Applied Ocean Research xxx (2017) xxx–xxx

Contents lists available at ScienceDirect

Applied Ocean Research

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

Investigation of drag crisis phenomenon using CFD methods

Peng Wen, Wei Qiu*

Advanced Marine Hydrodynamics Laboratory, Department of Ocean and Naval Architectural Engineering, Memorial University of Newfoundland, St. John's, Canada

ARTICLE INFO

Article history:
Received 30 March 2015
Received in revised form 13 June 2017
Accepted 19 July 2017
Available online xxx

Keywords:
Drag crisis
CFD
Turbulence modeling
RANS
DES
LES

ABSTRACT

When fluid flow passes a cylinder, the drag crisis phenomenon occurs between the sub-critical and the super-critical Reynolds numbers. The focus of the present studies was on the numerical prediction of the drag crisis based on CFD methods. In this work, block structured meshes with refined grids near the cylinder surface and in the downstream were employed. Both 2D and 3D simulations were performed using various turbulence models, including the SST $k-\omega$ model, the $k-\epsilon$ model, the SST with LCTM, the DES model, and the LES model. In the convergence studies, the effects of the grid size, the time step, the first grid size and the aspect ratio (for 3D simulations) on the solutions were examined. The errors due to spatial and time discretizations were quantified according to a V&V procedure. Validation studies were carried out for various Reynolds numbers between $Re = 6.31 \times 10^4$ and 7.57×10^5. The averaged drag force, the RMS of lift force and the Strouhal number were compared with experimental data. The studies indicated that standard 2D and 3D RANS methods were inadequate to capture the drag crisis phenomenon. The LES method however has the potential to address the problem.

1. Introduction

While the offshore oil and gas exploration is moving to greater water depths, one of the challenging issues is the fatigue of marine risers due to vortex-induced vibration (VIV). To address this issue, it is important to accurately predict the flow around the circular cylinder as well as its induced forces. Although the cylinder geometry is simple, it is challenging to predict the vortex dynamics using computational fluid dynamics (CFD) methods due to the flow instabilities in the wake, in the boundary layer and in the separating shear layer. The transition from laminar flow to turbulent flow and the boundary layer separation points depend on the Reynolds number. The flow passing a cylinder has been extensively studied by many researchers. Detailed description and analysis of the flow phenomenon can be found in the work of Williamson [1] and Schlichting [2].

The drag crisis phenomenon occurs around critical Reynolds numbers, $Re = 2\times10^5$–5×10^5, where the averaged drag coefficient drops drastically since the boundary layer transits from laminar flow to turbulent flow and the flow separates farther downstream. Studies have been carried out to investigate on the drag crisis phenomenon using theoretical, experimental and numerical methods. For example, Farell and Blessmann [3] and Bearman [4] conducted experiments on flow around a circular cylinder at a range of the critical Reynolds numbers. Similar tests were conducted at a large range of Reynolds numbers, 6×10^4–5×10^6, by Achenbach [5]. Some VIV experiments in towing tanks were carried out for both stationary and forced oscillating cylinders. For examples, de Wilde and Huijsmans [6] carried out high Reynolds number VIV experiments with a fixed circular cylinder; de Wilde et al. [7] conducted experiments with a transversely oscillating cylinder in steady current; de Wilde and Huijsmans [8] experimentally studied the 3D response of a part of a long riser in current; and de Wilde et al. [9] carried out VIV experiments on a rigid cylinder using particle image velocimetry (PIV). It was shown in the experimental results by de Wilde and Huijsmans [6] that a significant drag loss occurred at $Re = 10^4$–10^6.

CFD methods have been extensively used to study the drag crisis phenomenon of stationary or forced cylinders. For example, Rosetti et al. [10] applied a unsteady Reynolds Averaged Navier–Stokes equation (URANS) solver with the Local Correlation Transition Model (LCTM) to simulate the flow at high Reynolds numbers. Rosetti et al. [11] used a three-dimensional URANS solver and the SST $k-\omega$ turbulence model to predict the forces on the cylinder at various Reynolds numbers and estimate the numerical errors according to the verification and validation (V&V) procedure [12]. Ong et al. [13] used RANS and the $k-\omega$ turbulence model to simulate the cross flow around a two-dimensional circular cylinder

* Corresponding author.
E-mail address: qiuw@mun.ca (W. Qiu).

http://dx.doi.org/10.1016/j.apor.2017.07.012
0141-1187/© 2017 Elsevier Ltd. All rights reserved.

at high Reynolds numbers, $Re = 10^6$, 2×10^6 and 3.6×10^6. Vaz et al. [14] carried out both two-dimensional and three-dimensional simulations for a stationary smooth circular cylinder using ANSYS-CFX10 with various turbulence models. The drag loss in the critical region was however not well captured. Benchmark studies on the flow passing a stationary cylinder were organized by the ITTC Ocean Engineering Committee [15]. All the participants employed the RANS methods in their simulations and the drag crisis phenomenon was however not captured.

The large eddy simulation (LES) method has recently been applied to simulate the flow passing circular cylinders at high Reynolds numbers. Studies have demonstrated that the drag loss could be better predicted by the LES methods. For example, the drag loss was well predicted using the LES model in the work of Botterill et al. [16]. They also compared the results based on the Smagorinsky model and the dynamic sub-grid scale model. It was indicated in their studies that the dynamic sub-grid scale model was required to simulate the drag crisis. The Smagorinsky model however misrepresented the turbulent eddy viscosity at the wall which led to inaccurate prediction of the separation point. de With et al. [17] and Catalano et al. [18] also used the LES method to simulate the cross flow around a circular cylinder.

The detached eddy simulation (DES) method is another approach that has also been employed to address the problem. For example, Vaz et al. [14] used the DES model to predict the drag and lift forces on a stationary cylinder at two Reynolds numbers, $Re = 9.3\times10^4$ and 5.5×10^5.

Efforts have also been made to simulate the turbulent flow passing circular cylinders using the direct numerical simulation (DNS) method. For example, Dong et al. [19] employed the DNS model to simulate the turbulence flow past stationary and forced oscillating cylinders at a Reynolds number of 10^4.

This paper presents studies of flow passing a stationary circular cylinder at various Reynolds numbers in the drag crisis region. Both two-dimensional and three-dimensional computations were carried out based on RANS, DES and LES methods using a commercial software package, Star-CCM+. The effects of grid resolution, time step, y^+ value, aspect ratio of the cylinder, and turbulence models on the solutions were studied. Numerical errors due to spatial and time disrectizations were estimated according to the V&V procedure proposed by Eça and Hoekstra [20]. The predicted averaged drag coefficients, the Strouhal numbers and the RMS values of lift coefficients are presented and compared with experimental results.

2. Numerical methods

In the simulation of the unsteady viscous flow around a fixed circular cylinder, it is assumed that the fluid is Newtonian and incompressible. Various numerical models were employed in this work, including the two-dimensional RANS model, the three-dimensional RANS model, and the three-dimensional LES and DES models, to solve the turbulent flow problem. The theoretical aspects of the computational methods are summarized below.

2.1. RANS model

The governing RANS equations for the three-dimensional incompressible viscous flow consist of the continuity equation and the momentum equations as follows:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

$$\rho\frac{\partial u_i}{\partial t} + \rho u_j\frac{\partial u_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\right] + \frac{\partial}{\partial x_j}(-\rho\overline{u_i'u_j'}) \tag{2}$$

where u_i, $i = 1$, 2 and 3, denote the time-averaged velocity components along x-axis, y-axis and z-axis, respectively, ρ is the density of the fluid, μ is the dynamic viscosity of the fluid, p is the pressure, δ_{ij} is the Kronecker delta, and $-\rho\overline{u_i'u_j'}$ are the Reynolds stresses which can be computed based on the Boussinesq hypothesis using the eddy viscosity models, or be solved from the transport equations based on Reynolds stress models.

In the eddy viscosity models, it is assumed that the Reynolds stresses are related to the mean velocity gradients, the turbulent kinetic energy, $k = \overline{u_i'u_j'}/2$, and the eddy viscosity, μ_t, i.e.,

$$-\rho\overline{u_i'u_j'} = \mu_t\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right) - \frac{2}{3}\rho k\delta_{ij} \tag{3}$$

The turbulent kinetic energy, k, can be solved from the transport equation, and the turbulence dissipation rate, ϵ, or the specific dissipation rate, ϵ/k, can be solved from the transport equations in turbulence models such as $k-\epsilon$ and $k-\omega$ models.

In addition to the standard two equation turbulence models, the LCTM, a correlation-based transition model, was also employed. The detailed formulation of the LCTM can be found in the work by Langtry [21] and Menter et al. [22]. Two additional transported quantities are involved in the LCTM. The first one is the transition Reynolds number based on momentum thickness, $\bar{Re}_{\theta t}$, which is a non-local variable. The evaluation of non-local variables is avoided by calculating $Re_{\theta t}$ using an empirical correlation in the freestream and diffusing it to the boundary layer. Based on empirical correlations, $\bar{Re}_{\theta t}$ is then used to calculate the critical Reynolds number, $Re_{\theta c}$, and the transition length, F_{length}. The transport equation for the intermittency factor, γ, can then be solved. The intermittency factor is a trigger of local laminar/turbulent state and is coupled with the $k-\omega$ SST model. In short, two critical correlations are needed in this model, one for $Re_{\theta t}$ and the other one for $Re_{\theta c}$ and F_{length}. The choices of the empirical correlation can be varied in different CFD codes. In Star-CCM+, the first correlation is based on the work of Langtry [21] and the second correlation follows the empirical correlation in the work of Suluksna et al. [23]. As two additional transport equations are solved, the LCTM model requires more iterative steps and time to converge.

2.2. LES model

Due to the different behaviors of large and small eddies in the turbulent flows, RANS models cannot be employed in all applications. The small eddies are nearly isotropic, and the large eddies are however more anisotropic and are affected by the mean flow. Instead of describing all scales of eddies using a single turbulence model as in RANS, LES models compute large eddies with time-dependent simulations and account for the effects of small scale eddies using a sub-grid scale (SGS) model. A spatial filter is used to separate the large and small eddies. By exploiting the linearity of the filtering function, the continuity equation and momentum equations for LES are in the same form as those for the RANS model in Eqs. (1) and (2). The sub-grid scale stresses, $-\rho\overline{u_i'u_j'}$, obtained from the filtering is similar to the Reynolds stresses in the RANS formulation, where the over-bar indicates a filtered flow variable.

The SGS stresses are modeled according to the Boussinesq approximation in Eq. (3). The sub-grid scale viscosity, μ_t, can be

Please cite this article in press as: P. Wen, W. Qiu, Investigation of drag crisis phenomenon using CFD methods, Applied Ocean Research (2017), http://dx.doi.org/10.1016/j.apor.2017.07.012