UNSTEADY FREE SURFACE WAVES BY DOMAIN DECOMPOSITION APPROACH

E. F. Campana and A. Iafrati
INSEAN - Italian Ship Model Basin - Rome, Italy

1 Introduction

Several numerical techniques have been recently developed to deal with the complex free surface topology induced by the wave breaking (*e.g.* Tulin & Landrini 2000, Sussman & Dommermuth 2000). However, although these approaches are strictly needed in the free surface region, their use far from the interface is expensive and neither really necessary. For instance, when studying the wave breaking induced by a submerged body an accurate description of the flow field close to the body is not needed while the computational effort can be focused about the free surface. For this reason, a domain decomposition approach is built which couple the Navier-Stokes solver, employed close to the interface, with a potential solver able to describe the flow in the body region.

In a previous work (Iafrati *et al.* 2001), an unsteady Navier-Stokes solver coupled with a Level-Set technique has been developed to study wave breaking induced by bodies moving beneath the free surface. In this work this approach is used in the free surface region, where the unsteady Navier-Stokes equations are solved by using a finite difference approach, while a Level-Set technique is used to follow the interface dynamics. Throughout the boundary of the computational domain velocity is assigned. In the body region, a boundary integral representation of the velocity potential is used with Neumann boundary conditions on the body contour and at inflow and outflow. Concerning the boundary conditions on the matching surface, two approaches are developed. In the first one, say Dirichlet type (DT), both the pressure and the velocity fields obtained from the Navier-Stokes solver are used within the unsteady Bernoulli's equation to obtain the velocity potential on the matching surface. In the second one, say Neumann type (NT), only the normal velocity obtained from the Navier-Stokes solver is used as a boundary condition on the matching surface. In both cases, the solution of the boundary integral formulation provides the velocity field on the matching to be used as boundary condition for the Navier-Stokes solver at the next time step. It is important to remark that, although here applied only in 2D problems, the proposed approaches are easily applicable also to 3D flows.

In the following a brief description of the Navier-Stokes solver and of the Level-Set technique is given while further details can be found in Iafrati *et al.* (2001) and in Iafrati *et al.* (2000). The domain decomposition approach is applied to the wave system generated by a bump on the bottom of a channel and by a submerged hydrofoil. Comparisons with the full Navier-Stokes solution and with results obtained by a fully non linear boundary element solution are presented.

2 Two fluid Navier-Stokes solver

The flow of air and water is approximated as that of a single incompressible fluid whose density and viscosity change across the interface. In an Eulerian frame of reference, local fluid properties changes with time only due to the interface motion. If surface tension and turbulence effects are neglected, the dimensionless unsteady Navier-Stokes equations in generalized coordinates are:

$$\frac{\partial U_m}{\partial \xi_m} = 0 \tag{1}$$

$$\frac{\partial}{\partial t}(J^{-1}u_i) + \frac{\partial}{\partial \xi_m}(U_m u_i) = -\frac{1}{\varrho}\frac{\partial}{\partial \xi_m}\left(J^{-1}\frac{\partial \xi_m}{\partial x_i}p\right)$$
$$-J^{-1}\frac{\delta_{i2}}{Fr^2} + \frac{1}{Re\,\varrho}\frac{\partial}{\partial \xi_m}\left(\mu G^{mn}\frac{\partial u_i}{\partial \xi_n}\right) \tag{2}$$

where u_i is the $i-$th cartesian velocity component and δ_{ij} is the Kronecker delta. The quantity

$$U_m = J^{-1}\frac{\partial \xi_m}{\partial x_j}u_j \tag{3}$$

is the volume flux normal to the ξ_m iso-surface and J^{-1} is the inverse of the Jacobian. In equation (2)

$$Fr = \frac{U_r}{\sqrt{gL_r}} \ , \ Re = (U_r L_r \varrho_w)/\mu_w$$

are the Froude and Reynolds number, respectively. U_r,L_r are reference values for velocity and length while ϱ_w, μ_w are the values of density and dynamic viscosity in water and are used as reference values. The quantity

$$G^{mn} = J^{-1}\frac{\partial \xi_m}{\partial x_j}\frac{\partial \xi_n}{\partial x_j} \tag{4}$$

is the mesh skewness tensor.

The numerical solution of the Navier-Stokes equations is achieved through a finite difference solver on a non staggered grid. Cartesian velocities and pressure are defined at the cell centers whereas volume fluxes are defined at the mid point of the cell faces and are computed by using a quadratic upwind scheme (QUICK) to interpolate cartesian velocities.

The momentum equation is integrated in time with a semi-implicit scheme: convective terms and the off-diagonal part of the diffusive ones are computed explicitly with an Adam-Bashfort scheme while a Crank-Nicolson discretization is employed for the diagonal part of the diffusive terms. A fractional step approach is used: an auxiliary velocity field is obtained by neglecting the pressure term on the right hand side of the momentum equation (*predictor step*) and in a second stage (*corrector step*) the velocity field is updated by adding a pressure correction contribution. The latter is obtained by enforcing continuity thus yielding to a Poisson equation which is solved by using a multigrid technique. When the velocity is assigned at the boundary of the computational domain, Neumann boundary conditions are obtained for the Poisson equation.

3 Free surface motion via Level-Set technique

In order to reconstruct the distribution of fluid properties in the computational domain, the actual location of the interface has to be captured. In the level-set technique fluid properties are assumed to be related to the signed normal distance from the interface $d(\boldsymbol{x}, t)$. At $t = 0$ this function is initialized assuming $d > 0$ in water, $d < 0$ in air and $d = 0$ at the interface (Sussman *et al.* 1994). The generic fluid property f is assumed to be $f(d) = f_w$ if $d > \delta$, $f(d) = f_a$ if $d < -\delta$ and

$$f(d) = (f_w + f_a)/2 + (f_w - f_a)/2\ \sin(\pi d/(2\delta))$$

otherwise. In the above expression δ is the half width of a transition region introduced to smooth the jump in the fluid properties and it is chosen so that the jump covers at least four cells. During the evolution the distance is transported by the flow, thus the equation

$$\frac{\partial d}{\partial t} + \boldsymbol{u} \cdot \nabla d = 0 \tag{5}$$

is integrated to update the distribution of the distance function. The interface being a material surface, its location is captured by reconstructing the level $d = 0$. In order to damp disturbances outgoing from the computational domain, a numerical beach model is introduced in the equation for the distance. Two beach regions are introduced close to the two boundaries of the computational domain. If $y = 0$ is the still water level, in the beach regions equation (5) takes the following form:

$$\frac{\partial d}{\partial t} = \boldsymbol{u} \cdot \nabla d - \nu(d + y) \tag{6}$$

where ν is zero at the inner limits of the beaches and grows quadratically toward the boundaries of the computational domain. Results presented below are obtained by using $\nu = 2$ at the end of the domain.

To keep constant in time the width of the transition region the distribution of the distance function is periodically reinitialized by computing, at each cell center, the minimum distance from the interface. This point is found to be very important when using the level-set technique in conjunction with the domain decomposition approach. First attempts (Iafrati *et al.* 2000) exhibited an excessive damping of the produced following wave. The reason for this excessive damping has been found to be in the need to use a time step smaller than the maximum Δt allowed by the Courant constraint ($CFL = 1$). Indeed, when integrating the motion of the interface with a time step much smaller than that allowed by the condition $CFL = 1$, a decay of the accuracy occurs (Sussman & Puckett 2000). On the basis of the above consideration, the re-initialization of the distance function is carried out with a period equal to the time step given by the respect of the condition $CFL = 1$.

4 Domain decomposition

The whole fluid domain is subdivided in two regions: a free surface region and a body region (Fig. 1). In the former the method described in the previous sections is used. In the latter, a potential flow is assumed and a boundary integral representation for the velocity potential ϕ is adopted. Two procedures, differing by the way in which the information are exchanged between the two domains, are developed. According to this, depending on the type of coupling, an overlapping (matching region) of the two domains can exist.

To illustrate the matching procedures, the wavy flow induced by a bump on the bottom of a channel is considered. The bump, whose shape is described by the formula

$$y(x) = -1 + 0.1(1 - 8x^2 + 16x^4) \;\; x \in (-0.5, 0.5),$$

suddenly starts with a constant velocity, $\boldsymbol{U}_\infty = (-1, 0)$, from the right to the left. In the results presented below it is assumed $Fr = 0.707$ and $Re = 10000$.

The wavy flow is studied in a frame of reference attached to the body (Fig. 1). The computational domain extends in the horizontal direction from $x = -14$ to $x = 14$ with the numerical beach model (6) used in $x \in (-14, -8)$ and in $x \in (8, 14)$. In the vertical direction the domain extends from the bottom at $y = -1$ up to $y = 0.4$. The extension of the matching region depends on the type of approach used to couple the two solutions.

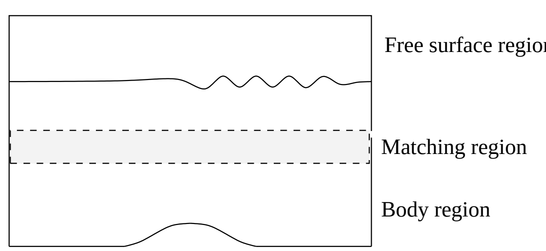

Figure 1: *Sketch of the zonal approach.*

With regard to the solution of the body region, Neumann boundary condition is applied on the body contour and at the inflow and outflow boundaries. Regardless of the type of coupling adopted, the solution of the body region provides the velocity field to be used as boundary condition on the bottom boundary of the free surface domain.

4.1 Dirichlet type

In this case the bottom boundary of the free surface region coincides with the upper boundary of the body region and panels coincide with the bottom face of the cells of the Navier-Stokes domain.

The Navier-Stokes solver provides the pressure and velocity distributions along the matching surface. Those are used within the unsteady Bernoulli's equation to update the distribution of velocity potential used as a Dirichlet boundary condition for the boundary element solution. The solution of the boundary integral problem provides the distribution of the velocity component normal to the matching surface, while the tangential component can be directly computed by the tangential derivative of the velocity potential along the matching surface.

4.2 Neumann type

For this type of coupling, an overlapping of the two domains is needed. In this case the panel distribution on the upper boundary of the body domain coincides with the bottom faces of the cell row $j = jo$ of the Navier-Stokes domain.

The solution of the Navier-Stokes equation provides the velocity distribution on the upper boundary of the body domain. This is used as a Neumann boundary condition for the boundary element solution. The latter gives back the distribution of the velocity potential all along the boundary, allowing the evaluation of the velocity field at any point inside the domain and in particular on the bottom boundary of the free surface region, where they are used as boundary condition for the Navier-Stokes solver at the next step.

5 Numerical results

To validate the domain decomposition and to compare the effectiveness of the two approaches, comparisons with the solution obtained by using the Navier-Stokes solver on the whole fluid domain (NS) are established. In applying the domain decomposition approach, the grid resolution employed in the free surface region is essentially the same of that used for the NS. Furthermore, to have a fair comparison, when using the Navier-Stokes solver in the whole fluid domain, a free-slip boundary condition is applied on the bottom of the channel.

In both the domain decomposition approaches the bottom boundary of the free surface region is located at $y = -0.35$. In the NT, the upper boundary of the body region is fifteen cells above ($jo = 15$, that is $y = -0.20$). A study of the dependence of the solution on the overlapping extension is performed.

A key issue of the unsteady domain decomposition is the control of the evolution of the boundary condition at the matching surface. To avoid that boundary conditions change too much from one step to another, the required time step is smaller than the maximum time step allowed by the stability limit $CFL = 1$. Results presented below are carried out at $\Delta t = 1/600$ while the stability limit would require, for the grid adopted, $\Delta t = 1/50$. Depencence of the solution on the time step is analysed.

In Fig. 2, the free surface profile at $t = 150$, obtained by the two different domain decomposition approaches and by the NS, are shown. In Fig. 3, the velocity components exchanged at the interface are compared with the corresponding values obtained by NS.

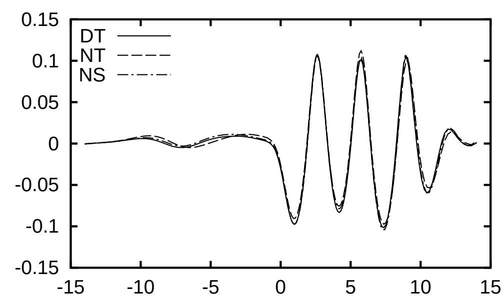

Figure 2: *Comparison between the DT and NT domain decomposition and the solution obtained by the Navier-Stokes solver throughout the fluid domain (NS).*

A very good agreement among the three solutions is observable, although the DT appears to perform slightly better.

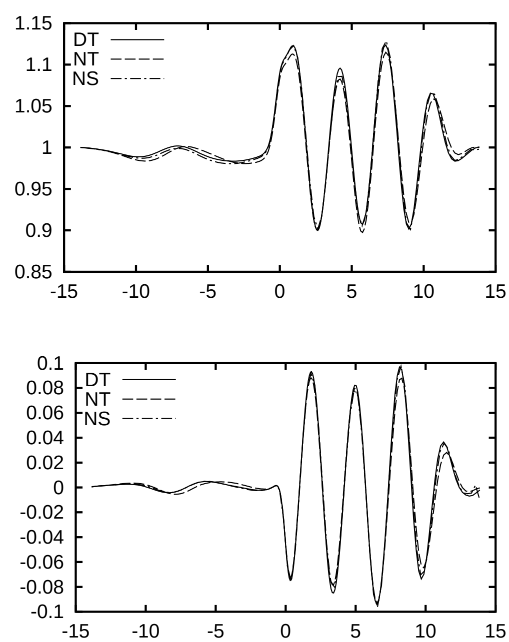

Figure 3: *Comparison of the u-velocity component* (top) *and v-velocity component* (bottom) *exchanged at the matching boundary.*

In Fig. 4, results obtained by using two different time steps are shown. The DT approach appears to be less sensitive to the time step. In Fig. 5, the effect of the width of the overlapping region in the NT approach is shown. No substantial differences occur, provided the overlapping region is larger than ten cells (for $\Delta t = 1/600$).

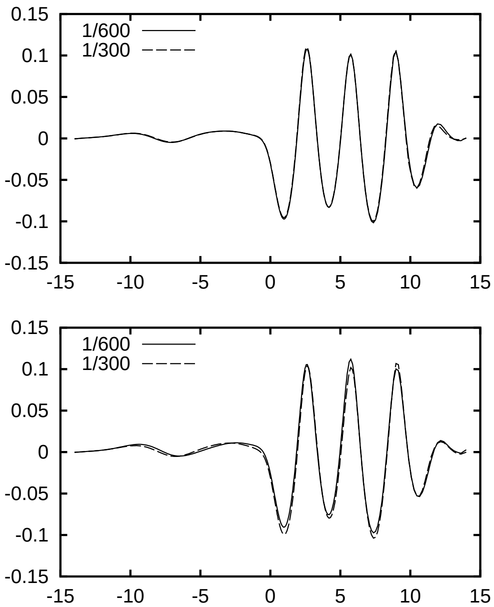

Figure 4: *Effect of the time step in the DT* (top) *and in the NT* (bottom) *approaches.*

As a case in which the advantages of using the domain decomposition are evident, the wavy flow generated by a submerged hydrofoil with an angle of attack is studied. Indeed, the use of a Navier-Stokes solver in the body region would require an important computational effort to correctly predict the flow about the hydrofoil, though one is much more interested in the free surface dynamics. This problem can be easily avoided by the domain decomposition approach described before, suitably modified by applying a Kutta condition at the trailing edge of the hydrofoil.

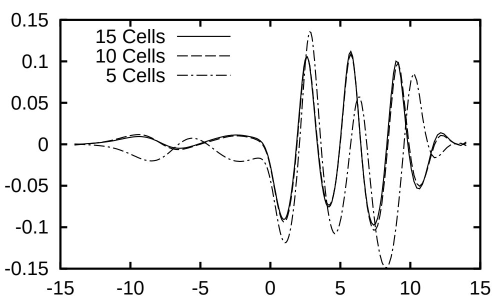

Figure 5: *Effect of the width of the overlapping region in the NT approach.*

The hydrofoil is a NACA 0012, 5 deg angle of attack and the submergence at the quarter of the chord is $y = -1.034$. The Froude number is $Fr = 0.567$ and the Reynolds number, based on the chord length, is $Re = 10000$. In Fig. 6, the wavy flow generated by the impulsive start of the hydrofoil is shown at $t = 11$ against the fully non linear boundary element solution (BEM) and the experimental data by Duncan (1983). Although numerical solutions have not yet reached a steady state, a good agreement is displayed by the domain decomposition approach.

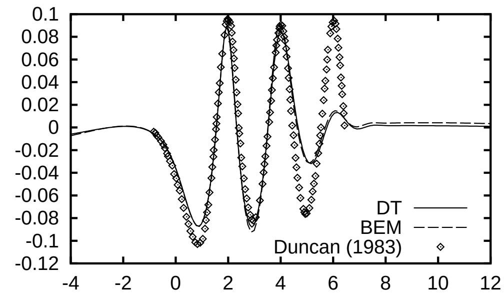

Figure 6: *Flow about a submerged hydrofoil: comparison with the fully non linear boundary element solution and the experimental results by Duncan (1983).*

Acknowledgments

This work was financially supported by the *Ministero Trasporti e Navigazione* in the frame of the INSEAN research plan 2000-02 and by the *Office of Naval Research*, under grant *N*.000140010344, through Dr. Pat Purtell. Authors also wish to thank Dr. D. Dommermuth for the useful discussions they had during his visit at INSEAN.

References

DUNCAN, J.: *The breaking and non - breaking wave resistance of a two-dimensional hydrofoil*, J. Fluid Mechanics, **126**, 507-520 (1983).

IAFRATI, A., DI MASCIO, A., CAMPANA, E.F.: *A level-set technique applied to unsteady free surface flows*, Int. Journal for Numerical Methods in Fluids, in press (2001).

IAFRATI, A., OLIVIERI, A., PISTANI, F., CAMPANA, E.F.: *Numerical and experimental study of wave breaking generated by a submerged hydrofoil*, Proc. 23rd ONR Symposium on Naval Hydrodynamics (2000).

SUSSMAN, M., SMEREKA, P., OSHER, S.J.: *A level set approach for computing solutions to incompressible two-phase flow*, Journal of Computational Physics, **114**, 146-159 (1994).

SUSSMAN, M., DOMMERMUTH, D.: *The numerical simulation of ship waves using cartesian grid methods*, Proc. 23rd ONR Symposium on Naval Hydrodynamics (2000).

SUSSMAN, M., PUCKETT, G.: *A coupled level-set and volume-of-fluid method for computing 3D and axisymmetric incompressible two-phase flows*, Journal of Computational Physics, **162**, 301-337 (2000).

TULIN, M., LANDRINI, M.: *Breaking waves in the ocean and around ships*, Proc. 23rd ONR Symposium on Naval Hydrodynamics (2000).