Computers & Fluids 115 (2015) 256–269

Contents lists available at ScienceDirect

Computers & Fluids

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

Current blockage in a numerical wave tank: 3D simulations of regular waves and current through a porous tower

H. Santo [a,b,*], P.H. Taylor [a,b], W. Bai [b], Y.S. Choo [b]

[a] Department of Engineering Science, University of Oxford, Parks Road, Oxford OX1 3PJ, United Kingdom
[b] Centre for Offshore Research & Engineering, Department of Civil and Environmental Engineering, National University of Singapore, 1 Engineering Drive 2, Singapore 117576, Singapore

ARTICLE INFO

Article history:
Received 29 September 2014
Accepted 2 April 2015
Available online 9 April 2015

Keywords:
Porous tower
Current blockage
Morison loading
Actuator disc

ABSTRACT

This paper introduces a new numerical approach for the estimation of the global hydrodynamic loads on space-frame offshore structures exposed to combined waves and current. We provide numerical evidence for reduced fluid loading on offshore structures – current blockage, which serves as an extension to the analytical, computational and experimental work of Taylor et al. (2013) and Santo et al. (2014a, 2014b). A full 3D free-surface turbulent flow is simulated for a porous tower in a numerical wave tank. This is intended to model waves and current through a jacket or compliant tower, both space-frame structures. Comparisons are made between the numerical simulations and experiments conducted by Allender and Petrauskas (1987) on a scale-model jacket structure from the Gulf of Mexico, and the current blockage model presented previously in Taylor et al. (2013). Three different flows are simulated: steady current, regular waves with no current and regular waves with an in-line current. Overall, good agreement in terms of peak total forces is achieved, showing that the force reduction on such structures due to current blockage effects is real and significant. Additional information on force time history and flow visualisation are presented from the numerical simulations. Flow visualisation for waves and current reveals that the form of the global mean wake is simple at the structure but becomes complex well downstream. The simple form of the flow at the tower is responsible for the global force reduction being predictable using a modified version of the Morison equation (Morison et al., 1950). This paper also demonstrates the novel use of a porous tower as a simple representation for the complex geometry of real space-frame structures when exposed to combined large waves and significant in-line current, an approach which could be considered for possible incorporation into offshore design practice.

© 2015 Elsevier Ltd. All rights reserved.

1. Introduction

Taylor [15] reported an early study of current blockage model which was derived using actuator disc theory. This can be termed ‘simple current blockage’ (SCB) as it only accounts for current–structure interaction, thus is suited for pure steady flow only. It has been used as a part of the standard design method for space-frame offshore structures after it was incorporated in the API design guidelines in 1994 [2]. Recently, Taylor et al. [16] presented a more complete model for regular waves with an in-line current to improve the Morison equation [12]. Here this is termed ‘full current blockage’ (FCB) as it accounts for wave–current–structure interaction. That FCB model was expressed in terms of peak force, and was later extended to include prediction of the compete force time history over a wave cycle in Santo et al. [14], where extensive experimental validations were also performed for current blockage in regular oscillations plus mean flow. The FCB model was also validated numerically in Santo et al. [13] where planar flow through a porous block was modelled numerically. Having demonstrated experimentally and numerically that the force reduction on an obstacle array due to current blockage effects in regular oscillations plus mean flow is real and significant, we now proceed to incorporate the effects of realistic depth-varying wave kinematics with an in-line current flow with free-surface effects in Computational Fluid Dynamics (CFD) simulations.

This paper investigates the effect of current blockage on a typical space-frame offshore structure by simulating 3D regular wave flow through a porous tower in a numerical wave tank via CFD simulations. In this way, more realistic water particle kinematics could be included, and the integrated effect of current blockage through the water column could be analysed. As the dominant physical process we seek to model is the reduced mean flow within

* Corresponding author at: Department of Engineering Science, University of Oxford, Parks Road, Oxford OX1 3PJ, United Kingdom.

E-mail address: harrif.santo@eng.ox.ac.uk (H. Santo).

http://dx.doi.org/10.1016/j.compfluid.2015.04.005
0045-7930/© 2015 Elsevier Ltd. All rights reserved.

and near a space-frame offshore structure over distances of the order of the width of the structure (not the local flow around individual cylinders within the structure), a 3D porous tower made up of porous cells with a specified level of resistance is used to represent the complex geometry of the structure and to represent the bulk of the flow structure and the associated fluid–structure interaction.

The aim of this paper is to test the modelled global forces on space-frame structures by reproduction of experimental results measured by Allender and Petrauskas on a scaled jacket model subjected to a wide range of regular wave heights and steady tow speeds [1]. With the extra loading contribution from the waves superimposed on top of the steady current flow over a structure, extra resistance thus extra blockage is expected. Allender and Petrauskas observed significant flow blockage occurred in their tests, which they reported "requiring the use of a lower C_d of 0.7–0.8 for waves plus current from a C_d of 1.3–1.6 for waves alone to fit the peak forces of the standard Morison theory with the measured results" [1]. Unfortunately, they did not publish any plots of force time histories for the experimental model, only the *peak* values.

This paper will attempt to reproduce their reported peak forces with a single and consistent set of Morison coefficient $C_d = 0.9$ for drag and $C_m = 2.0$ for inertia for regular waves with and without current over a wide range of wave heights and current speeds. We also provide additional information on the force time histories and the visualisation of the flows. Direct comparisons between the numerical simulation and the measured peak forces will be made. We will demonstrate that good agreement can be achieved between the numerical simulations and the measured data.

One novel part of this paper is the demonstration of the use of a quadratic resistance porous tower as a simple model for the complex geometry of a real space-frame offshore structure, where the drag resistance and the inertia contribution could be calibrated and modelled. Even though the local flow structures are not modelled at the scale of the individual structural elements within a jacket-type platform, the global flow behaviour is reasonably well represented. This technique could potentially be incorporated into a standard offshore design practice to investigate current blockage effects.

2. Numerical methods

In this section, we first present the numerical methods necessary to simulate the propagation of regular waves through a porous tower in a numerical wave tank.

2.1. Governing equations

The governing equations for the two-phase combined flow of water and air are the Reynolds-averaged Navier–Stokes equations coupled with the continuity equation for incompressible flows, with an additional momentum sink term to account for the effect of the porous tower in the numerical simulation:

$$\nabla \cdot \mathbf{u} = 0 \quad (1)$$

$$\frac{\partial \rho \mathbf{u}}{\partial t} + \nabla \cdot [\rho \mathbf{u}\mathbf{u}^T] = -\nabla p^* + \nabla \cdot [\mu \nabla \mathbf{u} + \rho \boldsymbol{\tau}] - \mathbf{S} + [-(\mathbf{g} \cdot \mathbf{x})\nabla \rho + \sigma_T \kappa_\gamma \nabla \gamma] \quad (2)$$

where ρ is the fluid density, $\mathbf{g}$ is the acceleration due to gravity, $\mathbf{u} = (u, v, w)$ is the fluid velocity field in Cartesian coordinates, p^* is the pressure in excess of hydrostatic pressure, defined as $p^* = p - (\mathbf{g} \cdot \mathbf{x})\rho$, μ is the dynamic viscosity, $\mathbf{x} = (x, y, z)$ is the local Cartesian coordinates, and $\boldsymbol{\tau}$ is the specific Reynolds stress tensor.

Here, momentum lost from the flow is accounted for via a sink term, which is proportional to a nonlinear drag loss term (a Morison-type quadratic resistance with the $\mathbf{u}|\mathbf{u}|$ form, so that the flow can be in any direction) and a bulk inertia term which models the local Morison inertia contribution due to potential flow-type distortions over scales of the order of the width of the individual cylinders in a real space-frame offshore structure. Hence, in the case of a simple homogeneous porous tower:

$$\mathbf{S} = \frac{1}{2}\rho F \mathbf{u}|\mathbf{u}| + C'_m \frac{\partial \rho \mathbf{u}}{\partial t} \quad (3)$$

where F is the Forchheimer resistance parameter and C'_m is the equivalent of the standard Morison inertia coefficient, C_m, but here defined in the porous tower context. The nonlinear drag loss term is treated explicitly as an additional sink term in the momentum equation, while the bulk inertia term is grouped together with the unsteady term and solved implicitly in time.

The last two terms in Eq. (2) in square brackets are for numerical convenience for volume of fluid (VOF) method, and only active in the region where cell is partially filled with air, elsewhere these terms are zero. The $\sigma_T \kappa_\gamma \nabla \gamma$ term describes the surface tension effect using the CSF (Continuum Surface Force) model of Brackbill et al. [5], where σ_T is the surface tension coefficient, and κ_γ is the surface curvature. γ is a scalar field used to represent the fraction of a cell volume filled with water, with $0 \leqslant \gamma \leqslant 1$; 0 for air, and 1 for water.

The equations are solved simultaneously for the two immiscible fluids together with the transport equation used to track the fluids. The transport equation is similar to the volume of fluid (VOF) method of Hirt and Nichols [9], but with an additional compression technique to limit the numerical diffusion of the interface profile. The compression technique is developed by OpenCFD®, and the documentation can be found in Berberović et al. [4].

The unsteady, incompressible and 3D two-phase flow equations of motion are solved with the finite volume method [6] using the open-source code OpenFOAM® (http://www.openfoam.com). This study uses the numerical wave tank 'waves2Foam' developed and released by Jacobsen et al. [11]. The momentum–pressure coupling is solved with the PISO (Pressure-implicit Split-Operator) iterative algorithm [10].

This study uses an LES (Large Eddy Simulation) k-equation eddy-viscosity turbulence model. The one-equation eddy viscosity subgrid scale (SGS) model for incompressible flows using a modelled balance equation to simulate the behaviour of k is based on Fureby et al. [7], but with modification to the dimensionless model coefficients. The dimensionless model coefficients are given the default OpenFOAM® values $c_k = 0.094$ and $c_\epsilon = 1.048$.

2.2. Calibration of global force coefficients

Both F and C'_m can be calibrated by matching $\int \mathbf{S}\, dV_P$ with the vector form of the Morison force equation:

$$\mathbf{F}(t) = \frac{1}{2}\rho C_d A \mathbf{u}|\mathbf{u}| + C_m V \frac{\partial \rho \mathbf{u}}{\partial t} \quad (4)$$

where C_d is the Morison drag coefficient, A is the solid drag area of the components comprising the space-frame offshore structure (jacket or compliant tower), V is the volume of the structural components within the structure, and $V_P = A_f \times L$ is the volume of the modelled porous tower, where A_f is the frontal area of the structure represented in the porous tower and L is the width of the porous tower in downstream direction.

Thus, F can be calibrated by matching $C_d A/A_f = FL$, where $C_d A/A_f$ is termed the hydrodynamic loading in the actuator disc theory [16]. Note that the theoretical model is concentrated in a disc, while the numerical porous model is distributed over a volume. See Taylor et al. [16] for the details of the actuator disc theory

Download English Version:

https://daneshyari.com/en/article/7156990

Download Persian Version:

https://daneshyari.com/article/7156990

Daneshyari.com