A Numerical Method for Unsteady Wave Flows Around Submerged Obstacles

by C. de Jouëtte

Principia R.D, Le Berlioz, 06561 Valbonne cedex, France.

Summary

We consider the problem of the unsteady incompressible flow around a flat plate in a finite depth with free surface. The calculated flow is induced by the diffraction of an incoming Airy wave on the plate. The model is valid for three-dimensional flows, but only the 2D flows case is discussed here. The model takes into account the upstream reflected waves, the downstream transmitted waves and the vortex generation induced at the body boundary. The unsteady Euler or thin layer Navier-Stokes equations are solved using a fully implicit time scheme. At each physical time step, the solution of the differential equations system is obtained using a pseudo-unsteady approach. This iterative method permits to find the converged solution at each time step. The free surface is treated by means of a moving boundary fitted mesh and we apply the classical non-linear dynamic and kinematic conditions on this surface.

I. Problem formulation in physical plane

The fluid flow is computed around the flat plate in a domain bounded by the free surface SL, the bottom F and two upstream and downstream vertical boundaries AB,CD.

I.1 System of equations

The unsteady Navier-Stokes equations for incompressible fluids can be written as follows:

$$(1) \qquad div(\overrightarrow{U}) = 0 \quad , \frac{\partial \rho \overrightarrow{U}}{\partial t} + div(\rho \overrightarrow{U} \otimes \overrightarrow{U} + p\overline{I} - \overline{\tau}) = \rho \overrightarrow{g}$$

where ρ is the (constant) fluid density, $\overrightarrow{U}$ the velocity, p the pressure, $\overline{I}$ the unity tensor, $\overrightarrow{g}$ the gravitational acceleration, $\overline{\tau}$ the viscous stress tensor.

I.2 Boundary conditions

I.2.1 Plate and bottom

In the inviscid-fluid case, a zero normal velocity condition, $\overrightarrow{U}.\overrightarrow{n} = 0$, is applied on solid surfaces. In the case of a viscous flow the classical no-slip condition $\overrightarrow{U} = \overrightarrow{0}$ is assumed.

I.2.2 Free surface

a) Kinematic condition:

Let $F(x, y, t) = 0$ be the free surface equation in 2D flow. The slip condition on the free surface can be written:

$$(2) \qquad \frac{dF(x,y,t)}{dt} = \left(\frac{\partial}{\partial t} + \overrightarrow{U}.\overrightarrow{grad}\right) F(x,y,t) = 0 \qquad \text{on } F = 0$$

b) Dynamic condition :

The viscous stress tensor is neglected at the free surface, so that the pressure is equal to the constant atmospheric pressure p_a:

(3) $$p = p_a \qquad \text{on } F = 0$$

I.2.3 Upstream and downstream boundaries

The boundary conditions to be applied on the vertical upstream and downstream boundaries must express the property that the boundaries do not reflect outgoing waves. We use an Orlanski type open boundary condition [1].
Generally speaking, this condition applied to a variable ϕ assumes that, in the close neighbourhood of the bounbary, ϕ is governed by a transport equation of the form:

(4) $$\frac{\partial \phi}{\partial t} + C_\phi \vec{\nu}.\overrightarrow{grad}\phi = 0$$

where $\vec{\nu}$ is the outward unit normal, and $C_\phi > 0$. The transport velocity C_ϕ is unknown and is determined by applying (4) at the inner mesh point closest to the boundary where ϕ is known. Then (4) is applied at the boundary point to determine ϕ at this point.
On the upstream boundary AB, we impose the condition that the flow is the superposition of an Airy incoming wave and a reflected wave. The Orlanski condition is applied only to the reflected wave.
On the downstream boundary CD, the Orlanski condition applies to the complete flow.

II. Problem formulation in transformed plane

II.1 Coordinate transformation

An adaptative mesh is used in order to obtain a precise treatment of the free surface position. The grid is constructed so that the free surface, at each instant, is a mesh surface. The purpose of the coordinate transformation is to keep a fixed coordinate system (ξ, η) in the computational domain for each physical time step. Let us consider the general transformation of coordinates from the physical plane (cartesian coordinates x, y) to the computational domain (coordinates ξ, η):

(5) $$\xi = G_1(x, y, t) \qquad , \eta = G_2(x, y, t) \qquad , t' = t$$

or its inverse, from the computational domain to the physical domain:

(6) $$x = g_1(\xi, \eta, t') \qquad y = g_2(\xi, \eta, t') \qquad t = t'$$

Assuming that the free surface is the coordinate surface $\eta = \eta_L = constant$, the free surface equation can be then written as:

(7) $$y = g_2(\xi, \eta_L, t') = H_1(\xi, t')$$

The transformation imposes the condition that the bottom $(y = 0)$ is the surface $(\eta = 0)$, and that the breakwater boundary is a part of the surface $\eta = \eta_P = constant$.

II.2 Transformation of the equations

The equations are expressed in the curvilinear coordinates (ξ, η) defined by eqs. (6) or (7). The time derivatives will be denoted by $\frac{\partial}{\partial t}$ if they are calculated at fixed (x, y) , and by $\frac{\partial}{\partial t'}$ if they are calculated at fixed (ξ, η). If one uses the viscous thin-layer assumption [3], the equations (1) can be written with these coordinates in the conservative law form as:

$$\begin{aligned} &\frac{\partial}{\partial \xi}\left(\frac{\tilde{u}}{J}\right) + \frac{\partial}{\partial \eta}\left(\frac{\tilde{v}}{J}\right) = 0 \\ &\frac{\partial Q}{\partial t'} + \frac{\partial \hat{F}}{\partial \xi} + \frac{\partial \hat{G}}{\partial \eta} = \frac{\partial \hat{G}_v}{\partial \eta} + \frac{1}{J} S \end{aligned} \tag{8}$$

where J is the jacobian of the transformation.

$$J = \frac{\partial(\xi, \eta)}{\partial(x, y)} \tag{9}$$

and:

$$\begin{aligned} Q &= J^{-1}\begin{bmatrix} \rho u \\ \rho v \end{bmatrix} \\ \hat{F} &= J^{-1}\begin{bmatrix} \rho u(\xi_t + \tilde{u}) + \xi_x p \\ \rho v(\xi_t + \tilde{u}) + \xi_y p \end{bmatrix} \\ \hat{G} &= J^{-1}\begin{bmatrix} \rho u(\eta_t + \tilde{v}) + \eta_x p \\ \rho v(\eta_t + \tilde{v}) + \eta_y p \end{bmatrix} \\ S &= \begin{bmatrix} 0 \\ -\rho g \end{bmatrix} \end{aligned} \tag{10}$$

and u, v are the cartesian velocity components.
The classical viscous thin-layer assumption is employed by retaining only the viscous diffusion term in the η-direction, which is given by:

$$\hat{G}_v = \frac{\mu_{eff}}{J}({\eta_x}^2 + {\eta_y}^2)\begin{bmatrix} u_\eta \\ v_\eta \end{bmatrix} \tag{11}$$

μ_{eff} is the global viscosity, which may include the eddy-viscosity.

The contravariant velocity components used in the equations are defined as:

$$\begin{aligned} \tilde{u} &= \xi_x u + \xi_y v \\ \tilde{v} &= \eta_x u + \eta_y v \end{aligned} \tag{12}$$

II.3 Free surface condition

The free surface condition (2) can be written, using the equation (7) of this surface:

$$\frac{d}{dt}\left(y - g_2(\xi, \eta_L, t')\right) = 0, \qquad \eta = \eta_L \tag{13}$$

hence:

$$v - \frac{\partial g_2}{\partial t'} - \frac{d\xi}{dt}\frac{\partial g_2}{\partial \xi} = 0, \qquad \eta = \eta_L \tag{14}$$

where:

(15) $$\frac{d\xi}{dt} = \xi_t + u\xi_x + v\xi_y$$

The relation (14) provides the value of $\frac{\partial g_2}{\partial t'}$ on the free surface, ξ_t being obtained from the definition itself of the coordinate transformation. In fact, for our problem, it is not necessary that the coordinates lines ($\xi = constant$) vary with the time, and we can take $\xi_t \equiv 0$.
If one knows $\frac{\partial g_2}{\partial t'}$ for $\eta = \eta_L$, the coordinate transformation allows one to calculate $\frac{\partial g_2}{\partial t'}$ at any point of the plane (ξ, η) (then at any mesh nodes), and then to determine how the coordinate transformation varies with the free surface movement during the time.

III. Time discretization and pseudo-compressibility method

III.1 Discretization of the flow equations

Equations (8) are discretized at time $t^{n+1} = (n+1)\Delta t$ by means of a second order fully implicit time scheme:

(16) $$\begin{aligned} &\frac{\partial}{\partial \xi}\left(\frac{\tilde{u}}{J}\right)^{n+1} + \frac{\partial}{\partial \eta}\left(\frac{\tilde{v}}{J}\right)^{n+1} = 0 \\ &\frac{3Q^{n+1} - 4Q^n + Q^{n-1}}{2\Delta t} + \tilde{L}(Q^{n+1}, p^{n+1}) = \left(\frac{1}{J}\right)^{n+1} S \end{aligned}$$

where:

(17) $$\tilde{L}(Q^{n+1}, p^{n+1}) = \left(\frac{\partial \hat{F}}{\partial \xi}\right)^{n+1} + \left(\frac{\partial \hat{G}}{\partial \eta}\right)^{n+1} - \left(\frac{\partial \hat{G}_v}{\partial \eta}\right)^{n+1}$$

The method of pseudo-compressibility for solving the steady incompressible flows discussed in [4,5] can be extended in order to solve the equations (16) of primitive variables Q^{n+1} and p^{n+1} by taking into account the pseudo-unsteady system (18) presented below. To simplify the formulation the superscript (n+1) on the unknown variables at the time (n+1) is suppressed in the following development. The system is solved by making iterations on the pseudo-time τ until convergence is obtained.

(18) $$\begin{aligned} &\frac{\partial}{\partial \tau}\left(\frac{1}{J}\tilde{\rho}\right) + \frac{\partial}{\partial \xi}\left(\frac{\tilde{u}}{J}\right) + \frac{\partial}{\partial \eta}\left(\frac{\tilde{v}}{J}\right) = 0 \\ &\frac{\partial \tilde{Q}}{\partial \tau} + \frac{3Q - 4Q^n + Q^{n-1}}{2\Delta t} + \left(\frac{\partial \hat{F}}{\partial \xi}\right) + \left(\frac{\partial \hat{G}}{\partial \eta}\right) - \left(\frac{\partial \hat{G}_v}{\partial \eta}\right) = \left(\frac{1}{J}\right) S \end{aligned}$$

where:

(19) $$\tilde{Q} = J^{-1}\begin{bmatrix} \tilde{\rho}u \\ \tilde{\rho}v \end{bmatrix}$$

Here $\tilde{\rho}$ is a new variable called pseudo-density, and the pressure p is given as an explicit function of $\tilde{\rho}$:

(20) $$p = G(\tilde{\rho})$$

In the inviscid flow case, the system of equation (18) must satisfy the hyperbolicity condition, which leads to the conditions [5]:

(21) $$\tilde{\rho} > 0 \quad \text{and } G'(\tilde{\rho}) > 0$$

We take $p = C_0 Ln(\tilde{\rho}) + C_1$, where C_0 and C_1 are constant.

To solve system (18) in τ, we use the numerical method developed previously by Jameson et al. [2] and Vatsa [3] ; this is a finite-volume method based on a space-centered scheme, second and fourth order artificial viscosity terms, Runge-Kutta time stepping,and implicit residual smoothing. A local time step is used.

III.2 Boundary treatment by the compatibility relations

The boundary treatment at the free surface and at the upstream and downstream boundaries is based on the use of the compatibility relations (CR) deduced from the system (18) which is hyperbolic with respect to τ in the inviscid case. In this approach, one applies the CR associated with positive or null eigenvalues (for the outward normal direction to the boundary) and the missing CR (associated with negative eigenvalues) are replaced by the boundary conditions [6].

IV. Code development

A code has been developed for 2D and 3D flows based on the method described above. Unsteady flows without free surface have been computed and we are in the process of implementing the free surface boundary condition. Two-dimensional test cases without free surface will be presented at the workshop.

References

[1] Orlanski, Journal of Computational Physics, vol. 21, 1976 pp 251-269.

[2] A.Jameson, W.Schmidt, and E.Turkel "Numerical Solutions of the Euler Equations by Finite Volume Methods Using Runge Kutta Time Stepping Schemes", AIAA Paper 81-1259 (1981)

[3] V.N.Vatsa "Accurate Numerical Solutions for Transonic Viscous Flow over Finite Wings" , J.Aircraft 24, vol. 24, pp 377-385 (1987)

[4] H.Viviand, "Systèmes Pseudo-instationnaires pour le calcul d'écoulements stationnaires de fluides Parfaits", Publication ONERA 1983-4 (1983).

[5] C.de Jouëtte, H.Viviand, S.Wornom, J.M.Legouez, "Methods for Incompressibe Flow Calculation", 4th International Symposium on Computational Fluid Dynamics, University of California at Davis, sept. 9-12,1991

[6] L.Cambier, W.Ghazzi, J.P.Veuillot, H.Viviand, "Approach for the Computation of Viscous Transonic Flows by Unsteady Type Methods", in Computational Methods in Viscous Flows, Vol. 3, W.G.Habashi (Ed.) Pineridge Press, 1984

DISCUSSION

YUE: Do you account for the normal and tangential gradients of the (viscous) stress tensor in your free-surface dynamic boundary conditions?
DE JOUETTE: No, we neglect the viscous stress tensor in the free-surface dynamic condition.

RAVEN: Artificial compressibility methods have originally been proposed for steady problems. To apply it to unsteady flows, you use an iteration loop for pseudo-time, for each time step.

- How many iterations are needed for this inner iteration loop?
- Do you think the greater robustness of compressible codes is worth the large amount of extra CPU time spent, compared to a truly incompressible approach?

DE JOUETTE:
1 - The iterations number depends on the level of convergence you want to obtain. For example, in order to have log(residual) ≤ -3, (with residual =

$$\sqrt{\left(\frac{\partial\tilde{\rho}}{\partial t}\right)^2+\left(\frac{\partial\tilde{\rho}u}{\partial t}\right)^2+\left(\frac{\partial\tilde{\rho}v}{\partial t}\right)^2}\ ),$$

you need around 20-25 iterations.
2 - On one hand, the first tests we made with unsteady flows (without moving mesh) give us some CPU time which can be compared to other incompressible approach. And on the other hand, we want to introduce an implicit scheme in pseudo-time and to implement a multigrid method to impose convergence.

YEUNG: We have developed some rather elaborate algorithms that can handle steep waves for finite-difference method. This has been applied to inviscid-flow and viscous flow problem with a body near or in a free surface. They are described in the following works:
a) Yeung & Vaidhyanathan, *Int. J. of Numer. Methods in Fluids,* vol. 14, 1992;
b) Yeung & Ananthakrishnan, *J. of Engrg. Math.,* vol. 26, 1992.
You may find them helpful.

GRUE: By applying your method it is expected that acoustic streaming appears at the bodies. For 2-D bodies in free surface flows a constant circulation around the bodies then occurs. This circulation greatly affects the wave forces, as for example observed experimentally by John Chaplin (1984), *J. Fluid Mech.* It will be interesting if your method can reproduce the experimental observations. To my knowledge, this has not yet been done.