Available online at www.sciencedirect.com

ScienceDirect

Journal of Ocean Engineering and Science 3 (2018) 205–217

www.elsevier.com/locate/joes

Original Article

# Efficient three-dimensional high-resolution simulations of flow fields around cylinders

Hanxu Zheng[a,b], Jiasong Wang[a,b,*]

[a] School of Naval Architecture, Ocean and Civil Engineering, Shanghai Jiao Tong University, Shanghai 200240, China
[b] MOE Key Laboratory of Hydrodynamics, Shanghai Jiao Tong University, Shanghai 200240, China

Received 19 January 2018; received in revised form 31 July 2018; accepted 6 August 2018
Available online 29 August 2018

**Abstract**

Few works use the fully three-dimensional computational fluid dynamic method to simulate the flow fields around the marine pipes with large aspect ratios due to the huge computation cost. In the present work, an operator-splitting method is used to efficiently solve the three-dimensional Reynolds Average Navier–Stokes governing equations of the fluid flow around pipes by separating the problem as a combination of a two-dimensional problem in the horizontal plane and an one-dimensional problem in the vertical direction. A second order total variation diminishing finite volume method is used to solve the model. The precision of the present model is validated by comparing the present numerical results of two typical three-dimensional cases with the available experimental and numerical results. The simulation results with a commercial software are also included in the comparison and the present model shows a higher performance in terms of computational time.


*Keywords:* CFD; Cylinder; Cube; Operator-splitting; TVD; Three-dimensional.

## 1. Introduction

The long offshore marine riser structural responses under the practical ocean conditions have been numerically studied by many researchers as reflected in the reviews by Bearman [1], Williamson [21] and Norberg [11]. The computational fluid dynamic (CFD) method has been considered as a superior method in the relevant flow field simulations that provide the fluid forces exerted on to the structure. However, due to the huge computation cost, it has not been widely used in the study of the marine pipes with large aspect ratios. Many works use the wake oscillating method [15] to approximately calculate drag and lift coefficients since it is simple and less computational resource consuming. In a recent work of Nishi and Doan [10], the authors used this model to study the vortex-induced vibration (VIV) suppression mechanism of the damping devices. However, it is a simplified model and the turbulent flow traits are omitted. The parameters of the model rely on certain experimental data and may not be fit for the other practical cases. The discrete vortex method (DVM) is also usually used by the researchers [22]. It models the vortex motion in the wake of flow past bluff bodies, requires small calculation resources and provides less precise simulation results than the ordinary CFD methods.

In the research works where the Navier–Stokes(N–S) equations are used as the governing equations of the fluid flow, the structures usually have confined aspect ratios with $L/D < 100$. Pontaza and Chen [12] used the large eddy simulation (LES) method to study the VIV of the circular cylinders undergoing two degrees of freedom and the cylinder had an aspect ratio of $L/D = 3$. In the work of Pontaza et al. [13] the LES method is used to simulate the three-dimensional (3-D) flow past the smooth and the rough circular cylinders either bare or out fitted with helical strakes while the studied cylinders have an aspect ratio of $L/D = 9$. Zhao et al. [23] used the 3-D N–S equations to study the VIV of an elastically mounted rigid circular cylinder in the steady current and the cylinder

* Corresponding author.
*E-mail address:* jswang@sjtu.edu.cn (J. Wang).

https://doi.org/10.1016/j.joes.2018.08.001

**Nomenclature**

| | |
|---|---|
| $u$ | component of velocity in the $x$ direction |
| $v$ | component of velocity in the $y$ direction |
| $w$ | component of velocity in the $z$ direction |
| $p$ | pressure |
| $k$ | kinetic energy |
| $\omega$ | dissipation rate relative to kinetic energy |
| $t$ | time |
| $\beta$ | pseudo compressibility coefficient |
| $\nu$ | kinetic viscosity |
| $\mu$ | dynamic viscosity |
| $\nu_T$ | turbulence viscosity |
| $y'$ | distance between the center of mesh element and nearest wall boundary |
| $G$ | turbulence production term |

in the research had an aspect ratio of $L/D = 7$. In the work that studies the structures with large aspect ratios, the two-dimensional (2-D) strip method is usually used in order to reduce the size of the numerical solutions. A few fully 3-D research works have been done on the large aspect ratio structures by some scholars. Constantinides et al. [2] and Holmes et al. [4] used a fully 3-D numerical method to predict the VIV of a riser with $L/D = 1407$ while very coarse triangular fluid mesh was used to keep the size of the problem tractable. Huang et al. [5] used a chimera model combined with the LES and the Reynolds Average Navier–Stokes (RANS) to predict the 3-D VIV response for an 10 meter riser that has an aspect ratio of $L/D = 482$ in experiment. A resource consuming finite-analytic method is used to solve the model in this work. Wang et al. [18] used the RANS equations closed with the $k - \omega$ turbulence model to simulate a three dimensional VIV case for a 500-meters-long marine riser with $L/D = 937$ using a supercomputer while the computational cost is very huge and the software run for a very long time. From the previous studies, the huge computation cost has been found to be the major obstacle in using the fully 3-D CFD method to simulate the flow fields around the cylinders with large aspect ratios.

To effectively simulate the flow fields surrounding the practical marine structures with large aspect ratios, we seek to build a numerical model that consumes fewer computation resources while maintains a certain level of numerical precision. Since the precision and the computation cost of the RANS model lies between the LES and the DVM, it is selected in the present model. An operator-splitting method is also used to reduce the computation cost while it brings no additional numerical error to the simulation results of the cases with uniform cross-sections. The shear stress transport (SST) turbulence model [9] is used to close the RANS model and a total variation diminishing (TVD) finite volume method (FVM) is used to solve the model. The TVD-FVM method of Wang et al. [20] is effective for solving the governing equations of the fluid with structured fluid meshes. This numerical method is combined with the element velocity vector transform (EVVT) method [19] to solve the problems with curvilinear boundaries while no extra effort is needed to deal with the complex curvilinear coordinates. To validate the precision of the present model, the flow over a ground-mounted cylinder and cube is simulated and compared with the available experimental and numerical results. To examine the efficiency of the present numerical method, the case with the ground-mounted cube is simulated using a commercial software CFX and the numerical efficiency is compared with that of the present model.

The flow over a circular cylinder placed on a rigid floor has been a subject studied by many investigators. It is chosen as one of the cases in the present work because the 3-D effect in the flow field can be extremely apparent. The horseshoe vortex emerges at the front bottom of the cylinder due to the reverse of the downward flow on the upstream side of the cylinder. On the downstream side of the pipe, a kind of vortex shedding that is similar to the 2-D Karman vortex street emerges. The flow over a ground-mounted cube can be more complex due to the sharp edges and the flow over the top. The sharp edges make the flow separation easier to occur and the flow over the top will have profound effects on the vortex structures in the wake of the flow field.

The experimental research on the ground-mounted circular cylinder has been carried out by many researchers. In the present work, the experimental results of Dargahi [3] are used as the benchmark. Dargahi [3] provided a series of experimental results including the pressure distribution, the velocity distribution and the position of the primary horseshoe vortex. Several scholars numerically studied this case and their results are compared with the present numerical result. Tseng et al. [16] used a LES turbulent model (cell-averaged sub-grid scale turbulence model) and MacCormack's explicit predictor-corrector scheme to simulate the 3-D turbulent flow field around the circular pier. Roulund et al. [14] used the RANS and the SST turbulence model to simulate the flow field around the circular cylinder while a HYBRID scheme that combines a first-order upwind difference scheme and a second-order central difference scheme was used to solve the model. In the numerical work of Zhao et al. [24], the author also used the RANS closed with the SST turbulence model to simulate the fluid flow while a finite element method was used to solve the model.

Martinuzzi and Tropea [8] provided the classic experimental results for the flow past a ground-mounted cube and the experimental setup conditions are used in the present numerical simulation of this case. Several simulation results of this case using the other numerical models are compared with the present numerical result. Lakehal and Rodi [7] used a steady model to simulate the flow past a surface-mounted cube. The author used different variations of the $k - \varepsilon$ turbulence model to close the governing equations and a central difference FVM was used to solve the model. A simulation of this case is also performed with the commercial software CFX in order to validate the efficiency of the present code. The SST turbulence model is also used in this simulation and the model is solved