A Fully-Coupled VOF Method For Free Surface Simulation

Yann Andrillon, Bertrand Alessandrini
Laboratoire de Mécanique des Fluides (CNRS UMR6598)
Ecole Centrale de Nantes - FRANCE

ABSTRACT

We present in this paper a fully coupled method for free surface flows computations using Navier-Stokes equations and Volume Of Fluid interface capturing method. This approach allow to study very complex hydrodynamics phenomena like breaking waves, impact, jet unable to simulate by interface tracking algorithm. Fully coupled approach allow to obtain high level of accuracy specially for unsteady problems. Numerical results on wave propagation and flows in tank are presented and successfully compared with experiments.

KEY WORDS: VOF Method, Fully-Coupled Algorithm, Navier Stokes Equations, Viscous Flow, Free Surface Flow, Wave Breaking.

1.INTRODUCTION

The behavior of complex moving fluid interfaces is important in many hydrodynamics applications. We present in this paper a new numerical approach for free surface flow computation using Volume of Fluid capturing method [6, 13, 12, 15] associated with Fully Coupled formulation to solve Navier-Stokes equations. Fully Coupled method [5, 1] makes it possible to obtain a high degree of accuracy on unsteady calculations whereas the use of a Volume Of Fluid technique for free surface effects is very interesting for the flexibility and the versatility of the global algorithm. In the following a brief description of this method is given. While further details can be found in O. Ubbink [15] for VOF technique and in E. Didier [4] for Fully-Coupled formulation on unstructured grids.

Two dimensional flow case have been investigated to validate the accuracy of the formulation. First one is the liquid sloshing in a tank with various amplitude resulting to strong wave breaking [7] [2]. Second one is a collapsing of a liquid column [9] allows to illustrate the versatility of the current solvers and the ease to predict merging and rupturing interfaces. Finally, first results of wave propagation are showed here.

2. NAVIER-STOKES EQUATIONS

An unsteady Navier-Stokes solvers coupled with a Volume Of Fluid technique has been used to simulate air and water flow. The whole computational domain is given by both fluids (air and water) that are discretized using a one fluid formulation with changing physical properties (viscosity and density) across the interface. A conservative approach for Navier-Stokes equations is used in order to well maintain the mass of each phase.

$$\int_V \frac{\partial u_i}{\partial t} dV + \int_S u_i \left(\vec{u}.\vec{n}\right) dS$$
$$+ \int_S \frac{P}{\rho} \vec{i}_i.\vec{n}\, dS = - \int_S \frac{\mu}{Re\rho} \left(\vec{\nabla}\vec{u}.\vec{n}\right) dS \quad (1)$$
$$\int_V \frac{1}{Re\rho} \left(\vec{\nabla}\mu\right).\left(\vec{\nabla}.\vec{u}\right) dV + \int_V \frac{1}{Fr^2} \frac{\vec{g}}{g} dV$$

$$\int_S \vec{u}.\vec{n}\, dS = 0 \quad (2)$$

with

$$Re = \frac{U_o\, L\, \rho_1}{\mu_1} \qquad \text{et} \qquad Fr = \frac{U_o}{\sqrt{gL}} \quad (3)$$

2.1 Fully-Coupled discretization

In order to maintain a good accuracy a second order fully implicit discretization adapted for unstructured grids is used here associated with a Rhie and Chow [14]interpolation technique for discrete pressure equation to avoid checker board oscillations in pressure solution. Resulting linear system for the pressure, the velocities and the pseudo-velocities is solved exactly using BI-CGSTAB algorithm preconditioned by incomplete LU decomposition.

The whole unknowns system (U,V,P,U*,V*) is defined on the centers of each cell. and the set of pseudo velocities (U*,V*) is defined according to discrete Navier-Stokes equations (5).

$$U_{ip} + U^*_{ip} + \frac{g_{ip}}{a_p}.P_p + \sum \frac{g_{in}}{a_p}.P_n = 0 \qquad i \in \{1,2\} \quad (4)$$
$$U^*_{ip} + \sum \frac{a_n}{a_p} U_{in} = \frac{F_i}{a_p} \quad (5)$$

Pressure equation is obtained using the condition of incompressibility as follow :

$$P_p + \sum \frac{c_n}{c_p} P_n - \frac{\overrightarrow{b_p}}{c_p} . \overrightarrow{u_p^*} - \sum \frac{\overrightarrow{b_n}}{c_p} \overrightarrow{u_n^*} = \frac{F_p}{c_p} \quad (6)$$

and the resulting matrix of the whole discrete problem is presented above :

$$\begin{bmatrix} I & -I & G \\ C & I & 0 \\ 0 & D & -DG \end{bmatrix} \begin{pmatrix} U \\ U^* \\ p \end{pmatrix} = \begin{pmatrix} 0 \\ F_{U^*} \\ F_p \end{pmatrix} \quad (7)$$

Figure 1 show the convergence of the non-linear solution at each time step. the accuracy of fully-coupled algorithm allow a reduction of 10 order in 6 to 7 non linear iteration, that is absolutely imppossible to obtain using a weak coupling.

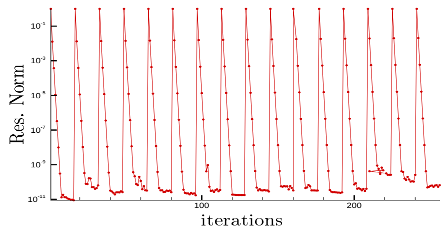

Fig.1 Convergence of the system during no-linear iteration

2.2 Volume Of Fluid method

Volume Of Fluid method need a new variable : the liquid concentration c and consequently a fourth equation in order to close the linear system : the convection of this concentration deduced directly from the mass conservation.

$$\rho = c\rho_1 + (1-c)\rho_2 \quad (8)$$
$$\mu = c_1 + (1-c)\mu_2 \quad (9)$$

and the advection equation:

$$\frac{\partial c}{\partial t} + \frac{\partial (cu_j)}{\partial x_j} = 0 \quad (10)$$

The discretization of the convective part of the volume fraction equation needs special attention. Its discretization should neither produce diffusion nor unbounded values. So the scheme must satisfy that the computed fluxes of volume fraction do not underflow or overflow the cells. If the first order upwind scheme, is unconditionally stable and always produce a bounded solution, it is to much diffusive to keep a small interface size. On the other hand the downing scheme is unstable but recover a sharp interfaces. Then it is interesting to use a special scheme using the latter properties. The CISCAM scheme adopted here have been developped by O. Ubbink [15]. This scheme is based on a Normalized Variable Approach (NVA), that allows complex mixing of high order scheme as QUICK or CBC [11, 8].

3. COMPUTATION PROCEDURE

The developed numerical method is applied to simulate several kinds of unsteady free surface flows, from simple to complex interface topologies. In the first application, an oscillating flow test case is used to evaluate the ability to conserve potential energy. In second application, results from a simple numerical wave tank is presented. At least an usual dam breaking with breaking free surface is compared with experiments.

3.1 2D sloshing tank

Tadjbakhsh & Keller [7] have developed a theory on sloshing liquid in a tank under the influence of gravity. In this case the fluid has initially an interface defined by a half cosine period with amplitude 0.005 m. The simulation domain is a structured mesh 0.1 m long and 0.1 m height. The fluids is respectively air and water without viscosity. The theoretical period of sloshing of the first mode is:

$$P_s = 2\pi\sqrt{gk\tanh(kh)} = 0.3739s \quad (11)$$

To compare theoretical result and simulation the figure 2 show plots of position of the interface at the left boundary against time for the six periods.

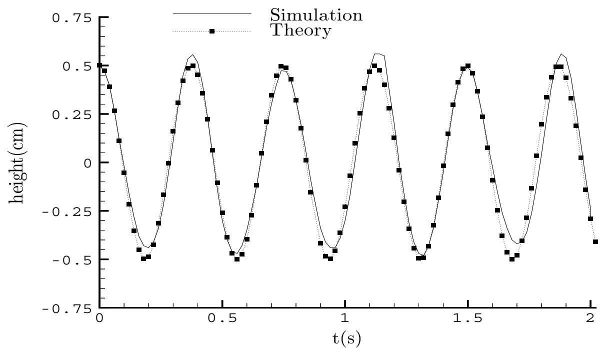

Fig.2 height of free surface on the left side

Another simulation of oscillating flow has been simulated. And has been compared to experiment [2]. A tank is moved with an horizontal velocity. The calculations were performed on a 0.4 m long and 0.2 m height structured and destructured mesh of about 3500 control volume.

$$u(t) = \begin{cases} A(sin(2\pi f_1 t) - sin(2\pi f_2 t)) & t \leq 3.43s \\ 0 & 3.43s \leq t \end{cases} \quad (12)$$

with $A = 7.5.10^{-3}$m, $f_1 = 1.598$Hz and $f_1 = 1.307$Hz

For this case, the exact interface shape are available at several time step. The figures 3,4,5, give a comparison of numerical and showed a good agreement.

The transfers between the potential energy and kinetic energy is correct and numerical diffusion seems to be neglected.

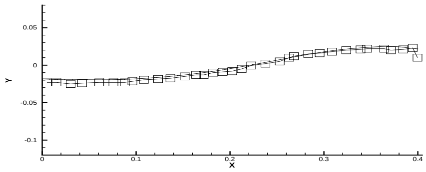

Fig.3 free surface at t=0.9625 s

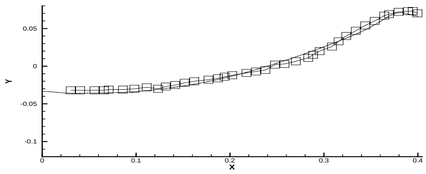

Fig.4 free surface at t=1.652 s

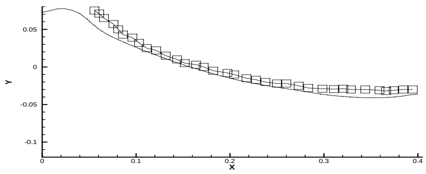

Fig.5 free surface at t=2.004 s

3.2 wave breaking

The collapse of a dam [9, 10] is a typical test case and demonstrate the ability of the method to compute transient fluid flow with free surface. The first one is made in a simple tank, and the second in a tank with obstacle.

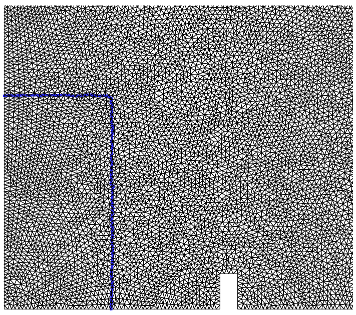

Fig.6 part of mesh

As showed on figure 6, the water column, with a base length of 0.146 m and a height of 0.292m located on the right of a 0.6 m long and 0.7 height tank. which are meshed by 22000 elements.

Unfortunately measurements of the exact interface shape are unavailable, but others data as the speed of the wave front (fig:9) and the reduction of the column height (fig:10) are presented.

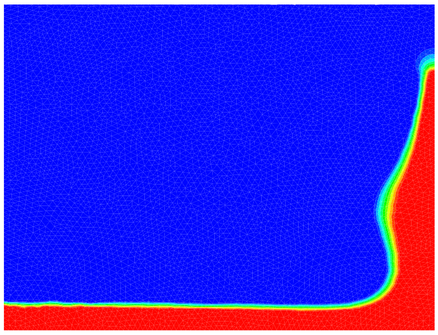

Fig.7 collapse of a dam: $t = 0.6$

Fig.8 collapse of a dam: $t = 0.6$

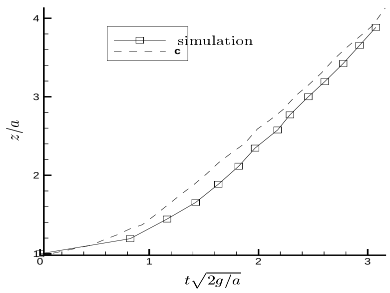

Fig.9 The position of the leading edge versus time

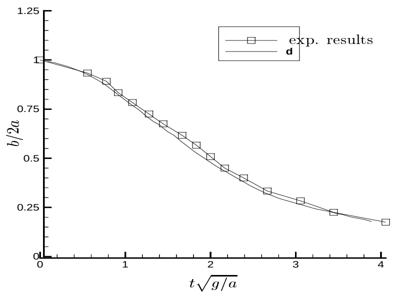

Fig.10 The height of the collapsing water column versus time

A more interesting version of a dam break occurs when an obstacle is located in the tank. As it is showed on figure 11 the free surface is particularly complex,after obstacle impact.

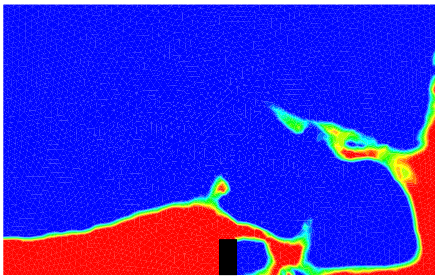

Fig.11 collapse of a dam with obstacle: $t = 0.65$

The comparison between experiments and calculation shows that the developed code can predict the behaviour of particularly complex free surface with a correct accuracy.

3.3 two dimensional wave tank

The main objective of this approach is to present a good wave propagation without amplitude damping or phase lag that are the two recurrent problems for CFD hydrodynamics simulations. The first step is to reproduce a monochromatic wave field, without breaking wave and check the accuracy with several validated formulation.

The domains is a 100m long and 15 m depth tank with a damping zone downstream the outlet boundary to avoid the wave reflection. A reffinement zone is used near the free surface in order to improve the transition between water and air.

Figure 12 shows a water fall picture of the wave propagation generated by a Dirichlet boundary condition on the velocities on the inlet boundary.

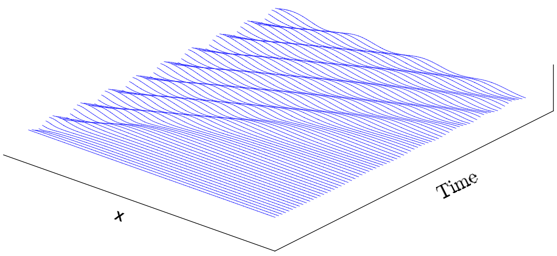

Fig.12 2D wave propagation test

4. CONCLUSION AND PERSPECTIVE

We have presentedhere a new numerical method with both advantages : The accuracy of the Fully Coupled algorithm first tested on Finite Differences convective formulation for tracking interface method and the robustness of Volume of Fluid capturing method to simulate very complex interfaces. The first two dimensional test cases show that we achieved our first objective. : the two dimensional solver is be robust and available over a wide range of hydrodynamics applications.

Near future work will validate the numerical wave tank on unstructured grids with breaking waves on a body before simulating three dimensional applications.

REFERENCES

[1] B Alessandrini and G Delhommeau. A multigrid velocity–pressure–free surface elevation fully coupled solver for turbulent incompressible flow around a hull calculations. *Proc 9th International Conference on Numerical Methods in Laminar and Turbulent Flows, Atlanta*, pages 1173–1184, 1995.

[2] P Corrignan. *Analyse Physique Des Phénomènes Associés Au Ballotement de Liquide Dans Les Réservoirs (Sloshing).* PhD thesis, Université de Nantes, 1994.

[3] M S Darwish. A new high–resolution scheme based on the normalized variable formulation. *Numerical Heat Transfer, Part B*, 24:353–371, 1993.

[4] E Didier. *Simulation D'écoulements À Surface Libre sur Des Maillages Déstructurés.* PhD thesis, Université de Nantes, 2001.

[5] M. Ferry. *Résolution Des Équations de Navier-Stockes Incompressibles En Formulation Vitesse-Pression Fortement Couplée.* PhD thesis, Université de Nantes, 1991.

[6] C W Hirt and B D Nichols. Volume of fluid (VOF) method for the dynamics of free boundaries. *Journal of computational physics*, 39:201–225, 1981.

[7] J.B. Keller I. Tadjbakhsh. Standing surface waves of finite amplitude. *J. Fluid Mech.*, 442-451:8, 1960.

[8] H Jasak, H C Weller, R I Issa, and A D Gosman. High resolution NVD differencing scheme for arbitrarily unstructured meshes. 0, 1996.

[9] W.J. Moyce J.C. Martin. An experimental study of the collapse of liquid columns on a rigid horizontal plane. *Philos. Trans. Roy. Soc. London*, A244:312–324, 1952.

[10] S Koshizuka, H Tamako, and Y Oka. A particle method for incompressible viscous flow with fluid fragmentation. *Computational Fluid Dynamics JOURNAL*, 4(1):29–46, 1995.

[11] B P Leonard. Bounded higher–order upwind multidimensional finite–volume convection–diffusion agorithms. *dans W.J. Minkowycz, E.M. Sparrow (eds), Advances in Numerical Heat Transfer, Chap. 1, Taylor and Francis, New York*, pages 1–57, 1997.

[12] M Perić. Basics of viscous flow CFD. *CFD for ship and offshore design, 31st Wegemt School*, 1999.

[13] M Perić and J H Ferziger. *Computational Methods for Fluid Dynamics.* Springer, second edition, 1997.

[14] C M Rhie and W L Chow. A numerical study of turbulent flow past an isolated airfoil with trailing edge separation. *AIAA Journal*, 21:179–195, 1983.

[15] O Ubbink. *Numerical prediction of two fluid systems with sharp interface.* PhD thesis, University of London, 1997.

Discussion Sheet

Abstract Title :	A fully-coupled VOF method for free surface simulation		
(Or) Proceedings Paper No. :	02	***Page :***	005
First Author :	Andrillon, Y.		
Discusser :	D. Howell Peregrine		

Questions / Comments :

In the dam break example the water falling down from the wall appeared not to hit the surface, but to be deflected. Why is this?

Author's Reply :
(If Available)

On the example showed during the presentation, the simulation has been made on an unstructured grid. And on this kind of grid the free surface zone is defined on about eight or ten elements. So as only the 0.5 value contour of the fraction of volume is shown, the water falling seems to be deflected, even though the thick interface zone of the water falling actually hits the still surface. In fact, this problem comes from a bad visualisation of the numerical solution.

Questions from the floor included; Touvia Miloh & Guo Xiong Wu.