A level set technique for computing 2D free surface flows

Mathias Vogt

Dept. of Naval Architecture and Ocean Engineering, Chalmers University of Technology,Sweden

Kuk-Jin Kang

Korea Research Institute of Ships and Ocean Engineering, Korea

Introduction

The free surface boundary conditions in viscous flow problems can be used in conjunction with basically two different kinds of grid arrangements at the surface. In the fixed grid approach the free surface is tracked and the cells through which the free surface pass include fractions of both fluids. In the moving grid approach a curvilinear grid is fitted to the free surface at every time step. A drawback with the moving grid method is its inability of handling breaking and merging. Due to this shortcoming a new fixed grid method, the level set technique, Sussman et al. (1993), is here used to solve the Navier-Stokes equations for the flow over a bottom bump and around a submerged hydrofoil. Results from numerical simulations with the level set technique are compared to results from moving grid calculations, Kang (1996) and experiments.

Level set method

The level set is a scalar function defined in both fluids with opposite signs in the two fluids. Each level is a subset of the level set function and the subset with a value of zero, the zero level set, is here the free surface. Initially, the function is set equal to the distance from the interface and for later times the value is obtained by setting its material derivative equal to zero. Depending of the sign of the level set function the density and the viscosity are given appropriate values. To smooth the jump at the interface the physical properties are smoothed in a band around the zero level set. One nice feature with this technique is that the interface does not have to be found explicitly but is stored in the information of the level set function. It is also straightforward to extend the formulation to three dimensions.

Simulations presented elsewhere include: a rising air bubble in water, a falling water drop in air and a water drop hitting a pool of water, Sussman et al. (1993). Zhou and Chomiac investigated the Rayleigh-Taylor instability occurring when a light fluid supports or accelerates a heavier one.

Numerical formulation

Equations of motion

The equations of motion are given by the dimensionless Navier-Stokes equations and the incompressible continuity equation. These are solved with the equation for the level set function, ϕ and the following system of equations is obtained

$$\bar{u}_t = -\bar{u}\cdot\nabla\bar{u} + \frac{\zeta^2\nabla^2\bar{u}}{Re\zeta^1} - \frac{\nabla\psi}{\zeta^1}$$

$$\nabla\cdot\bar{u} = 0$$

$$\phi_t = -\bar{u}\cdot\nabla\phi$$

where

$$\psi = p - \frac{y}{Fn^2}$$

$$Fn = \frac{U_0}{\sqrt{gL}}, \qquad Re = \frac{\rho_w \cdot LU_0}{\mu_w}$$

$$\zeta^i = \begin{cases} 1 & if \quad \phi < -\alpha \\ \zeta_a^{\ i}/\zeta_w^{\ i} & if \quad \phi > \alpha \\ \bar{\zeta}^i - \Delta\zeta^i \sin\left(\frac{\pi\phi}{2\alpha}\right) & otherwise \end{cases}$$

$$\bar{\zeta}^i = (\zeta_a^{\ i} + \zeta_w^{\ i})/(2\zeta_w^{\ i}), \qquad \Delta\zeta^i = (\zeta_w^{\ i} - \zeta_a^{\ i})/(2\zeta_w^{\ i})$$

and $\bar{u} = (u, v)$ is the velocity, p is the pressure, U_o is the uniform flow velocity, L the chord length of the hydrofoil or the channel depth in the bottom bump case and g is the gravitational acceleration. ζ_a^i and ζ_w^i are the density (i=1) and dynamic viscosity (i=2) for air and water, respectively. α is half the prescribed width of the band where the physical properties change.

Solution procedure

The fluid domain is discretized by a finite-volume formulation and the velocity and pressure are defined on a staggered grid system. The method used to update the velocity and pressure is a time splitting fractional step method combined with a velocity and pressure simultaneous iteration method. Convection terms are approximated by a third order upwind scheme and other spacial derivatives are discretized by second order differences, see Kang (1996). When the velocity has been computed within a time step the level set equation is solved. This moves the free surface according to the computed velocity field. Before updating the density and the viscosity the level set function is reinitialized by iterating

$$\phi_t = 1 - |\nabla\phi|$$

to steady state. This ensures that the gradient of the level set function is one, which means that the bandwidth is constant in time.

Boundary conditions

For the level set function the Neumann condition is used at the inflow boundary and at the outflow boundary and linear extrapolation is used at the bottom boundary and at the top boundary. On bodies the no-slip condition, $\bar{u} = \bar{0}$ is imposed, i.e. it is used at the bottom in the bottom bump case and on the foil surface in the submerged hydrofoil case. In the moving grid approach boundary conditions are applied on the free surface, Kang (1996). A uniform flow, $u = 1$, $v = 0$ and zero pressure, $p = 0$ is used as boundary condition at the bottom in the hydrofoil case and when using the level set technique also at the top surface. The same conditions are imposed at the inflow boundary. At the outflow the velocity and the pressure are linearly extrapolated. To avoid reflections of waves at the outflow boundary the level set function is damped with an artificial wave damping function γ in a damping zone as follows;

$$\phi_t = -\bar{u} \cdot \nabla\phi - \gamma(x)\,\phi|_{y=0}$$

where

$$\gamma(x) = \begin{cases} A\left(\frac{x - x_d}{x_o - x_d}\right)^2 & if \ x_d \le x \le x_o \\ 0 & otherwise \end{cases},$$

A is a constant, x_o is the x-coordinate at the outflow boundary and x_d is defined as

$$x_d = x_o - 2\pi Fn^2.$$

Since the initial free surface not necessarily coincides with a node the level set function at $y = 0$, $\phi|_{y=0}$ is calculated with linear interpolation. Both the level set function at $y = 0$ and the damping function are calculated once and stored.

Numerical examples

Two different cases have been numerically studied. A submerged NACA 0012 hydrofoil and a bottom bump with the topography described by

$$y = \frac{27E}{4l^3}x(x-l)^2$$

where E denotes the maximum height of the bump, l its length and x is the distance from the leading edge. Initially the flow was accelerated sinusoidally one time unit. The grid used for the bottom bump case was a single-block grid and for the hydrofoil case a two-block H-grid. Computations were carried out for different Froude numbers and the results are in good agreement with the measurements for both methods of determining of the free surface. Results for the foil case are shown in fig. 1. For low enough Froude numbers the generated wave downstream of the bottom bump starts to overturn and break. This phenomenon can be captured with the level set technique, see fig. 2. The result is only qualitative since the resolution is insufficient, but it shows that this method is capable of handling changes in topology.

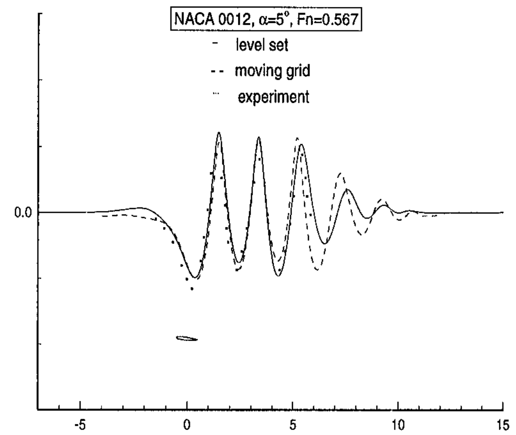

Fig 1. Comparison between numerical methods and experiment.

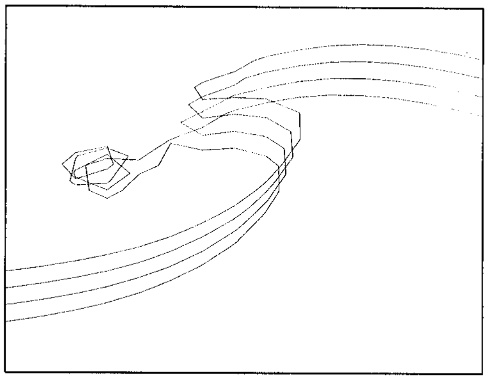
Fig 2. Overturning wave and the break-off of a water drop predicted by the level set technique. Four time steps are shown.

References

Sussman M., Fatemi E., Smareka P. (1993) "A level set approach for computing solutions to incompressible two-phase flow" *J. of comp. Phys.,* vol 114, nr 1, 146-159

Zhou G. & Chomiak J.(1995) "Numerical simulation of three-dimensional compressible/incompressible Rayleigh-Taylor instabilities of stationary and propagating density interface" *Numerical methods in laminar and turbulent flow '95*, vol 9, part 2, 1281-1293

Kang K-J. (1996) "Numerical simulation of nonlinear waves about a submerged hydrofoil" WWWFB '95, Hamburg

DISCUSSION

Wu T.Y.: I would like to encourage the authors to further develop this interesting method for it seems to provide a great deal of potential to future applications to stratified flow motions. In this regard I wish to know:

1) Have the authors applied this method to evaluate the interactions between sheared wind and ocean waves?

2) Have you seriously pursued an error estimate of your computational method?

3) Compared to your levelset equation $d\phi/dt = 0$, which is physically clear upon neglecting mass diffusion, your reinitiating condition seems artifical, empirical at best. To diffuse sharp transitions accross an interface for facilitating computations is fine but don't throw away its capability of predicting, e.g., when a sheared wind separates from an ocean wave.

Vogt M., Kang K-J.:

1) No

2) No

3) The numerical method cannot handle the steep gradients in the density that is present when the two fluids are water and air. The physical properties are therefore smoothed in a band around the interface. This is unphysical unless the band is extreemely thin. The reinitialization procedure itself does nothing but sets the gradient of the levelset function in the normal direction of the interface to one, without changing the position of the interface.