Applied Mathematics Letters 27 (2014) 79–84

Contents lists available at ScienceDirect

Applied Mathematics Letters

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

Free surface wave interaction with an oscillating cylinder

Canan Bozkaya [a], Serpil Kocabiyik [b,*]

[a] Department of Mathematics, Middle East Technical University, Ankara, Turkey

[b] Department of Mathematics and Statistics, Memorial University of Newfoundland, St. John's, NL, A1C 5S7, Canada

ARTICLE INFO

Article history:
Received 24 June 2013
Accepted 30 July 2013

Keywords:
Free surface
Cylinder
Forced oscillation
Wake dynamics
Lift force
Computation

ABSTRACT

The numerical solution of the special integral form of two-dimensional continuity and unsteady Navier–Stokes equations is used to investigate vortex states of a horizontal cylinder undergoing forced oscillations in free surface water wave. This study aims to examine the consequence of degree of submergence of the cylinder beneath free surface at Froude number 0.4. Calculations are carried out for a single set of oscillation parameters at a Reynolds number of $R = 200$. Two new locked-on states of vortex formation are observed in the near wake region. The emphasis is on the transition between these states, which is characterized in terms of the lift force on the cylinder and the instantaneous patterns of vortex structures and pressure contours in the near wake.

1. Introduction

The interaction of a free surface wave motion with moving cylindrical bodies has been principally the subject of experimental studies. Computations of nonlinear viscous free surface problems including cylindrical bodies are relatively few [1,2]. In these studies the free surface effects on the near wake development and vortex formation modes are investigated. Neither fluid force descriptions nor the link between fluid forces and the changes in wake structure is discussed. In this paper, a viscous incompressible two-fluid model with a circular cylinder is investigated numerically. The present two-fluid model involves the fluids in the regions Ω_1 and Ω_2 with densities ρ_1, ρ_2 and dynamic viscosities μ_1, μ_2 entering into the domain with uniform velocity U at the inlet and leaving through the outlet boundary. The circular cylinder of radius d is submerged in the fluid region Ω_2 at the distance h^* below the undisturbed free surface. Initially, an infinitely long circular cylinder whose axis coincides with the z-axis is at rest, and then, at time $t = 0$, the cylinder starts to perform streamwise oscillations about the x-axis. The imposed oscillatory cylinder displacement is assigned $x(t) = A\cos(2\pi ft)$. The aims of the present investigation are (i) to characterize the possible states of the wake and (ii) to establish a link between the changes in wake dynamics of the cylinder and lift force at three different submergence depths.

The relevant dimensionless parameters are the Reynolds number $R_2 = Ud/\nu_2$ ($R_1 = Ud/\nu_1$); the forcing amplitude of the cylinder oscillations, $A = A^*/d$; the frequency ratio, f/f_0, with $f = df^*/U$ and $f_0 = df_0^*/U$ being the dimensionless forcing frequency of the cylinder oscillation and the natural vortex shedding frequency for the corresponding stationary cylinder case in an unbounded medium; the cylinder submergence depth, $h = h^*/d$; and the Froude number, $Fr = U/\sqrt{dg^*}$. Here, $\nu_1 = \mu_1/\rho_1$, $\nu_2 = \mu_2/\rho_2$ are the kinematic viscosities of the fluids in Ω_1 and Ω_1, respectively, f^* is the dimensional forcing frequency of cylinder oscillation, f_0^* is the dimensional natural vortex shedding frequency of a stationary cylinder, g^* is the acceleration due to gravity, $\vec{g}^* = (0, g^*, 0)$, $t^* = td/U$ is the dimensional time, and t is the dimensionless time. The dimensionless fluid pressure, p, is defined by $p/\varepsilon = p^*/\rho_2 U^2$, where $\varepsilon = \rho_1/\rho_2$ when $\vec{x} \in \Omega_1$, and $\varepsilon = 1$ when $\vec{x} \in \Omega_2$.

* Corresponding author. Tel.: +1 709 864 8797.
E-mail addresses: bcanan@metu.edu.tr (C. Bozkaya), serpil@mun.ca (S. Kocabiyik).

0893-9659/$ – see front matter
http://dx.doi.org/10.1016/j.aml.2013.07.009

2. Numerical solution and validation summary

The special integral form of two-dimensional continuity and unsteady Navier–Stokes equations are solved in their primitive variables' formulation using an existing finite volume scheme. Detailed features of the numerical method and systematic validations have been outlined in [2], and only a brief description of points of direct relevance to the computations will be provided here; further details of the implementation and validations can be found in [2]. The governing equations are the two-dimensional continuity and the Navier–Stokes equations (when a solid body is present) given by

$$\frac{d\mathbb{V}}{dt} + \int_{\mathbb{A}} (\vec{u} \cdot \vec{n})\, dS = 0, \tag{1}$$

$$\frac{d}{dt}\int_{\mathbb{V}} \vec{u}\, dV + \int_{\mathbb{A}} (\vec{n} \cdot \vec{u})\vec{u}\, dS = -\frac{1}{\varepsilon}\int_{\mathbb{A}\cup\mathbb{I}} p\vec{n}\, dS + \frac{1}{R}\int_{\mathbb{A}\cup\mathbb{I}} \vec{n} \cdot \nabla\vec{u}\, dS + \int_{\mathbb{V}} \vec{F}\, dV \tag{2}$$

where $\mathbb{V}$ and $\mathbb{A}$ are the fractional volume and area, respectively, open to flow within the computational cell, V; $\mathbb{I}$ is the length of the fluid–body interface open to flow; $\vec{u}$ is the dimensionless velocity vector, where $\vec{u} = (u, v, 0)$; $\vec{n}$ is the outward unit normal vector; and S is the control volume boundary. These dimensionless quantities are defined in terms of their dimensional counterparts: $x = x^*/d$, $y = y^*/d$, $u = u^*/U$, $v = v^*/U$; $V = V^*/d^2$, $S = S^*/d$, $\mathbb{V} = \mathbb{V}^*/d^2$, $\mathbb{A} = \mathbb{A}^*/d$, $\mathbb{I} = \mathbb{I}^*/d$. The external force, $\vec{F} = (-a_1, 1/Fr^2 - a_2, 0)$, is due to the dimensionless gravity force, $\vec{g} = (0, 1/Fr^2, 0)$, and the dimensionless acceleration of the non-inertial frame of reference, $(-a_1, -a_2, 0)$. The boundary conditions are no-slip of the fluid on the cylinder surface, $u = 0$, $v = 0$; the uniform stream at the inflow, $u = U - v_1$, $v = -v_2$; and the free slip conditions at the top and bottom boundaries of the computational domain, $\partial u/\partial x = 0$, $v = -v_2$. The well-posed open boundary conditions, $\frac{1}{R}\partial u/\partial x + \bar{h}/Fr^2 = p$, $\partial v/\partial x = 0$, are enforced at the outflow boundary. Here, $\bar{h}$ is the height of the fluid at the outflow boundary. The uniform flow is used as the initial condition. It is assumed that at time $t = 0$, the free surface is undisturbed.

Finite volume discretization of the governing equations is performed based on the aggregated-fluid approach, using a single set of Eqs. (1)–(2) in the computational domain, $\Omega = \Omega_1 \cup \Omega_2$. The values of fluid properties are set to $\rho_1/\rho_2 = 1/100$ and $\mu_1/\mu_2 = 1/100$ (or $\nu_1/\nu_2 = 1$) following the work of Reichl et al. [1]. The Reynolds numbers in the fluid regions Ω_1 and Ω_2 are the same ($R \equiv R_1 = R_2$) which are varied by altering the viscosity μ (or ν). The main computational difficulty is solving the governing equations in an inertial frame of reference which results in pressure spikes. Professor Arthur E.P. Veldman's group at University of Groningen attempted to overcome this difficulty unsuccessfully for more than a decade (see e.g., [3]). In the present study, the computational difficulty is eliminated by employing a non-inertial frame of reference. The free surface interface is discretized with the volume-of-fluid method [4]. Its advection in time is performed based on the strictly mass conserving volume-of-fluid advection method for two-dimensional incompressible flows [5]. For the moving fluid–body interface the fractional area/volume obstacle representation method [6] and the cut cell method [7] are employed. A second-order accurate central-difference scheme is used to discretize the governing equations in space in conjunction with a first-order explicit forward Euler scheme to advance the numerical solution in time.

The computational grid geometry is defined with respect to the *mean* position of the cylinder and by specifying the locations of inflow and outflow boundaries, L_1 and L_2, along the x-axis and the location of the top and bottom boundaries, L_3, along the y-axis. The numerical simulations are carried out using the computational code which was developed by S. Kocabiyik's research group at Memorial University. Code verification and validation testing were conducted by L.A. Mironova and C. Bozkaya during their doctoral and postdoctoral studies at Memorial University, respectively. The numerical grid, $L_1 = 20$, $L_2 = 30$, $L_3 = 40$, with 60 cells per cylinder diameter and $\Delta t = 0.005$ are found to be satisfactory and are checked carefully. Tests are conducted in the absence of a free surface for a stationary cylinder case at $R = 200$. The calculated value of natural shedding frequency, $f_0 = 0.198$, is within 0.1% of the accepted value of 0.197 [8]. The predicted values of the mean drag coefficient, $\widehat{C_D} = 1.3399$, and the maximum lift coefficient, $C_{L,\max} = 0.70$, are in good agreement with the previous numerical results of De Palma et al. [9] ($\widehat{C_D} = 1.34$, $C_{L,\max} = 0.70$). The x- and y-components of the dimensionless force, $\vec{F} = 2\vec{F}^*/(\rho d U^2)$, exerted by the cylinder on the fluid are the dimensionless drag and lift force coefficients (C_D, C_L):

$$C_D = \int_0^{2\pi} p\cos\theta\, d\theta + \frac{1}{R}\int_0^{2\pi} \frac{\partial ur}{\partial\vec{n}}\, d\theta, \qquad C_L = \int_0^{2\pi} p\sin\theta\, d\theta + \frac{1}{R}\int_0^{2\pi} \frac{\partial v}{\partial\vec{n}}\, d\theta \tag{3}$$

where $\vec{n} = (\cos(\theta), \sin(\theta), 0)$ is the outward unit normal to the cylinder boundary. In Table 1, the effects of h on the predicted values of $Fr|_L$, $\bar{f}_0/f_0$, and $\widehat{u}$ at $R = 180$, $Fr = 0.4$, $h = 0.22, 0.55, 0.70$ are compared with the numerical results of Reichl et al. [1]. The u-velocity is averaged based on the free surface height, $h|_L$, in the region directly above the cylinder. The local Froude number, $Fr|_L$, is calculated using $Fr|_L = \bar{u}/\sqrt{(gh|_L)}$. Here $\bar{u}$ is the maximum u-velocity in the region directly above the cylinder at the time when the lift coefficient reaches its maximum. The predicted results are in excellent agreement with the numerical results of Reichl et al. [1].

3. Results and conclusions

The numerical simulations are carried out in the presence of a free surface at $h = 0.25, 0.5, 0.75$ and $Fr = 0.4$ for the case of $R = 200 : A = 0.13$ and $f/f_0 = 3.75$. Small amplitude excitation at $f/f_0 = 3.75$ is chosen outside the fundamental