1 Introduction

The flow field around a ship is extremely complex, even for the simplest case of motion through calm water with constant forward speed. In particular, many vortical structures are originated by the ship motion. Some of them are directly related to ship breaking waves (Landrini et al. 2001). In other cases, vorticity is created at the hull boundary and shed along and downstream the ship.

In this paper, we report our ongoing investigations aimed to gain fundamental understanding of the fluid dynamic processes connected with the motion of a ship. In particular, we consider a two-dimensional prototype problem consisting in a vertical flat plate, piercing the air-water interface, and moving forward with known velocity. This rather simple problem is meant to be roughly representative of the fluid phenomena occurring around the bow of a blunt ship and near a transom stern.

![Image]

Fig. 1: Sketch of the considered problem and nomenclature adopted.

The problem is studied numerically by a Navier-Stokes solver with a Level-Set technique to capture the air-water interface. Details of the method are presented and numerical results are discussed. Preliminary data from a companion experimental investigation, still in its infancy, are presented.

2 Numerical modeling

Background fluid-flow solver  We assume that the evolution of a two-fluid system is governed by the Navier-Stokes equations

\[
\nabla \cdot \mathbf{u} = 0
\]

\[
\frac{D\rho}{Dt} = -\nabla p + 2\nabla \cdot \mathbf{u}D + 2\sigma \kappa \delta_{i} n + \rho g
\]

where the density \( \rho \) and the dynamic viscosity \( \mu \) vary sharply across the interface. The term \( 2\sigma \kappa \delta_{i} n \) is the capillary force, with \( \sigma \) the surface tension, \( n \) the normal to the interface, \( \kappa \) half the interface curvature and the Kronecker delta \( \delta_{i} \) is equal to unity on the interface and zero elsewhere. Finally,

\[
(D)_{ij} = D_{ij} = \frac{1}{2} \left( \frac{\partial u_{i}}{\partial x_{j}} + \frac{\partial u_{j}}{\partial x_{i}} \right)
\]

is the rate of strain tensor.

A second-order approximation in time of Eq. (1) can be written as:

\[
\frac{\rho \mathbf{u}^{n+1/2} - \rho \mathbf{u}^{n-1/2}}{\Delta t} = \frac{\mathbf{F}(\mathbf{u})^{n+1/2} - \mathbf{F}(\mathbf{u})^{n-1/2}}{\Delta t} + \nabla \cdot \mathbf{j} + \frac{\rho \mathbf{u}^{n+1/2} - \rho \mathbf{u}^{n-1/2}}{\Delta t} + \nabla \cdot \mathbf{j} + \nu \nabla^{2} \mathbf{u}^{n+1/2}
\]

and is solved through a predictor-corrector scheme. To simplify the following description, we name with \( F(\mathbf{u}) \) the terms that will be approximated through the Taylor expansion in the two steps, that is

\[
F(\mathbf{u}) = -(\mathbf{u} \cdot \nabla) \mathbf{u} + \frac{2 \nabla \cdot \mathbf{u} D}{\rho} + \frac{2 \sigma \kappa \delta_{i} n}{\rho} + g
\]

In the predictor step, the density at the time \( n + 1/2 \) is substituted with that at \( n - 1/2 \), and the term \( F(\mathbf{u})^{n+1/2} \) is obtained through a Taylor expansion form the previous time steps. The term containing the pressure gradient is written as:

\[
\frac{\nabla p^{n+1/2}}{\rho^{n+1/2}} = \frac{\nabla p^{n-1/2}}{\rho^{n-1/2}} + \frac{\nabla p^{n-1/2}}{\rho^{n-1/2}}
\]

and the two-steps procedure to obtain the new velocity becomes:

\[
\mathbf{u}^{n+1} = \mathbf{u}^{n} + \Delta t \left\{ \frac{\mathbf{F}(\mathbf{u})^{n+1/2} - \nabla \mathbf{p}^{n+1/2}}{\rho^{n+1/2}} \right\}
\]

In case of incompressible fluids, the previous equation becomes

\[
\nabla \cdot \mathbf{u}^{n+1} = \nabla \cdot \left( \frac{\rho^{n+1/2} \mathbf{u}^{n+1/2} + \rho^{n+1/2} \mathbf{u}^{n+1/2}}{\rho^{n+1/2} + \rho^{n+1/2}} \right)
\]

and it gives a first guess for \( \rho_{k+1}^{n+1/2}, \rho_{k+1}^{n+1/2}, \) and \( \rho_{k}^{n+1/2} \). In the iterative corrector procedure the term \( F(\mathbf{u})^{n+1/2} \) is obtained through a centered Taylor expansion, and the pressure gradient is written as

\[
\frac{\nabla p_{k}^{n+1/2}}{\rho_{k}^{n+1/2}} = \frac{\nabla p_{k}^{n+1/2}}{\rho_{k-1}^{n+1/2}} + \frac{\nabla p_{k}^{n+1/2}}{\rho_{k-1}^{n+1/2}}
\]

with \( k = 1, 2, \ldots \) the iterative step. For the predictor part

\[
\mathbf{u}^{n+1} = \mathbf{u}^{n} + \Delta t \left\{ \frac{\mathbf{F}(\mathbf{u})^{n+1/2} - \nabla \mathbf{p}_{k-1}^{n+1/2}}{\rho_{k-1}^{n+1/2}} \right\}
\]

and using the divergence-free condition

\[
\nabla \cdot \mathbf{u}^{n+1} = \nabla \cdot \left( \frac{\rho_{k-1}^{n+1/2}}{\rho_{k-1}^{n+1/2}} \right)
\]

From which we can work out pressure, velocity, density and viscosity at the new iterative step until the convergence is satisfied.

For the spatial discretization, we have used staggered grid and \( x \)- and \( y \)-derivatives have been calculated using a second-order approximation and ENO schemes.
Interface capturing The interface between the fluids is traced using \( \phi \) the level-set function \( \phi \). A narrow band around the interface is characterized by its signed distance from the air-liquid interface. It is used as well to smooth the discontinuity of density and viscosity at the free surface. In particular the density is written as \( \rho = f(\phi) \). So the continuity equation becomes:

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u} + \rho \mathbf{V}_F \phi) = 0
\]

that is the transport equation for the distance function.

As the velocity field \( \mathbf{u} \) is not compatible with the displacement of a proper distance function a reinitialization of \( \phi \) is necessary after a given number of time steps. The reinitialization, as introduced by Osher and Sethian [2], uses the equation

\[
\frac{\partial \phi}{\partial t} + \nabla \cdot (\phi \nabla \phi) = 0
\]

where \( \phi \) evolve in the pseudo time \( \tau \) until stationary conditions are obtained. The standard solving procedure is based on an ENO scheme for the calculation of the spatial derivatives. In such schemes, an error is introduced at the interface and [3] proposed to solve Eq. (3) by

\[
\phi_{i,j}^{n+1} = \phi_{i,j}^n - \Delta \tau \left( \frac{\Delta \phi_{i,j}^n}{\Delta x} \right) - D_{i,j}
\]

where

\[
D_{i,j} = \frac{2\phi_{i-1,j}^n + \phi_{i,j}^n}{\Delta x^2} + \frac{\Delta \phi_{i,j}^n}{\Delta y^2}
\]

Though this procedure reduces the rounding error caused by the level-set reinitialization, some smoothing is still visible and further improvement is gained by using, for the interface cells,

\[
D_{i,j}^{n+1} = \frac{\phi_{i-1,j}^n + \phi_{i,j}^n}{\Delta x^2} + \frac{\Delta \phi_{i,j}^n}{\Delta y^2}
\]

This formulation does not smooth the oscillations at the interface, hence a weighted combination of the two methods has been introduced at the interface:

\[
\phi_{i,j}^{n+1} = \phi_{i,j}^n - \Delta \tau \left( \frac{\Delta \phi_{i,j}^n}{\Delta x} \right) - (a_1 D_{i,j} + a_m D_{i,j}^{n+1})
\]

and, from our experience, \( a_1 = 0.8 \) and \( a_m = 0.2 \) results the best choice.

3 Discussion

The considered problem is characterized by the following non-dimensional groups

\[
Fr = \frac{U_M}{\sqrt{gh}}, \quad Re = \frac{\rho_v U_M b h}{\mu_w}, \quad We = \frac{h \rho_v U_M^3}{\sigma}
\]

where \( h \) and \( U_M \) are the initial submergence and the maximum velocity of the plate, respectively. The subscript \( w \) indicates the water properties. In the following, lengths are made non-dimentional by \( h \), and the force values are divided by \( \rho gh \).

In the numerical simulation, the plate velocity is described by a Heaviside function \( U(t) = U_M H(t) \). Figures 2-3 show the Navier-Stokes (laminar) simulation for \( Fr=0.8 \) and \( Re \approx 0(10^8) \). Surface-tension effects have been modeled (\( \text{Wenzel} \)). At the beginning, the fluid runs up along the forward side of the plate (moving from left to right) and is sucked down on the lee side. At the same time, the starting vortex appears and progressively grows as time passes, until it is shed downstream the plate. After that, a relatively small amount of vorticity is injected in a rather thin shear layer.

On the forward side, the water reaches the maximum run up and then slightly falls down, accompanied by the formation of a plateau of water whose extension in the forward direction increases as time passes. The water front steepens gradually and a forward-plunging jet appears. As the breaking process develops, the water level on the upstream side of the plate decreases and, later on, increases again, maybe in a cyclical fashion, though the length of simulation is too short for a definitive judgment. The breaking process is characterized by the formation of a cavity entrapping air and the splashing up of the water after the impact of the plunging jet. As observed in experiments (Bowmar 1989) and numerical simulations (Tatin and Landrati 2000), the splash up may evolve into another breaking cycle with the formation of a second forward-plunging jet and a (possibly weaker) second splash-up structure according to the strength of the initial breaking front. So far, we have not investigated this process on a long time scale and the simulation is stopped before the first splash up completely collapses down, under the action of the gravity. In any event, this breaking process is an important source of vorticity, besides the vorticity created at the body boundary, though in the former case the phenomenon is intrinsically inviscid and related to the change of topology of the fluid domain.

On the lee side, the starting vortex sumps down the interface which evolves into a backward breaker. Also in this case, a cavity entrapping air is formed and a residual clockwise circulatory region is observed. Closer views of the rotational regions created upstream and downstream through the breaking process are given in figure 4.

Figure 5 shows the pressure force acting on the plate for \( Fr = 0.8 \) and the separate contributions coming from right and left sides. The force is positive in the positive \( x \)-direction. At the (impulsive) start, a high overpressure on the left side and a deep depression on the right one (pressure impulse) are observed. This phenomenon is soon to decrease, leaving a positive force on the left side and a negative force on the right, the latter mainly due to a depressional area around the lower tip of the plate. This depression becomes the core of the nascent vortex, and its separation is accompanied by the first peak of the right force. After the starting vortex is detached, the pressure decreases again around the lower tip and the force acting on that side is dominated by gravitational effects. On the left side, after the sudden start, the continuous piling up of water results in the peak load synchronized with the maximum run up. As soon as the water front propagates away from the solid boundary, the force decreases and does not show any signature of the interface breaking observed immediately after. In the final part of the simulation, the small-amplitude oscillations of the left force are due to the oscillation of the water level on the forward side of the plate.

For smaller Froude number a similar evolution is observed, and it is not reported for the limited space available. For \( Fr=1.2 \), figure 6, a more vigorous vortex shedding is found without any breaking events of the interface downstream the plate. Interestingly, for this speed the interface moves down and reaches the edge of the plate, resembling the case of a dry transom stern in a ship. In general, for all the considered cases, on the upstream side the interface evolves into a forward-plunging jet.
with strength increasing with the Froude number.

Similar numerical computations has been performed in [5] for the inviscid free-surface problem, with a vortex sheet emanating from the sharp immersed edge of the plate by using a suitable Kutta condition. They have evidenced the presence of three regimes as for the interaction between the free surface and the vortex sheet, namely a subcritical regime, $Fr < 0.7$, where no significant interactions between a single branched spiral vortex and the interface occurs before the breaking event; a transcritical regime, $0.7 < Fr < 1.0$, where the free surface stretches the vortex sheet, causing its roll up, even though the interaction is limited; a supercritical regime, $Fr > 1.0$, where the vorticity affects significantly the free-surface motion. For the pre-breaking regime, our results are consistent with these find-
Fig. 4: Velocity fields near the cavity entrapping air originated by the folding and breaking of the interface for Fr=0.8. Top: forward cavity. Bottom: backward cavity.

Fig. 5: Pressure loads acting on the left (top) and the right (center) sides of the plate for Fr=0.8. The resulting force is plotted in the bottom figure.

Fig. 6: Interface and vorticity field in water for Fr=1.2. Non-dimensional time increases from top to bottom (τ = 0.6, 1.2, 1.8, 2.4) induced by the breaking.

Fig. 7: Interface and vorticity field in water for Fr=0.6 and τ = 3.

nevertheless, our numerical scheme still presents some problem related to the numerical diffusion of the interface across the surface cells, which presently limits our analysis capability. The most important feature of the surface capturing schemes, above all of the one adopting the level set, is the possibility to leave the mesh unaltered throughout the calculation as the interface is followed using an analytical function φ directly linked to density and viscosity. As the interface is smeared over at least two cells, there is a diffusion error in the solution of the Poisson equation which has to be added to the numerical diffusion error in the calculation of the convection term and to the error in the mass conservation.
Likely, the effect of these errors is a wrong approximation of the mass transport in the area where high vorticity is generated. For example, on the upper part of the plunging jets there is a vortex whose wrong numerical handling create sometimes a nose-up effect. However, it can be reduced using an appropriate advection scheme as the superbee scheme. Also, we note that the size of the submerged air bubbles decreases in time because of the present limits of the method. Therefore, further improvements are needed to handle accurately on a longer time scale the post-breaking evolution.

Figure 8 shows the forces acting on the right side of the plate for different velocities. As the maximum velocity increases, the depressional area downstream of the plate becomes deeper, causing a shift of the forces to higher values. At higher Froude number, we observe an oscillation in the force diagram because of the vortex shedding. This is apparent for \( Fr = 1.2 \). Interestingly, for this case, the force on the right side vanishes because of the reaching of dry conditions. Second vortex, later no more effects can be noticed on that side of the plate.

4 First experimental results

We briefly report our preliminary results from experiments performed in a flume 420 mm wide, 18 m long and a still water depth of 700 mm. An aluminum vertical plate is towed along the flume, and the initial draft is 47 mm, see figure 9. Displacement and velocity are measured by an optical encoder and used to drive the numerical simulations. Comparison of numerical and experimental results for \( Re \approx 30000 \), \( We \approx 10000 \) and \( Fr \approx 1 \) are ongoing. Standard water density, viscosity and surface tension have been used, though fluorescent material has been added to the water, and temperature and pressure conditions have not been monitored. Figure 11 shows the experimental and numerical evolutions for the corresponding instants of time. The agreement is qualitatively rather good, and also the vortical structures luckily evidenced by some ventilation effects are visible. Actually, some disturbances on the free surface downstream the plate (now moving from left to right) are detectable, partly due to some leakage from the sides of the plate and the walls of the flume. The plunging of the upstream jet is delayed in case of the numerical simulation because of the problems previously mentioned. Also, some influence of surface tension can also be seen: the jet observed in the experiments is more rounded than the numerical one.

We have just modeled the surface tension and figure 10 reports a very preliminary, yet encouraging, result.

Acknowledgements: INSEAN research activity is supported by the Italian Ministero delle Infrastrutture e Trasporti through INSEAN Research Program 2000-02.

References

Fig. 11: Flow generated by the motion of a vertical flat plate. Left: experimental visualization. Right: numerical computations, interface location and vorticity contours. (t = 0.24 s, 0.27 s, 0.38 s, 0.43 s)