Abstract submitted to the Eleventh International Workshop on Water Waves and Floating Bodies

Computation of Free-Surface Flows Using Moving Grids

S. Muzaferija, M. Perić and S.-D. Yoo
Institut für Schiffbau, Universität Hamburg, Germany

Introduction

In this paper we propose a new method for solving the free-surface flow problems. The conservation equations for mass, momentum and space [1] are discretized using finite volume method with a colocated variable arrangement on block-structured or unstructured grids. The grid fits the free surface and moves with it. The integration in time is performed using the implicit Euler (first order) or three time levels (second order) schemes. Spatial discretization is based on linear interpolation and central differences (second order). An iterative solution method is used to solve the set of coupled non-linear equations. Linearized equations for Cartesian velocity components, pressure correction, and free surface location are solved in turn. Within each time step this sequence, called outer iteration, is repeated until the non-linear equations and all boundary conditions are satisfied. Linear equation systems are solved using a pre-conditioned conjugate gradient type of solver. The iterations performed within the linear equation solver are termed inner iterations.

The dynamic boundary condition at the free surface is taken into account when solving the momentum equations. At this stage the free surface shape and the pressure at it are prescribed (values from the previous outer iteration). The free surface pressure is not corrected in the pressure-correction equation, which is derived from the mass conservation equation; however, the velocities at the free surface are corrected. As a result, the discretized continuity equation is satisfied both globally and in each control volume, but the kinematic boundary condition is not satisfied: there are non-zero mass fluxes through the free surface. These mass fluxes are then compensated by displacing the free surface, i.e. the flow through the free surface is prevented by moving the free surface. At the end of each time step the mass conservation is satisfied in all control volumes and the kinematic condition at the free surface is also satisfied. The method is suitable for solving both steady and unsteady flow problems with free surface. Since it is implicit, large time steps can be used when solving steady flow problems.

Mathematical Formulation and Boundary Conditions

The conservation equations in integral form for space, mass and momentum, for a spatial region of volume Ω bounded by a closed surface S, read:

$$\frac{\mathrm{d}}{\mathrm{d}t}\int_{\Omega}\mathrm{d}\Omega - \int_{S}\mathbf{v}_s\cdot\mathbf{n}\,\mathrm{d}S = 0\,, \tag{1}$$

$$\frac{\mathrm{d}}{\mathrm{d}t}\int_{\Omega}\rho\,\mathrm{d}\Omega + \int_{S}\rho(\mathbf{v}-\mathbf{v}_s)\cdot\mathbf{n}\,\mathrm{d}S = 0\,, \tag{2}$$

$$\frac{\mathrm{d}}{\mathrm{d}t}\int_{\Omega}\rho\mathbf{v}\,\mathrm{d}\Omega + \int_{S}\rho\mathbf{v}(\mathbf{v}-\mathbf{v}_s)\cdot\mathbf{n}\,\mathrm{d}S = \int_{S}\boldsymbol{T}\cdot\mathbf{n}\,\mathrm{d}S + \int_{\Omega}\mathbf{b}\,\mathrm{d}\Omega\,. \tag{3}$$

Here ρ is the fluid density, $\mathbf{v}_s$ is the velocity of the boundary of a control volume (CV), $\mathbf{v}$ is the fluid velocity, $\mathbf{n}$ is the outward unit normal vector at the surface, $\boldsymbol{T}$ represents the stress tensor and $\mathbf{b}$ stands for body forces. The stress tensor is defined as

$$\boldsymbol{T} = \mu\left(\mathrm{grad}\,\mathbf{v} + \mathrm{grad}\,\mathbf{v}^T\right) - p\boldsymbol{I}\,, \tag{4}$$

where p is the static pressure, μ is the dynamic viscosity of fluid, and $\boldsymbol{I}$ is the unit tensor. On the free surface both kinematic and dynamic conditions must be satisfied:

$$[(\mathbf{v} - \mathbf{v}_s) \cdot \mathbf{n}]_{fs} = 0 \quad \text{or} \quad \dot{m}_{fs} = 0 \,, \tag{5}$$

where $\dot{m}_{fs}$ is the mass flux through the surface, and:

$$\begin{aligned} (\mathbf{n} \cdot \boldsymbol{T}) \cdot \mathbf{n}|_l &= -(\mathbf{n} \cdot \boldsymbol{T}) \cdot \mathbf{n}|_g \,; \\ (\mathbf{n} \cdot \boldsymbol{T}) \cdot \mathbf{t}|_l &= (\mathbf{n} \cdot \boldsymbol{T}) \cdot \mathbf{t}|_g \,, \end{aligned}$$

where $\mathbf{t}$ is a unit vector tangential to the free surface and indices l and g denote liquid and gas, respectively. At inlet the velocity and free surface height are given. At outlet, wave transmissive conditions are used. No slip condition on walls is used for viscous flow simulation and the slip condition is applied in the case of inviscid flow simulation. In case of a turbulent flow, additional scalar equations (e.g. for the turbulent kinetic energy and its dissipation rate) need to be solved.

Discretization Procedure

The solution domain at time t_0 is subdivided into a finite number of CVs by a block-structured or unstructured grid. The flow regions with large variable changes can be locally refined. The conservation equations are applied to each CV. The surface and volume integrals are approximated using the midpoint rule:

$$\int_{S_c} f \, \mathrm{d}S \approx f_c S_c \,, \quad \int_{\Omega} f \, \mathrm{d}\Omega \approx f_C \, \Delta\Omega \,, \tag{6}$$

where f_c is the value of f at the surface center S_c and f_C is the value at CV center. The cell face values of variables and their gradients are determined by assuming linear shape functions (which is, on regular grids, equivalent to linear interpolation and central difference approximation, respectively). The discretized flux approximations are implemented using the deferred correction approach, i.e. only the contribution from nearest neighbors and lower-order approximations is treated implicitly, while the correction term lags one outer iteration.
For the pressure-velocity coupling the SIMPLE method is used [2]. The time integration is performed using the implicit Euler method if the steady flow is to be solved. The implicit three time levels scheme is employed for unsteady flows requiring temporally accurate simulation. The conservation equation for space can be discretized as follows:

$$\frac{(\Delta\Omega)^{n+1} - (\Delta\Omega)^n}{\Delta t} = \sum_c (\mathbf{v}_s \cdot \mathbf{n})_c \, S_c \,, \tag{7}$$

where the superscripts n+1 and n indicate the new and old time levels, respectively. The left-hand term in Eq. (7) can be expressed through the sum of the volumes swept by the CV faces during time Δt:

$$\frac{(\Delta\Omega)^{n+1} - (\Delta\Omega)^n}{\Delta t} = \frac{\sum_c \delta\Omega_c}{\Delta t} \,. \tag{8}$$

The volume flux caused by the movement of the CV face can therefore be expressed as follows:

$$\dot{\Omega}_c = (\mathbf{v}_s \cdot \mathbf{n})_c \, S_c = \frac{\delta\Omega_c}{\Delta t} \,. \tag{9}$$

The boundary velocity of the CV does not need to be explicitly calculated; instead, the volume flux can be calculated from the known position of the CV vertices at both time levels.
The grid movement is governed by the movement of the free surface. According to the kinematic boundary condition, the non-zero mass fluxes which remain after solving the momentum and

pressure-correction equations within one outer iteration have to be compensated by displacing the free surface. The displacement volume can be expressed as:

$$\delta \dot{m}_{\mathrm{fs}} - \rho \frac{\delta \Omega'}{\Delta t} = 0 \quad \text{or} \quad \delta \Omega' = \frac{\delta \dot{m}_{\mathrm{fs}} \Delta t}{\rho} . \tag{10}$$

Many possibilities exist for linking the coordinates of grid points at the free surface to the displacement volumes $\delta\Omega'$; one is described by Lilek [3]. Here we use an iterative correction approach, in which the displacement of the cell face S_c in the direction of a unit vector $\mathbf{e}$ is approximated as:

$$h_c = \left(\frac{\delta \dot{m}_{\mathrm{fs}} \Delta t}{\rho S \mathbf{n} \cdot \mathbf{e}} \right)_c . \tag{11}$$

When the outer iterations for one time step converge, the height correction h will become negligible since $\delta\dot{m}$ goes to zero. Overturning waves can not be computed without restructuring the grid and dynamically adjusting the direction of vector $\mathbf{e}$. In the examples presented here the grid points were moved only vertically. The solution algorithm can be summarized as follows:

- Solve momentum equations using specified pressure at the current free surface;
- Solve pressure-correction equation using zero pressure-correction condition at the current free surface;
- Correct free surface shape to enforce kinematic boundary condition;
- Iterate until no further adjustment is necessary;
- Advance to the next time step.

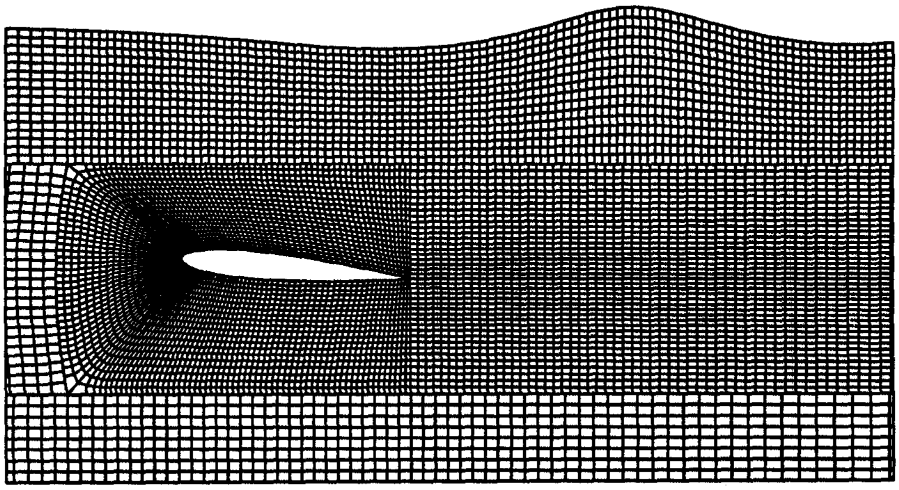

Fig. 1: Numerical grid used for the calculation of flow around hydrofoil

Application of the Method

The method has been applied to several free surface flows in which no wave breaking occurs. Here the results of prediction of flow around a NACA-0012 profile under the free surface at 5° angle of attack of and of sloshing in a large tank with and without obstacles will be presented. Figure 1 shows the non-matching block-structured grid used to calculate the flow around the hydrofoil. In Fig. 2 the free surface shape calculated on three grids is compared with experimental data of Duncan [4]. Figure 3 shows the free surface shape and the velocity field at one time instant during sloshing in a large tank with obstacles. These results and the

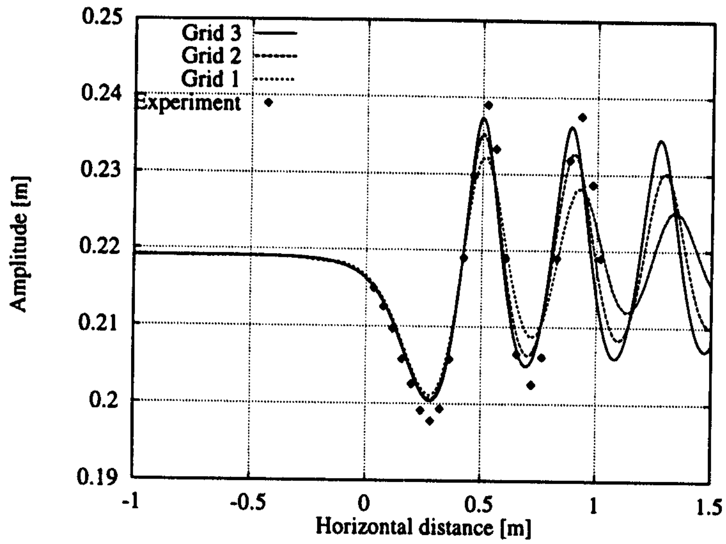

Fig. 2: Comparison of free surface shape above hydrofoil calculated on three grids with experimental data of Duncan [4]

capabilities of the solution method will be discussed in detail at the workshop. In particular the grid will be locally refined near walls and around obstacle corners to resolve boundary and shear layers more accurately, and the results will be compared with solution for inviscid flow.

References

1. Demirdžić, I., Perić, M.: Int. J. Num. Meth. Fluids, **10**, 771-790, 1990
2. Patankar, S.V., Spalding, D.B.: Int. J. Heat Mass Transfer, **15**, 1787-1806, 1972
3. Lilek, Ž.: PhD thesis, University of Hamburg, 1995
4. Duncan, J.H.: J. Fluid Mech., **126**, 507-520, 1983

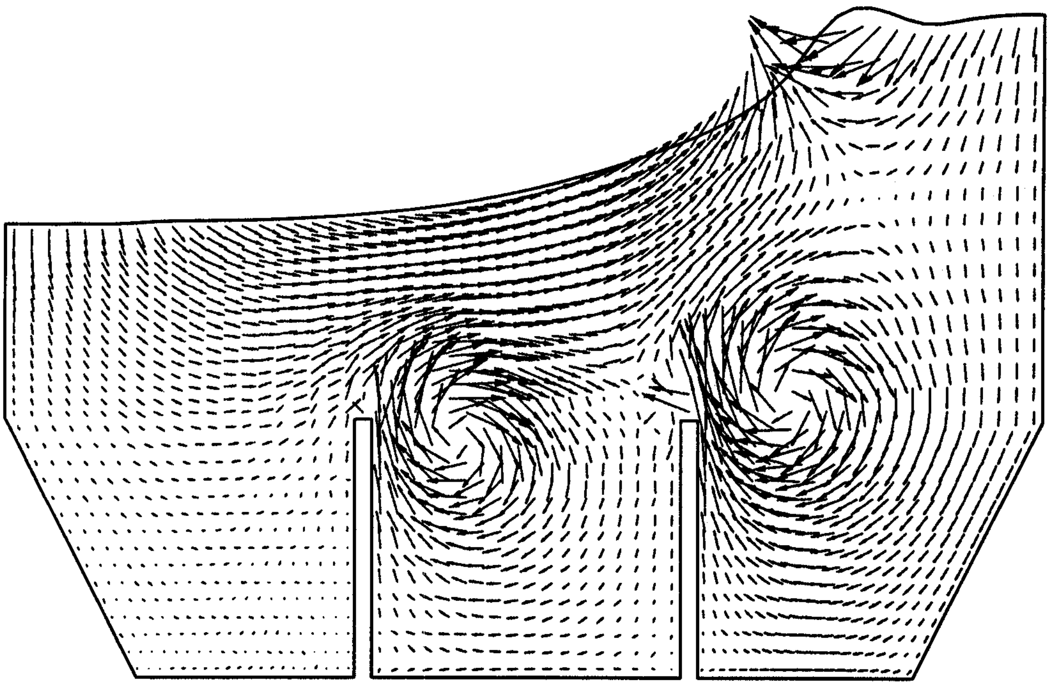

Fig. 3: Free surface shape and velocity vectors at one instant of time during sloshing in a large tank with obstacles

DISCUSSION

Delhommeau: In your treatment of a viscous fluid, did you use a refinement of the grid or a wall function?

Muzaferija et al.: The near-wall effects are modelled using wall-functions.