8th. World Congress on Computational Mechanics (WCCM8)
5th European Congress on Computational Methods in Applied Sciences and Engineering (ECCOMAS 2008)
June 30 –July 5, 2008
Venice, Italy

NUMERICAL SIMULATION OF TWO-PHASE FLOW WITH AN OSCILLATING CIRCULAR CYLINDER BASED ON A VISCOUS INCOMPRESSIBLE TWO-FLUID MODEL

*** S. Kocabiyik[1], L. A. Mironova[2] and O. I. Gubanov[3]**

[1,2] Department of Mathematics and Statistics
Memorial University of Newfoundland
St. John's, NL, Canada A1C 5S7
serpil@math.mun.ca, larisa@math.mun.ca

[3] Department of Mechanical Engineering
McGill University
Montreal, QC, Canada H3A 2K6
oleg@cim.mcgill.ca

Key Words: *Incompressible, Viscous, Unsteady, Two-Fluid Model, Cylinder Oscillation, Computation.*

ABSTRACT

The unsteady hydrodynamic forces on a moving body and resulting free-surface deformations are of considerable importance in many engineering applications and theoretical interest. The interaction of a free surface wave motion with moving bodies has been principally the subject of experimental investigations[1,2]. Computations of nonlinear viscous free surface problems including cylindrical bodies are relatively few[3,4]. A literature survey reveals that there are no computational studies considering the problem of unsteady, laminar, two dimensional flow of a viscous incompressible fluid past a circular cylinder subject to streamwise oscillations in the presence of a free surface. The same problem is considered in the present study. The key to the numerical method used in this paper is the use of a single set of governing equations, in their special integral form, for the whole flow domain. The governing equations are the integral form of the well known fractional area/volume obstacle representation (FAVOR) equations with both a moving solid body and a free surface. The differential form of the FAVOR equations are originally introduced by Hirt and Sicilian[5] for an incompressible, inviscid fluid case when the solid body occupies a fraction of the computational domain. The method of solution is based on the fixed Cartesian grid finite volume method. A second-order accurate central-difference scheme is used to discretize the governing equations in space in conjunction with the first-order explicit forward Euler scheme to advance these equations in time. The cell merging procedure is utilized which allows to develop a spatial discretization scheme that preserves the global second-order accuracy. This done by merging the velocity cells cut by a fluid-body interface with the appropriate neighbour fluid cells to form the trapezoidal cells near the cylinder. A combination of a B-spline and polynomial interpolations is then used to evaluate convective and diffusive fluxes near the cylinder surface. The preliminary numerical experiments conducted in the present study shows that when the body moves through the fixed staggered grid (i.e., when the inertial frame of reference is considered) the pressure cell which belongs to the body at the time step $t = t^n$ may become a fluid cell at the next time step, $t = t^{n+1}$, and the continuity equation will be discretized there. Since at the time step $t = t^n$ the velocities in the pressure cell do not satisfy the mass balance exactly, the pressure field has to do extra work to restore the mass balance in the pressure cell at the time step $t = t^{n+1}$. This extra work seems to reflect as a spike in

the pressure. Fekken and Kleefsman *et al.*[6,7] attempted to overcome this difficulty but they failed to eliminate these pressure spikes. In the present model, the pressure spikes are eliminated by using of a non-inertial frame of reference. The free surface is discretized using the volume of fluid method (VOF) which is originally introduced by Hirt and Nichols[8]. The free surface is locally tracked in every cell by the VOF function, F, which represents the volume of fluid per unit volume ($0 \leq F \leq 1$) and satisfies a scalar transport equation. Since the free surface interface is free to move, its advection in time can be performed after the new velocity field has been calculated from the governing equations. A piecewise constant reconstruction of the free surface is used, where the free surface is displaced by altering the F values in the cell. In the work by Hirt and Nichols, the VOF method leads to the appearance of the flotsam and jetsam, which are small droplets disconnecting the flow at the free surface. Additionally, the gain/loss of the fluid volume may appear due to rounding of the F function. In the present numerical algorithm, the geometrical VOF method due to Scardovelli *et al.*[9] is adopted to overcome the above mentioned difficulties. The resulting advection algorithm ensures the implementation of the principle reconstruction constraint as the *exact mass conservation in two dimensional flows* (see Scardovelli *et al.*[8] for details). The numerical simulations are conducted at a fixed Reynolds number, $R = 200$, and at the fixed displacement amplitude-to-cylinder radius ratio of $A = 0.13$. Results show the existence of asymmetric modes of vortex formation in the cylinder wake at different values of unsteady loading on the cylinder, which is characterized by the ratio of excitation frequency, f, to the Kármán vortex shedding frequency, f_0. For this paper, the frequency ratio is chosen from $f/f_0 = 1$ to 3. The relation between these vortex shedding modes and fluid forces on the cylinder surface is discussed. Several test cases have been calculated using the present approach including uniform flow past a stationary cylinder in the presence of a free-surface. The results compare well with other experimental and numerical data.

REFERENCES

[1] O. Cetiner and D. Rockwell. " Streamwise oscillations of a cylinder in a steady current. Part 2: Free-surface effects on vortex formation and loading". *J. Fluid Mech.*, **427**, 29–59, 2001.

[2] J. Carberry. *Wake states of an oscillating cylinder and of a cylinder beneath a free-surface*, PhD Thesis, Monash University, Australia, 2002.

[3] P.J. Reichl, K. Hourigan and M.C. Thompson. " Flow past a cylinder close to a free surface". *J. Fluid Mech.*, **533**, 269–296, 2005.

[4] O. I. Gubanov *Design of CFD code using high level programming paragdigms: Free surface flows with arbirarily moving bodies*, MSc Thesis, Memorial University of Newfoundland, Canada, 2006.

[5] C. W. Hirt and J. M. Sicilian. " A porosity technique for the definition of obstacles in rectangular cell meshes ". *Proc. Forth Intl. Conf. on Ship Hydrodynamics*, Washington, District of Columbia, 1985.

[6] G. Fekken. *Numerical simulation of free-surface flow with moving rigid bodies.*, PhD Thesis, University of Groningen, The Netherlands, 2004.

[7] K. M. T. Kleefsman, G. Fekken, A. E. P. Veldman, B. Iwanowski and B. Buchner. " A Volume-of-Fluid based simulation method for wave impact problems ". *J. Comp. Phys.*, **206**, 363–393, 2005.

[8] R. Scardovelli, E. Aulisa, S. Manservisi and S. Zaleski. " A geometrical area-preserving volume-of-fluid advection method ". *J. Comp. Phys.*, **192**, 355–364, 2003.

[9] C.W. Hirt and B.D. Nichols. Volume of fluid method for the dynamics of free boundaries. *J. Comput. Phys.*, **39**, 201–225, 1981.