11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
E. Oñate, J. Oliver and A. Huerta (Eds)

CUT-CELL METHOD: APPLICATION TO WATER WAVES GENERATED BY A SUBMERGED OBSTACLE.

J. Dambrine[1], N. James[1] and G. Rousseaux[2]

[1] Université de Poitiers - Laboratoire de Mathématiques et Applications UMR 7348, Futuroscope Chasseneuil France, Firstname.Name@math.univ-poitiers.fr [2] Axe HydEE, Université de Poitiers, Département d'Aérodynamique ISAE, ENSMA, UPR CNRS 3346, Institut P Prime, Futuroscope Chasseneuil France, Germain.Rousseaux@univ-poitiers.fr

Key words: Immersed boundary methods, Parallel computing, Multiphase flows.

Abstract. This study addresses a problem of wave and current interactions. Experiments performed in a wave channel in which a flow is imposed over an obstacle have shown very interesting features such as wave blocking and the generation of blue shifted waves. These experiments can be used as analogue models of gravity in order to understand some phenomena such as the Hawking radiation of black holes.

The linear theory of gravity waves, based on the assumption of an irrotational fluid can explain some aspects of the phenomenology. However, in order to fully understand the propagation of waves in such a context, it is necessary to use more complex models. We hence focused our research on the numerical simulation of the free-surface Navier-Stokes equations in the presence of obstacles.

Several techniques, avoiding the generation of conformal meshes, will be used to take into account the presence of both moving interface and obstacle in a two dimensional incompressible fluid flow. In [1], an original cut-cell method was developed to preserve the second order accuracy of the MAC scheme when enforcing Dirichlet boundary conditions on an obstacle of arbitrary shape. In the present paper, the cut-cell method is extended to treat two-phases incompressible flows passing over a submerged rigid obstacle. The tracking of the interface Γ between the two phases (water and air) is achieved using the finite difference technique [5] which include the effects of surface tension and do not smear out the interface. We focus on the numerical simulation of current-obstacle interactions in a water channel.

1 Introduction

We consider the simulation of an incompressible flow composed of two immiscible liquid phases. The equations of motion of such heterogeneous incompressible viscous flow are

the Navier-Stokes equations:

$$\begin{aligned} \rho\left(\partial_t \mathbf{u} + (\mathbf{u}.\nabla)\,\mathbf{u}\right) &= \mu\triangle\mathbf{u} - \nabla p + \mathbf{f} && (1)\\ \nabla.\mathbf{u} &= 0 && (2)\\ \partial_t\rho + (\mathbf{u}.\nabla)\,\rho &= 0 && (3) \end{aligned}$$

where the mass density $\rho(\mathbf{x},t) \in \mathbb{R}$, the velocity $\mathbf{u}(\mathbf{x},t) \in \mathbb{R}^2$ and the pressure $p(\mathbf{x},t) \in \mathbb{R}$ are the unknowns at time t and position $\mathbf{x}$. We have respectively denoted by $\mathbf{f}(\mathbf{x},t) \in \mathbb{R}^2$ and μ, a body force field and the constant dynamic viscosity. The velocity across the interface Γ is continuous $[\mathbf{u}] = 0$ and the stress jump condition reduce to $[p] = -\sigma\kappa$, where γ is the coefficient of surface tension and κ is the local curvature of the interface.

Several numerical approaches have been introduced in the literature for multiphase flows, to accurately solve the equations with sharp interfaces. Let us mention Arbitrary Lagrangian-Eulerian methods [10], Level-Set approach [9], Volume-Of-Fluid approach [2] and diffuse interface methods [8].

Even if we expect to get smooth interfaces, breaking waves can occur in the water channel. Therefore, the level set method, originally introduced by Osher and Sethian [7], is well adapted for tracking the evolution of the interface between the two phases. Indeed, no special procedures are required in order to model topological changes of the front. The interface is represented as the zero level set of the function ϕ which satisfy:

$$\partial_t\phi + \mathbf{u}.\nabla\phi = 0 \tag{4}$$

Each phase being identified by a constant mass density, namely ρ_1 or ρ_2, the location of each liquid phase is tracked using the heaviside function H defined by $H(\phi) = 1$ if $\phi \geq 0$ and 0 otherwise, so that $\rho = \rho_1 + (\rho_2 - \rho_1)H$ is the variable-density. In order to ensure that ϕ stay a signed distance function, i.e. $||\nabla\phi|| = 1$, the reinitialization equation

$$\partial_\tau\phi + S(\phi_0)(||\nabla\phi|| - 1) = 0 \quad \text{, with} \quad S(\phi_0) = \frac{\phi_0}{\sqrt{{\phi_0}^2 + h^2}} \tag{5}$$

is iterated in virtual time τ, where ϕ_0 is the initial ϕ and h is the grid cell size. Then, convenient formulas for the interface normal and curvature can be used.

In complex geometries, the discretization of the Navier Stokes Equation by finite element or finite volume methods need to generate conformal meshes which is a challenging problem when the geometry gets complex. Indeed, generating a body-fitted grid can be even more expansive than computing the solution itself. An alternative is proposed with the Immersed Boundary (IB) methods. The aim of IB methods is to handle complex geometric configurations without the use of body-fitted meshes. Simulations are performed on cartesian grids so that the efficiency and robustness of cartesian grid solvers are achieved.

The numerical scheme proposed in [1] is based on the well-known second-order projection MAC scheme (see [12]). As in [13], the immersed boundary is geometrically represented by using the signed algebraic distance to the obstacle boundary. In fluid-cells,

that is mesh cells which are far enough from the immersed boundary, classical centered, second-order finite volume schemes are used. In this approach, the location of the velocity component is, as in [13], adapted to the geometry of cut-cells. However, the discrete pressure is placed at the center of the cartesian cells for both fluid-cells and cut-cells. In the vicinity of the obstacle, second-order interpolations using boundary conditions on the solid boundaries are introduced to evaluate the convective fluxes. This results in a local first-order approximation of the nonlinear terms in cut-cells. A pointwise approximation of the viscous terms is used in cut-cells. When boundary conditions on the immersed boundary can be used, a five-point stencil scheme for the viscous term is employed. Otherwise, a six-point first-order approximation is introduced. The resulting linear system is close to the five-point structure symmetric system obtained on cartesian mesh with the MAC scheme. A direct solver, based on a capacitance matrix method, is proposed. The efficiency of the solver is similar to the cartesian grid solver obtained with the MAC scheme. The incompressibility of the discrete velocity field is enforced up to the computer accuracy. While first-order truncation errors are locally introduced in the scheme in the cut-cells, a second-order global accuracy is recovered.

2 Numerical method

Let δt stand for the time step and $t_k = k\delta t$ discrete time values. Like in [11], a variable density projection method is employed for solving the system (1)-(2)-(3) :

(a) Transport the level set function from time t_k to t_{k+1} : the moving interface is tackled with the Level-Set technique (fifth order WENO scheme).

(b) Using $\frac{1}{2}(\phi^k + \phi^{k+1})$, determine the density repartition at intermediate time $t_{k+\frac{1}{2}}$.

(c) Predict the velocity at intermediate time $t_{k+\frac{1}{2}}$ by using the previous pressure field and the updated interface location.

$$\rho^{k+\frac{1}{2}}\left(\frac{\mathbf{u}^{k+\frac{1}{2}}-\mathbf{u}^{k}}{\delta t}+\left(\mathbf{u}^{k+\frac{1}{2}}.\nabla\right)\mathbf{u}^{k+\frac{1}{2}}\right)=\frac{\mu^{k+1}\triangle\mathbf{u}^{k+\frac{1}{2}}+\mu^{k}\triangle\mathbf{u}^{k}}{2}-\nabla p^{k-\frac{1}{2}}+\mathbf{f}^{k+\frac{1}{2}} \quad (6)$$

where the convective terms are treated with the Adams-Bashforth scheme :

$$\left(\mathbf{u}^{k+\frac{1}{2}}.\nabla\right)\mathbf{u}^{k+\frac{1}{2}}=\left(3\left(\mathbf{u}^{k}.\nabla\right)\mathbf{u}^{k}-\left(\mathbf{u}^{k-1}.\nabla\right)\mathbf{u}^{k-1}\right)/2$$

(d) Solve the variable coefficient Poisson equation on the pressure increment δp^{k+1} where both the variable coefficient and the solution itself may be discontinuous.

$$\nabla.\left(\frac{\nabla(\delta p^{k+1})}{\rho^{k+\frac{1}{2}}}\right)=\frac{1}{\delta t}\nabla.\mathbf{u}^{k+\frac{1}{2}} \quad (7)$$

(e) The resulting pressure is used to project the intermediate velocity field onto the space of divergence-free velocity field in order to obtain the velocity field at time t_{k+1}.

$$\mathbf{u}^{k+1} = \mathbf{u}^{k+\frac{1}{2}} - \delta t \frac{\nabla(\delta p^{k+1})}{\rho^{k+\frac{1}{2}}} \tag{8}$$

(f) Update the pressure field at time t_{k+1}.

$$p^{k+\frac{1}{2}} = p^{k-\frac{1}{2}} + \delta p^{k+1} \tag{9}$$

A standard MAC grid is used where p, ρ, and ϕ exist at the cell centers, while u and v are located at the appropriate cell edges. Away from both moving interface and obstacle, linear partial differential operators (divergence, gradient, Laplacian operator) and nonlinear terms are discretized on a fixed Cartesian grid by using standard second order Finite Differences approximations.

The main challenge here is to take into account the boundary conditions of the problem. The Immersed Boundary Method [1] is used in order to enforce the no-slip boundary condition on the rigid obstacle within the prediction step (6) and also for the discretization of both gradient and divergence operators in (7) and (8). In (6), we simply use the sign of the level set function to determine μ (resp. ρ) as the air or water viscosity (resp. density) in a sharp fashion. In particular, both μ and ρ are spatially constant on either side of the interface. The techniques presented in [4] for the variable coefficient Poisson equation are used to solve the equation (7) for the pressure. In addition, the pressure jump across the interface is accounted for in the same time. As it was mentioned in [5], one should take care to compute the derivatives of the pressure in (8) in exactly the same way as they were computed in (7). Many direct and iterative approaches have been employed to find the solution of the linear systems (6) and (7). For large problems, the faster solver we have found is an algebraic multigrid method (HYPRE BoomerAMG). The resolution of the linear systems has been implemented in parallel using the PETSc Fortran library.

3 Preliminary results

In this section we present a validation of our methods by focusing on the dispersive aspects of the waves generated at the interface by an initial perturbation. We consider the case where the domain is given by a $2m \times 0.5m$ box. In order to approximate the case of water waves we take the following values for the density:

$$\rho_1 = 1000\ kg.m^{-3}\,, \tag{10}$$

$$\rho_2 = 10\ kg.m^{-3}\,. \tag{11}$$

The viscosity is the same in the two phases and taken as: $10^{-3}\ Pa.s$. The initial height of the interface (plotted on figure 2) is given by:

$$h(x) = h_0 + 0.1\, e^{-10|x|}\,, \tag{12}$$

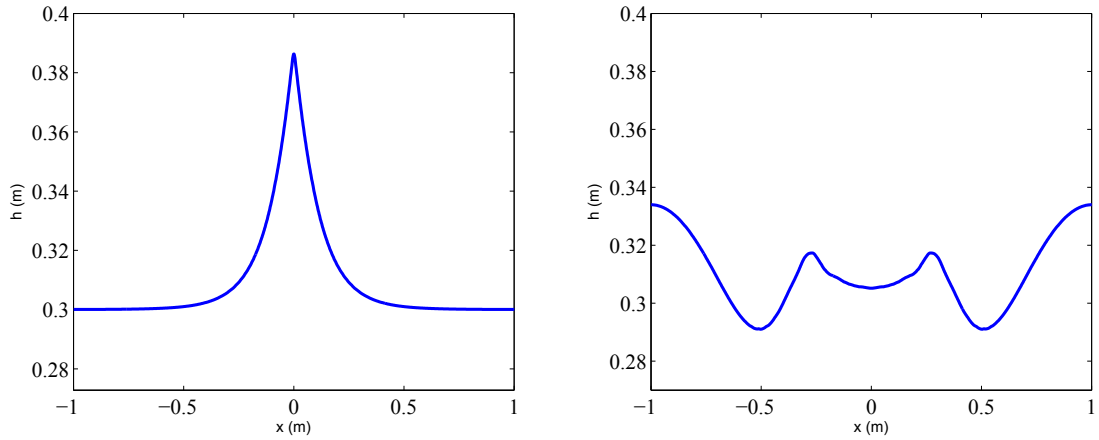

Figure 1: Right : plot of the initial water height profile ; left : the height profile after some time

where : $h_0 = 30cm$. By running a simulation on a 1024×512 grid with the above parameters we expect to recover the dispersion relation for water waves. In order to do this, we extract the water column height $H(x,t)$ from the level set functions produced by the simulation (see figure). By examining the properties of the square modulus of the fourier modes of H (the spectral power density) we expect to observe the locus of the wave numbers and frequencies that satisfy the dispersion relation. In the case of gravity waves in a two layer fluid (*i.e.* fluids with different densities), the dispersion relation is given by:

$$\omega^2 = \frac{g\,k\,(\rho_1 - \rho_2)}{\rho_1 coth(k\,h_1) + \rho_2 coth(k\,h_2)}\,. \tag{13}$$

In the case of $\rho_1 >> \rho_2$ and $k\,h_1 >> 1$, we obtain the classical dispersion relation for water waves in deep water:

$$\omega^2 = g\,k\,. \tag{14}$$

The figure shows the good agreement of the theory and the numerical simulations for the dispersion of gravity waves.

REFERENCES

[1] F. Bouchon, T. Dubois and N. James, A second-order cut-cell method for the numerical simulation of 2D flows past obstacles, *Computers & Fluids*, 80–91, 2012.

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

[3] A.J. Chorin, Numerical Solution of the Navier-Stokes Equations", *Mathematics of Computation*, **22**, 745–762, 1968.

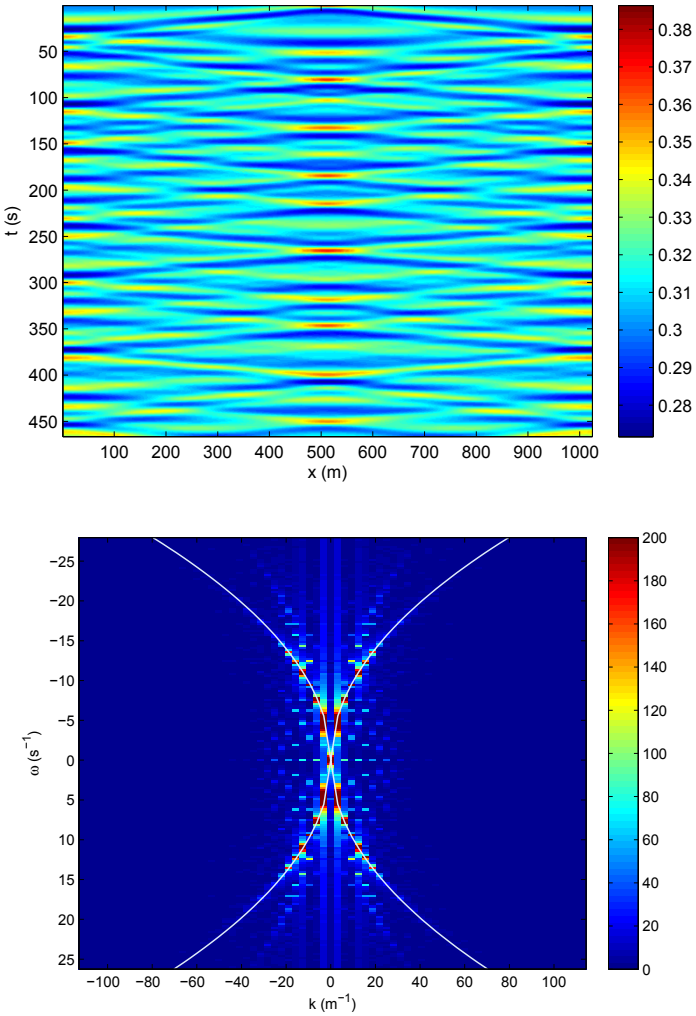

Figure 2: Upper figure: plot of the height of the interface (in color) as a function of x and t ; lower figure: color of the spectral power density (in color) as a function of the wave number and the frequency. In white we plotted the theoretical dispersion relation.

[4] X.D. Liu, R. Fedkiw and M. Kang, A Boundary Condition Capturing Method for Poisson's Equation on Irregular Domains, *J. Comput. Phys.*, **160**, 151–178, 2000.

[5] M. Kang, R. Fedkiw and X.D. Liu, A Boundary Condition Capturing Method for Multiphase Incompressible Flow, *Journal of Scientific Computing*, **15** 3, 323–360, 2000.

[6] S. Balay, J. Brown, K. Buschelman, V. Eijkhout, W.D. Gropp, D. Kaushik, M.G. Knepley, L. Curfman McInnes, B.F. Smith and H. Zhang, PETSc Users Manual,

ANL-95/11 - Revision 3.4, Argonne National Laboratory, 2013.

[7] S. Osher and J.A. Sethian, Fronts Propagating with Curvature-Dependent Speed: Algorithms Based on Hamilton–Jacobi Formulations, *J. Comput. Phys.*, **79**, 12–49, 1988.

[8] Y. Kim and C.S. Peskin, Numerical study of incompressible fluid dynamics with nonuniform density by the immersed boundary method, *Phys. Fluids*, 2008.

[9] Y.C. Chang, T.Y. Hou, B. Merriman and S.A. Osher, Level set formulation of Eulerian interface capturing methods for incompressible fluid flows, *J. Comput. Phys.*, 449–464, 1996.

[10] J.F. Gerbeau, T. Lelièvre and C. Le Bris, Simulations of MHD flows with moving interfaces, *J. Comput. Phys.*, 163–191, 2003.

[11] A.S. Almgren, J.B. Bell, D.L. Marcus, W.J. Rider and E.G. Puckett, A High-Order Projection Method for Tracking Fluid Interfaces in Variable Density Incompressible Flows, *J. Comput. Phys.*, 1997.

[12] F.H. Harlow and J.E. Welch, Numerical calculation of time-dependent viscous incompressible flow of fluid with free surface, *Phys. Fluids*, 2182–2189, 1965.

[13] Y. Cheny and O. Botella, The LS-STAG method: A new immersed boundary/level-set method for the computation of incompressible viscous flows in complex moving geometries with good conservation properties, *J. Comp. Phys.*, 1043–1076, 2010.

[14] T. Ye, R. Mittal, H.S. Udaykumar and W. Shyy, Numerical Simulation of two-dimensional flows over a circular cylinder using the immersed boundary method, *J. Comp. Phys.*, 209–240, 1999.