flow regimes, i.e. $Re > 10^6$. This is based on the comparison with published experimental data and numerical results.

1. Introduction

One of the classical problems in fluid mechanics is the flow around a circular cylinder. This represents an idealized bluff body flow which is of great interest for a wide range of engineering applications, such as hydrodynamic loading on marine pipelines, risers, offshore platform support legs,

* Corresponding author.
E-mail address: muk.c.ong@ntnu.no (M.C. Ong).

0951-8339/$ – see front matter
doi:10.1016/j.marstruc.2008.09.001

etc. Many of these engineering applications are often subject to flow conditions corresponding to very high Reynolds number ($Re = U_\infty D/\nu$) flows with typical values of $O(10^6)$–$O(10^7)$. This covers the supercritical ($3.5 \times 10^5 < Re < 1.5 \times 10^6$) to transcritical ($Re > 4 \times 10^6$) flow regimes. A detailed definition of the flow regimes is given by Sumer and Fredsøe [22]. Here U_∞ is the free stream velocity; D is the cylinder diameter; and ν is the kinematic viscosity. These very high Reynolds number flow conditions are hard and expensive to achieve in an experimental setup requiring appropriate experimental facilities, minimizing human and instrument errors during measuring hydrodynamic quantities, etc. Therefore an attractive alternative is to use Computational Fluid Dynamics (CFD) to obtain the essential hydrodynamic quantities needed for engineering design. For example, Schulz and Meling [20] used a multi-strip method to analyze the flow-structure interaction of long flexible risers. This was achieved by solving the two-dimensional (2D) Unsteady Reynolds-Averaged Navier–Stokes (URANS) equations (using a Spalart–Allmaras turbulence model) in conjunction with a finite element structural dynamic response model. A number of individual 2D CFD simulations of cross sections along the riser were combined with a full 3D structural analysis to predict overall vortex-induced vibration (VIV) loads and displacement of the riser. They showed that a relatively modest number of sections could be used to capture multi-modal VIV responses in long risers. Chaplin et al. [4] compared laboratory measurements of the VIV of a vertical model riser in a stepped current with predictions obtained with 11 different numerical models. It was shown that empirical-based models were better in predicting cross-flow displacements than the CFD-based models. This might be due to uncertainties in CFD turbulence modeling and the modeling technique of the vortex-shedding interacting with dynamic response of the structure (see Ref. [4]). Thus, more computational and experimental research on high Reynolds number flows over a rigid cylinder section is necessary in order to gain better understanding on dynamic responses of slender marine structures.

To date, not many numerical simulations have been performed to predict very high Reynolds number flows ($Re > 10^6$) around a smooth circular cylinder due to the complexity of the flow. Direct Numerical Simulation (DNS) of flows at such very high Reynolds numbers is not presently possible because of the high demand on the computational resources. Among the few numerical results reported in the open literature (for $Re > 10^6$) are those of Catalano et al. [3] and Singh and Mittal [21]. Catalano et al. [3] applied 3D Large Eddy Simulation (LES) with wall modeling as well as URANS using the standard high Reynolds number $k-\varepsilon$ model of Launder and Spalding [12] with wall functions, for $0.5 \times 10^6 < Re < 4 \times 10^6$. Singh and Mittal [21] performed their studies for $100 < Re < 1 \times 10^7$ using a 2D LES method. Catalano et al. [3] mainly focused on the cases of $Re = 0.5 \times 10^6$ and 1×10^6 when the drag coefficient is recovering from the drag crisis (sudden loss of drag at $Re = 2 \times 10^5$). Their numerical results captured the delayed boundary layer separation and reduced drag coefficients correctly right after the drag crisis. They concluded that the LES results were considerably more accurate than the URANS results at $Re \sim 1 \times 10^6$. However, they also commented that the LES results became less accurate compared with the experimental data at higher Reynolds numbers due to their insufficient grid resolution. Singh and Mittal's [21] main objective was to investigate a possible relationship between the drag crisis and the instability of the separated shear layer. Their computations were able to capture the sudden reduction in drag coefficient close to the critical Re. Even though their study primarily focuses on the flow in the subcritical regime ($300 < Re < 3 \times 10^5$), they also presented some results for the flow beyond the supercritical flow regime, i.e. $Re > 10^6$.

The standard high Reynolds number $k-\varepsilon$ model has been incorporated into most commercial CFD codes. When used in conjunction with wall functions, it is generally taken as being computationally less expensive than LES and DNS. Nevertheless, the model has been well documented for several shortcomings, especially in the subcritical flow regime where the drag crisis occurs. Franke et al. [7] and Tutar and Holdø [23] evaluated numerically the detailed experiments of Cantwell and Coles [2] at $Re = 1.4 \times 10^5$. Franke et al. [7] applied URANS with the standard high Reynolds number $k-\varepsilon$ model of Launder and Spalding [12]; Tutar and Holdø [23] used both the standard high Reynolds number $k-\varepsilon$ model and non-linear $k-\varepsilon$ models. Their results were mainly obtained for the flow in the subcritical flow regime at the start of the drag crisis. Both studies concluded that the $k-\varepsilon$ models give an inaccurate prediction of flows with strong anisotropic turbulence. Catalano et al. [3] presented time-averaged drag coefficients for $Re = 1 \times 10^6$, 2×10^6 and 4×10^6, Strouhal number for $Re = 1 \times 10^6$ and mean pressure distribution for $Re = 1 \times 10^6$ using URANS with a standard high Reynolds number $k-\varepsilon$