International Journal of Offshore and Polar Engineering
Vol. 14, No. 3, September 2004 (ISSN 1053-5381)

Numerical Investigation of Wave Interaction with a Fixed Vertical Circular Cylinder

P. Queutey, M. Visonneau and P. Ferrant*
CNRS-UMR 6598, Laboratoire de Mécanique des Fluides, Ecole Centrale de Nantes, France

ABSTRACT

The interaction between a vertical circular cylinder and regular waves is studied on a numerical basis under conditions defined in the EXPRO-CFD European Project (Gallagher, 2003). Computed data are obtained from either a coupled method between a Reynolds-averaged Navier-Stokes (NS) solver and a nonlinear inviscid solver or an improved multifluid approach developed in the Navier-Stokes solver. Also presented are first comparisons to experiments conducted by ECN and Sirehna.

INTRODUCTION

The interaction between a vertical circular cylinder and regular waves is studied on a numerical basis. During the past decade a number of numerical wave tanks has been developed to reproduce fully nonlinear effects observed in physical wave basins (Park and Kim, 1998). Conditions of simulations are those defined by the EXPRO-CFD European Project and concern low and high steep waves ranging from $L/2a = 20$ to 10 and $L/D = 2.5$ to 10, where L is the incident wave length, a the wave amplitude and $D = 0.508$ m the diameter of the cylinder. Numerical simulations are performed using a Navier-Stokes method (NS) developed by DMN (Division Modélisation Numérique) and a nonlinear inviscid solver (NLIS) from the DHN (Division Hydrodynamique Navale), both belonging to the CNRS-UMR research unit No. 6598 at Ecole Centrale de Nantes. Computed data are obtained from either a coupled method between the NS and the NLIS methods, or an improved multifluid approach developed in the NS solver.

NONLINEAR INVISCID SOLVER

The NLIS solver (Ferrant, 1998) rests on a potential flow method for fully nonlinear free-surface flows. The numerical solution of the fully nonlinear initial-boundary-value problem is based on the so-called mixed Eulerian-Lagrangian (MEL) approach. Starting from initial conditions, a 2-step procedure is followed: First, a mixed Dirichlet-Neumann boundary value problem is solved for the normal velocity and for the velocity potential on the free surface; then, free-surface elevation and the potential on the free surface are advanced in time using the kinematic and dynamic free-surface conditions as ODEs.

URANSE FLOW SOLVER

The NS flow solver uses the incompressible, unsteady Reynolds-averaged Navier-Stokes equations (URANSE). The solver is based on the finite volume method to build the spatial discretisation of the transport equations (Duvigneau, 2003). The face-based method is generalized to 2-dimensional, rotationally symmetric, or 3-dimensional unstructured meshes for which nonoverlapping control volumes are bounded by an arbitrary number of constitutive faces.

The velocity field is obtained from the momentum conservation equations and the pressure field is extracted from the mass conservation constraint, or continuity equation, transformed into a pressure equation.

In the case of turbulent flows, additional transport equations for modeled variables are solved in a way similar to that of the momentum equations. Some of the near wall low-Reynolds number turbulence models implemented in the NS solver are used in this context, ranging from 1-equation Spalart-Allmaras model (Spalart and Allmaras, 1992), to 2-equation K-ω closures (Wilcox, 1993). The *DES* model (Travin et al., 2000) can also be used if the statistical modelization of turbulence appears inaccurate to describe the large unsteady structures of the flow.

A 2$^{\text{nd}}$-order accurate 3-level fully implicit time discretisation is used, and surface and volume integrals are evaluated using 2$^{\text{nd}}$-order accurate approximations.

Incompressible and nonmiscible flow phases are modeled through the use of conservation equations for each volume fraction c_i of each phase:

$$\frac{\partial}{\partial t}\int_V c_i dV + \int_S c_i \vec{U}\cdot\vec{n}\, dS = 0 \tag{1}$$

where V is the domain of interest, or control volume, bounded by the closed surface S with a unit normal vector $\vec{n}$ directed outward, and $\vec{U}$ is the flow velocity. Then, the effective flow physical properties (viscosity and density) are obtained from each phase physical properties (μ_i and ρ_i) with the following constitutive relations:

$$\rho = \sum_i c_i\rho_i; \quad \mu = \sum_i c_i\mu_i; \quad 1 = \sum_i c_i \tag{2}$$

Interfacial reconstruction of c_i that defines the interface capturing scheme is based on the Gamma Differencing Scheme (Jasak, 1996). This scheme is modified in such a way that compressive effects are introduced through downward differencing, which is an acceptable compromise between accuracy and boundedness of the numerical solution for c_i, even with arbitrarily unstructured meshes. However, a major drawback is a limitation on the Courant number (typically lower than 0.3).

If the pressure-velocity coupling is performed by a segregated SIMPLE-like algorithm based on the classical Rhie and Chow

*ISOPE Member.

Received March 1, 2004; revised manuscript received by the editors July 19, 2004. The original version was submitted directly to the Journal.
KEY WORDS: Wave, cylinder, nonlinear inviscid, Navier-Stokes, turbulence, interface tracking, interface capturing.