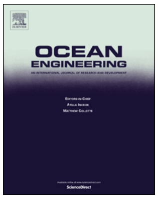
Ocean Engineering 118 (2016) 107–116

Contents lists available at ScienceDirect

Ocean Engineering

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



Simulation of vortex-induced motions of a deep draft semi-submersible in current

Chia-Rong Chen [a], Hamn-Ching Chen [b,*]

[a] College Station, TX 77845, USA
[b] Zachry Department of Civil Engineering, Texas A&M University, College Station, TX 77843-3136, USA

ARTICLE INFO

Article history:
Received 10 November 2015
Received in revised form
17 February 2016
Accepted 5 April 2016
Available online 13 April 2016

Keywords:
CFD
Deep draft semi-submersible
Vortex-induced motions (VIM)
Large eddy simulation
Corner geometry effect
Scale effect

ABSTRACT

The vortex-induced motion (VIM) of semi-submersible platforms becomes an important issue with the recent development of deep draft semi-submersible platforms. As a result of the increased draft, the semi-submersibles are susceptible to coherent vortex shedding, and the VIM increases significantly. The VIM of semi-submersibles is more complex than those of spars and mono-column hulls, due to the wake interaction of vortices shed from multiple columns. In the present study, numerical simulations are performed for a semi-submersible with four square columns subject to a current at a 45° incidence angle and allowed surge (in-line), sway (transverse), and yaw motions. Calculations were performed using the Finite-Analytic Navier–Stokes (FANS) code in conjunction with a moving overset grid approach. Computations are conducted over a wide range of reduced velocities, from pre-lock-in to post-lock-in conditions. Both the full scale and the 1:70 model platforms are studied and detailed results compared to check the scale effect. In addition, three corner geometries are simulated, and the semi VIM is found to be sensitive to the corner rounding. Comparisons are made with experimental data to demonstrate the capability of the present CFD approach.

1. Introduction

The vortex-induced motion (VIM) of semi-submersible offshore platforms becomes an important issue with the recent development of deep draft semi-submersibles. As a result of the increased draft to meet the payload requirement for challenging reservoirs in deep water, the semi-submersibles are susceptible to coherent vortex shedding, and the platform VIM increases significantly. This phenomenon impacts the fatigue life of the mooring system and risers greatly, and many model tests have been conducted by the offshore industry to address the issue. The VIM of semi-submersibles is more complex than those of spars and mono-column hulls due to the wake interaction of vortices shed from multiple columns. Experimental works on semi-submersibles were presented by Waals et al. (2007), Rijken and Leverette (2008), Hong et al. (2008), Magee et al. (2011), Xu et al. (2012) and Goncalves et al. (2012). More recently, unconventional semi-submersible designs were proposed to suppress VIM. Model tests were reported by Xu et al. (2012) for a semi with blisters added to the bases of the columns, and by Zou et al. (2013) and Antony et al. (2015a) for a dry tree paired-column design. However, the focus of the present study is on the conventional semi-submersible platforms. In general, the VIM of a deep draft semi is characterized by three degree-of-freedom motions, namely the surge (in-line), sway (transverse), and yaw motions, with the sway motion as the main concern in VIM. Model tests are usually performed in 1:100 to 1:50 scale, and the reduced velocity is the most important parameter used to interpret the motion responses. Most model tests revealed that the largest sway motion, namely the lock-in condition, was observed at reduced velocity around 6–8 for a semi towed in a 45° direction. See Waals et al. (2007), Rijken and Leverette (2008), Xu et al. (2012) and Goncalves et al. (2012). Lock-in occurs when the structure response frequency is nearly identical to the vortex shedding frequency, and the response motion becomes oscillatory with almost constant amplitude and period.

Whether the full scale structures have the response motions scaled up from the model tests remains a concern for many researchers. Rijken and Leverette (2008) cautioned that the reduction in the Reynolds number of the model might generate a different viscous flow, leading to different response motions between the model and prototype. Roddier et al. (2009) tested spar models of three different scales and showed those done in low Reynolds number resulted in higher amplitudes, suggesting the need to carry out the tests in high Reynolds number to give more accurate predictions for prototypes. Field measurements of deep draft semi VIM reported by Rijken and Leverette (2009) and Ma et al. (2013) both indicated smaller VIM than predicted by model test. However, the difference between field measurements and model tests could be caused by several factors, such as the

---

* Corresponding author.

http://dx.doi.org/10.1016/j.oceaneng.2016.04.005
0029-8018/

damping associated with the riser and mooring system and the field current profiles versus the uniform current condition in a model basin. The most realistic way to check the scale effect between the model and prototype is by conducting Computational Fluid Dynamics (CFD) simulations. Kim et al. (2011) used AcuSolve and StarCCM+ to simulate for a TLP. Tan et al. (2013) used StarCCM+ to simulate the VIM of semi-submersibles of two different drafts. Xu et al. (2012) also used AcuSolve. These CFD calculations with commercial codes simulated only for a few reduced velocities. Lee et al. (2014) used a Finite-Analytic Navier–Stokes (FANS) code and covered a much wider range of reduced velocity, and their results produced the typical pre-lock-in, lock-in, and post-lock-in behaviors. However, they made no attempt to compare with existing model tests. The objective of the present study is to investigate the validity of the scaling law by systematic numerical simulations over a wide range of reduced velocity for both the model and full scale semi, and to make comparison with the experimental data. The effect of geometry caused by minor rounding of the corners of the columns is also studied. In our simulation scenario, a conventional deep draft semi-submersible with four square columns is subject to a current at a 45° incidence angle and allowed in-line, transverse, and yaw motions.

2. Numerical method and setup

2.1. Methodology

In the present study, the Finite-Analytic Navier–Stokes (FANS) numerical method of Chen et al. (1990) and Pontaza et al. (2005) is employed to solve the unsteady, incompressible Navier–Stokes equation. A fully parallelized multi-block overset grid approach is utilized to accommodate the complex flow and relative motions between the semi-submersible hull, wake, and background grid blocks. The pressure and shear stresses on the hull surfaces are integrated to obtain the forces and moments, and a six degree-of-freedom motion program then solves for the surge, sway, and yaw motions. Lee et al. (2014) used the same FANS code and chose the two-layer k–ε turbulence model of Chen and Patel (1988). In the current study, the turbulent boundary layer and wake flow around the structure is solved with the large eddy simulation (LES), which was shown to provide accurate prediction of vortex-induced vibrations (VIV) of deep water risers in uniform and shear currents (Huang et al., 2010, 2012) as well as vortex- and wake-induced vibrations (WIV) of dual risers in tandem and side-by-side arrangements (Chen et al., 2013). Detached eddy simulation (DES) is also performed to see how it compares with the LES result, and the difference in semi VIM from the two turbulence models is shown to be insignificant in Section 3.1 on convergence tests.

In the finite-analytic formulation, the transport equations for mean momentum and turbulence quantities are linearized locally within each numerical element, and the pressure gradient terms are treated as known source functions. Using the natural solution of the linearized equations as boundary conditions along the edges of each element, the Navier–Stokes equations are solved by the method of separation of variables to obtain local analytic interpolants in terms of unknown neighboring nodal values of the velocity components and pressure. The coefficients associated with the local analytic interpolants are functions of the local velocity field and respond analytically to local flow conditions. In addition, the interpolant coefficients satisfy zeroth and first order consistency requirements and are always positive. These properties ensure no spurious energy modes and a stable scheme at high Reynolds numbers. The numerical scheme is completed with the hybrid SIMPLER/PISO pressure solver of Chen and Korpus (1993) and Pontaza et al. (2005), which satisfies the continuity equation at each time step.

The local analytic interpolants are constructed in a transformed space, thus the curvilinear elements handling complex geometries in a practical application are treated in the same way as for Cartesian elements. Furthermore, multi-block overset grid approach is adopted. This allows for great flexibility in judicial mesh refinements for complex geometries involving embedded and overlapping grids. Mass-conserved Lagrange interpolation is implemented at the fringes of the overlapping region for inter-grid communication. This allows for efficient simulation of arbitrarily large motions among various computational blocks without the need of tedious and costly grid-regeneration and mesh-deforming monitoring. For problems involving violent free surface flows, the FANS method has been used successfully in conjunction with the level-set method of Chen and Yu (2009) for the simulation of hurricane wave load on offshore platforms (Chen, 2010, 2013) and the bow and stern slamming of a containership in random waves (Chen and Chen, 2015).

2.2. Semi-submersible model and grid structure

The deep draft semi-submersible of four square columns and four pontoons of Waals et al. (2007) is selected as the structure for simulation. Fig. 1 illustrates the dimensions of the prototype in a side view sketch. The column width, L, is 14 m, the draft 35 m, and the column height above the pontoon, H, is 24.5 m. The mass is given as 4.4×10^{7} kg. The semi-submersible length and width are not provided and are estimated to be 70 m. The moment of inertia for yawing is also unavailable and estimated to be $4.97\times10^{10}\ \mathrm{kg\cdot m^2}$ with a radius of gyration of 33.6 m. The model tests were conducted in 1:70 scale. Therefore, computations are performed for both the full size and the 1:70 model. The free surface effect is negligible in the VIM study of surge, sway and yaw, so only the submerged part of the floater is considered in the simulations.

The columns were sharp cornered in Waals et al. (2007). It is still interesting to see if the rounding, even if very slight, in the corners may be a non-negligible factor in semi VIM. The corner rounding effect could be observed in Antony et al. (2015b) when various documented wind tunnel data for a fixed square column were compared. For the sharper cornered case, higher drag coefficients were measured at all incidence angles, while the lift coefficients were smaller in general though dependent on the incidence angle. Besides the column geometry, the pontoon geometry may also affect the deep draft semi VIM. Nevertheless, the present work concentrates only on the column geometry effect. Two corner radii are used in this study and are plotted to the scale in Fig. 2(a) for a quarter of the column cross section. The first is a very slight rounding with a radius of $r=0.5$ m for the full size semi. The diagonal column width is then $D=19.385\ \mathrm{m}=1.385\,L$, only 2.1% smaller than $\sqrt{2}L$. The corners still look fairly sharp, as depicted in Fig. 2(b). Fig. 2(c) shows the semi with a corner radius of $r=2.22$ m, which gives $D=17.96\ \mathrm{m}=1.283\,L$, 9.3% smaller than $\sqrt{2}L$.

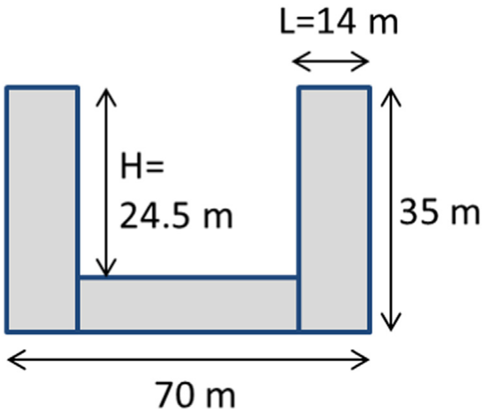


Fig. 1. Dimensions of the semi-submersible simulated.