# ON THE INFLUENCE OF BREAKING WAVE LOCAL GEOMETRY ON IMPULSIVE LOADS

Stéphane Abadie and Cyril Mokrani [1]

In this paper, we study the wave impact process with a multi-fluid Navier-Stokes model (THETIS). Preliminary simulations have been conducted, first on a plunging wave generated by unstable Stokes initial condition, and second, involving a dam breaking bore impact. In both cases, a convergence study shows pressure peak results instability when using different meshes. This is due to the incapacity of the model to ensure, after a certain time of computation, the exact same surface profile at impact when simulating a specific case with different meshes. This instable numerical behavior is somehow similar to peak pressure instabilities observed in experiments. This similarity shows the critical role played by local free surface shape at impact on impulsive loads. When initializing the model with a specific interface right at impact, convergence is observed and the pressure peaks are correctly assessed by the code for moderate intensity impact. However, further improvements are still needed especially regarding the interface tracking technique to simulate the most violent impacts involving the weaker dead rise angles. The paper also encourages us to use numerical simulations preferably to study impact flow at local scale.

*Keywords: wave impact, impulsive loads, pressure peaks, numerical modeling, Navier-Stokes, Volume of Fluid.*

## INTRODUCTION

Impulsive loads are generated when steep or breaking waves strike vertical obstacles, generating sudden and high pressure peaks followed by a more variable decay. A better understanding of this phenomenon is key for reducing the uncertainties when designing coastal structures in general and for the case of vertical breakwaters in particular (Oumeraci, 1994). The unstable nature of the phenomenon of impact has been identified for a long time when experimental studies showed that same wave conditions were able to create very different pressure signals (e.g., Witte, 1988). Two main phenomenon are recognized as being responsible for this particular behavior : first, the local free surface geometry during impact, largely studied in slamming or sloshing related studies (e.g. Brosset et al., 2011); second, the effect of the entrapped air, which may play a role as a compressible gas pocket (Lugni *et al*., 2010) or by enhancing the compressibility of water (Plumerault *et al*., 2012). Compared to laboratory studies, numerical modeling can provide an easier way to study each process separately with finer temporal and spatial resolution. But before being used as investigation tool, models have to be carefully validated. In this paper, we used an incompressible Navier Stokes model to better understand the role of free surface in wave impact. As we shall show, the question of validation finally turned out to be related to the unstable nature of the phenomenon.

## DESCRIPTION OF THE MODEL

The model THETIS is developed by University of Bordeaux I and CNRS in France. It is a research CFD code, solving the Navier-Stokes equations in 1-fluid formulation with a VOF interface tracking (Youngs, 1982). In this paper, water and air are simulated. In a 1-Fluid formulation, this can be achieved simply by variable density and viscosity in the equations. The VOF method appeared to play a crucial role in the results of impact simulations. The accurate second order PLIC-VOF method can be used alone in THETIS, nevertheless, violent impact flows may induce interface fragmentation generating multiple droplets that the code later cannot properly resolve. For that reason, a slight smoothing of the interface is carried out at each time step to avoid fragmentation. This procedure is controlled so that the interface thickness remains lower than a certain portion of the mesh cell (typically two mesh cells). To create vertical obstacles which will be submitted to wave impact, we used, either the limit of the domain with slip or no slip conditions, or the fictitious domains with penalty methods based on viscosity or permeability. Navier-Stokes equations are discretized on a velocity-pressure staggered cartesian mesh using the finite volume method. Once discretized, the incompressible Navier-Stokes equation is solved by an augmented Lagrangian algorithm. Time step is automatically calculated based on a CFL condition with a Courant number of order 0.1 to 0.5.

The model has been extensively validated in several studies among which some are related to waves (e.g., Abadie *et al*., 1998; Lubin *et al.*, 2006; Abadie *et al.*, 2010 ; Mory *et al*.,2011).

[1] IVS, EA 4581 SIAME, UPPA, Allee du Parc Montaury, 64600 Anglet, FRANCE.

## BASIC IMPACT PHYSICS

Before analyzing the simulation results, one need to highlight the link between the free surface and the impact magnitude as this link was key to interpret our simulation results. Moreover the simple physics, which will be explained hereafter, also allows to clarify the phenomenon of wave impact even

if this latter also depends on the behavior of the air phase. Let us consider the problem of the triangular jet sketched on Figure 1. At t=0, the flow exhibits an inclined interface between water and air making an angle $\alpha$ with the horizontal direction and the water mass has an initial velocity $V_0$. After a small time duration dt, the interface deforms due to the presence of an obstacle. The water volume $\Omega$, which goes up the obstacle, can be easily calculated by the considering the virtual triangular volume, which would cross the plane x=0 without the presence of a wall. The horizontal side of that triangle is equal to $V_0dt$ while the other one equals $V_0dt\tan(\alpha)$ leading to $\Omega=1/2(V_0dt)^2\tan(\alpha)$. $\tan(\alpha)$ tends to infinity as $\alpha$ tends to $\pi/2$, so the volume $\Omega$ also tends to infinity. This is a first way to understand why head-on impacts are so violent.

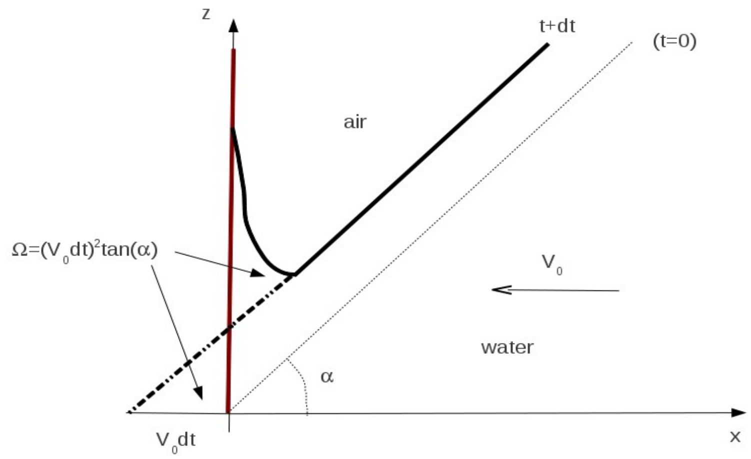



**Figure 1 : Triangular jet flow. Dotted line : initial conditions, thick black line : free surface at time t=dt, dotted-dashed line : free surface at t=dt without obstacle.**

In Figure 2, we now analyze the triangular jet same flow in terms of velocity, acceleration and pressure. Vertical velocity of geometrical point A is equal to $V_0\tan(\alpha)$. Water particles (B) just above A then have larger velocities so that A would not cross B with time. Before being in $\Omega$ at t=dt, these particles were only moving horizontally so that in dt their vertical acceleration can be assessed as $V_0\tan(\alpha)/dt$. Finally, Navier-Stokes equations written on the vertical axis neglecting gravity reads :

$$\rho\frac{DV}{Dt}=-\frac{\partial P}{\partial z} \qquad (1)$$

This equations means that a vertical acceleration needs a pressure gradient to be generated, hence, the existence of a pressure peak. Here again, vertical acceleration and pressure gradient increase with $\alpha$ , the flow presenting a singularity for $\alpha=\pi/2$ (i.e; acceleration and pressure gradient being infinite).

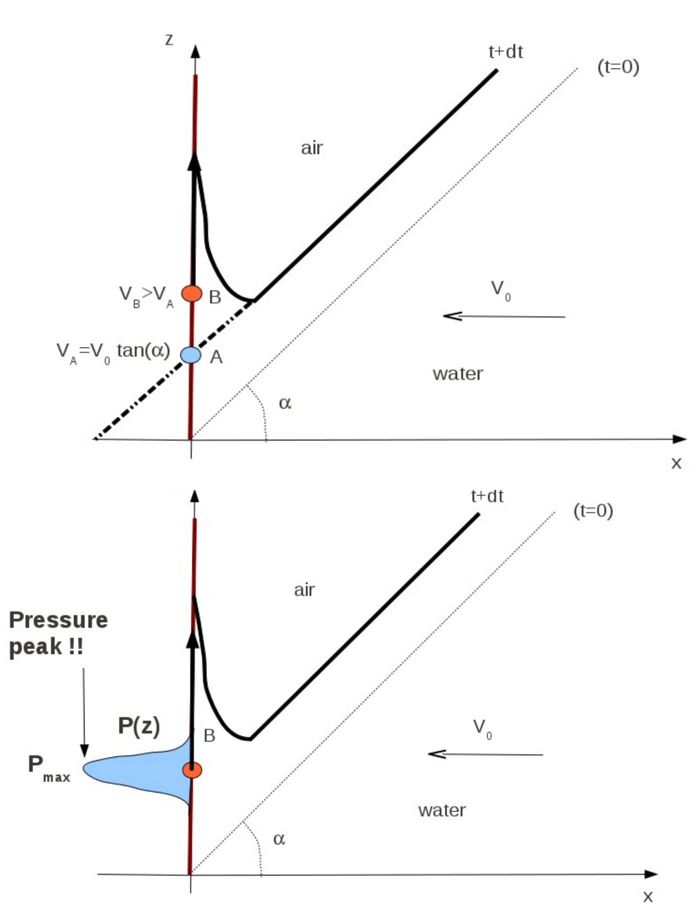



**Figure 2 : Triangular jet flow. top panel : velocity of particular points at the wall, bottom panel : pressure gradient necessary to accelerate the flow in B.**

## SIMULATION RESULTS

### Plunging breaking wave

The first physical problem which was simulated with THETIS is sketched in Figure 3a. A plunging breaker is obtained by initializing the computation with a first order Stokes wave with initial parameters $H_0 = h_0$ =1.3 m, $T_0$=3.024 s (respectively wave height, water depth and wave period). The large value of the initial wave steepness (i.e., 0.13) makes the computed wave rapidly become unstable and develop a violent plunging breaker jet. Note that this particular case was already studied in Vinje and Brevig (1981) using a Boundary Element Method. Abadie *et al.* (1998) showed that THETIS was able to correctly reproduce the kinematics and dynamics found in Vinje and Brevig (1981). A direct comparison of free surface profiles can also be found in Mokrani *et al.* (2010).

Simulations of impact flow (e.g., Figure 3b) have been done in two stages. First, the Stokes wave propagates in the domain, assuming lateral periodic boundary conditions (open boundary condition at the top limit and free slip on bottom). Once breaking point is reached, (i.e., when inflection point of the free surface profile has a vertical tangent) wave characteristics are used to initialize the impact simulation in a similar domain but this time, with free-slip lateral conditions instead of periodical. Moreover, an obstacle is placed at a controlled distance d from the breaking point. With this simple method, various types of impact can be easily generated and the influence of the breaker shape (i.e., by varying initial wave parameters) and relative position of the obstacle on pressure values generated on the wall, studied. This was the initial goal of these simulations (Mokrani *et al.*, 2010).

**a)** **b)**

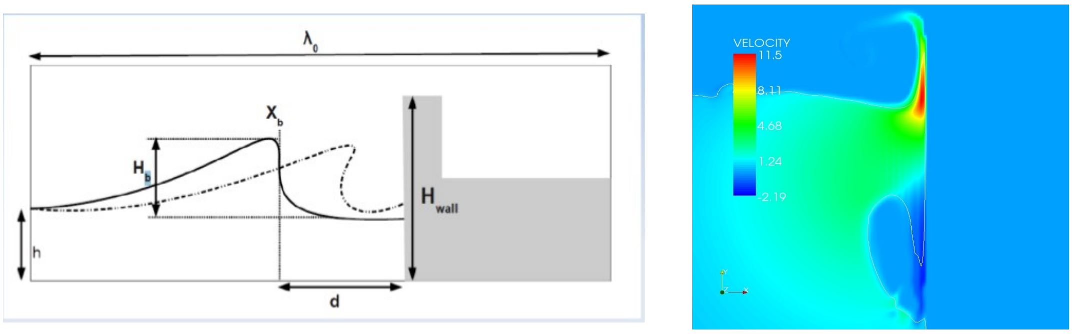
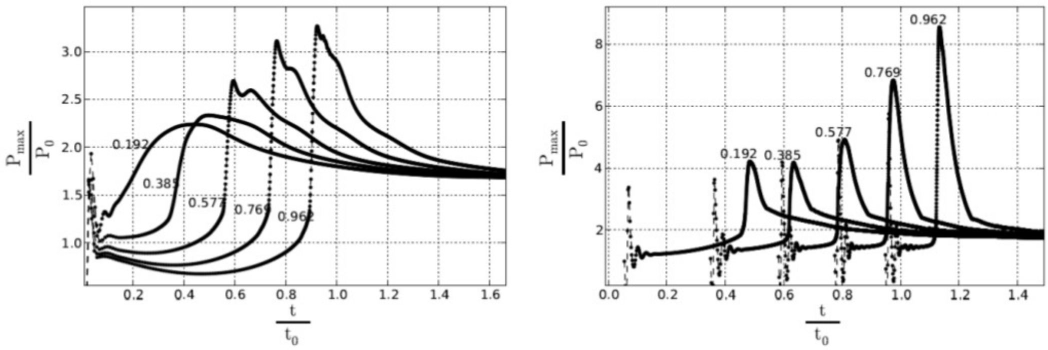



**Figure 3 : Numerical simulation of a plunging breaking wave impact on a vertical wall. a) Sketch of the case considered with parameters definition. b) snapshot of the flow velocity during impact.**

Figure 4 presents the time evolution of the maximum pressure on the wall for different values of d (i.e., distance between breaking point and obstacle). On Figure 4a, a relatively coarse mesh was used. Normalized pressure peaks calculated increase slightly from 2.2 to 3.3 with increasing distance d. With a finer mesh (Figure 4b), the trend is the same, however the values of pressure peaks differ significantly. This of course rises the question of convergence of the computations.




**a)** **b)**

**Figure 4 : Maximum value of normalized pressure ($P_0=\rho c_0^2$ with $c_0$ initial wave celerity) on the wall with non dimensional time ($t_0=h/c_0$) for different positions of the wall relatively to the breaking point (values of d*=d/h are indicated on each graph). a) : coarse mesh with $\Delta x^*=\Delta x/h=\Delta z^*=3.84\text{x}10^{-2}$, b): finer mesh with $\Delta x^*=\Delta z^*=9.61\text{x}10^{-3}$.**

We then fixed the obstacle position to d*=d/h=1.538 and proceeded to a convergence study by using finer and finer meshes with focus on the maximum value of the pressure on the wall. Results are presented in Figure 5. They show that there is no limit to the maximum pressure obtained in the mesh range tested. Computation divergence can be demonstrated if we assume the error to be a power function (Richardson, 1911).

## Dam breaking case

The case of a dam breaking with subsequent impact on a vertical wall has been largely studied in the literature. Experimentally, the flow is easily generated and as a “unique” event, the flow may be more “reproducible” than for instance in the case of irregular waves impacts. Here we tried to simulate with THETIS the simple 2D case studied in Hu and Kashiwagi (2004) (Figure 6). In this experiment, a pressure measurement is performed on the wall one centimeter above the bottom. Data corresponds to the average value calculated over height similar experiments. For initial condition given in Figure 6, Hu and Kashiwagi (2004) measured $P_{max}=1.388P_0$ with $P_0=\rho c_0^2$ and $c_0=\sqrt{gh_0}$ .

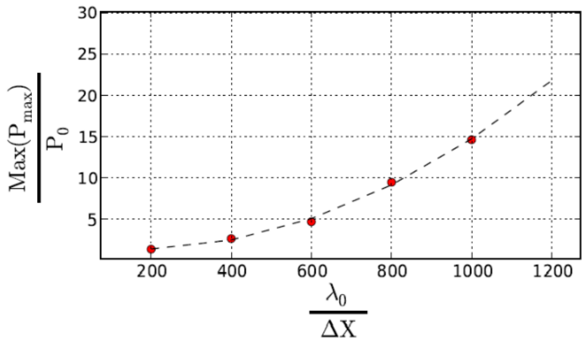



**Figure 5 : Normalized maximum pressure ($P_0$ defined in Figure 4) calculated by THETIS (red circle) for different meshes and parabolic interpolation of the results (dashed line).**

This case is simulated with THETIS using free-slip boundary conditions. Convergence is studied by using finer and finer meshes (i.e., from 128x128 to 522x522). In all simulations, vertical resolution is four times the horizontal resolution.

Results are presented in Figure 7. Figure 7a shows the different pressure signals obtained with various meshes. Here again the bigger the mesh, the higher the peak. On contrary, pressure rising times seem inversely proportional to mesh size. Only the coarser meshes give acceptable values as compared to experimental data. Like in Figure 5, Figure 7b shows the divergence of the pressure peak calculated. Interestingly, the pressure impulse obtained by integrating the pressure over the impact duration is shown to be convergent (Mokrani, 2012). Another conclusion in Mokrani (2012) is that pressure peak calculation doesn't diverge at every location on the wall. Indeed, divergence is only observed in a range of 2 cm with center point A. One centimeter above point A, pressure peak calculated converges toward a stable value when refining the mesh.

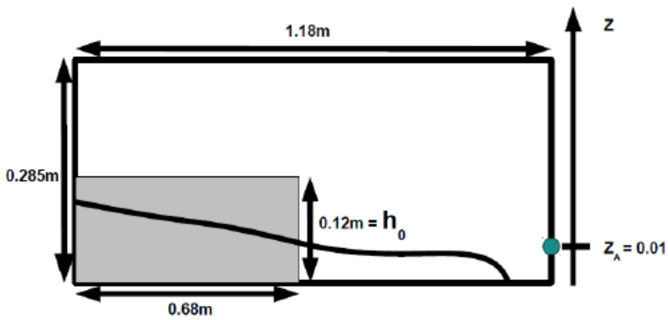
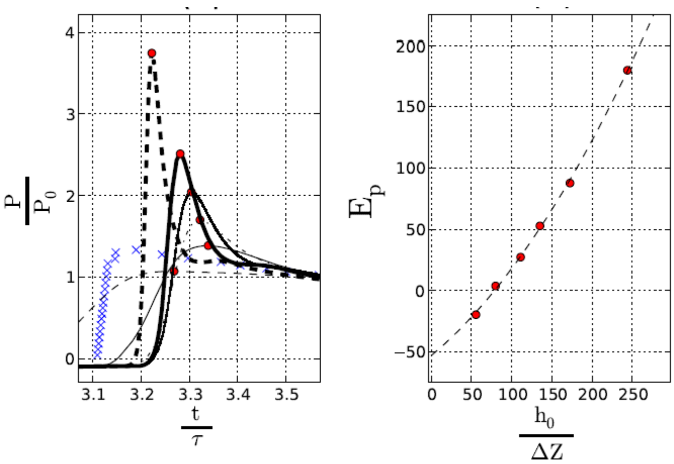



**Figure 6 : sketch of the experiment carried out by Hu and Kashiwagi (2004) and simulated in this paper with THETIS. Point A is the location of the pressure sensor during the experiment.**

## Local interfaces at impact

The case of triangular jet has been analyzed before in this paper and we highlighted the importance of the interface dead-rise angle at impact on the pressure gradient and hence, on the generated pressure peak. Even though breaking wave or dam breaking impacts involve curved interfaces, the process of pressure generation should not be too different from the one described in the linear interface triangular jet case. So the idea here is that differences in calculated pressure peaks may be due to differences in interface local slope at impact for both cases tested.

Indeed, Figure 8 shows a plot of the interface right before impact in the dam breaking case. We recall that the pressure signal is taken at z=1 cm. Obviously at this elevation, there are variations of the interface angle, the front being steeper and steeper with mesh size increasing. Mokrani (2012) shows that theses variations of angle are significant with a lower value around 20° for the coarsest mesh to about 50° for the finest. Such values are consistent with the strong increase observed in the error between simulated and measured pressure peak (Figure 7b).




**a)** **b)**

**Figure 7 : (a) : Time evolution of pressure at point A for different meshes : (- -) : 128x128, (–) : 192x192, (-.-.) : 256x256, (...) : 320x320, (– ) : 384x384, (- - ) : 522x522, (• ) : Maximum values calculated , (x x ) : pressure measurement at A from Hu and Kashiwagi (2004) - (b) : (. . ) : Relative error (in %) on the pressure peak for different meshes, (− − ) : parabolic interpolation of the results.**

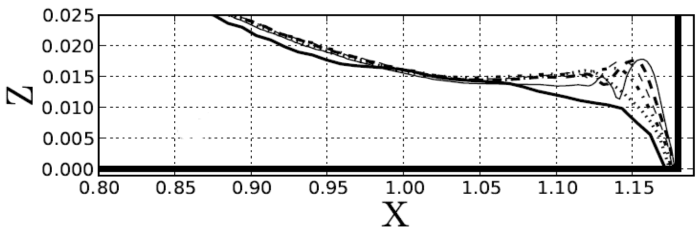
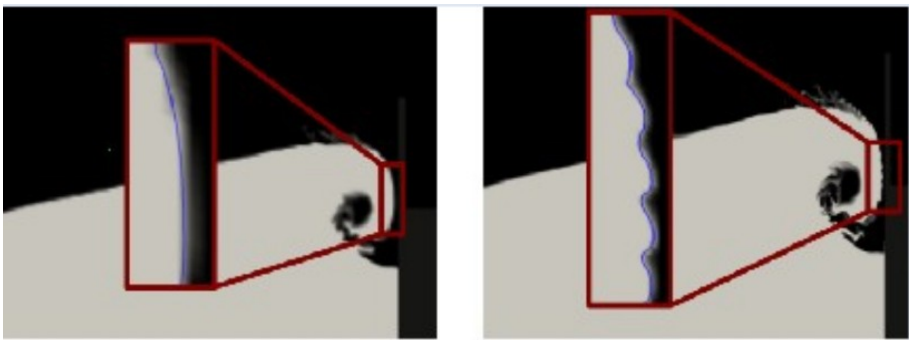



**Figure 8 : Water air interface just before impact for different meshes in the dam breaking case. (- -) : 128x128, (–) : 192x192, (-.-.) : 256x256, (...) : 320x320, (– ) : 384x384, (- - ) : 522x522.**

The same conclusion can be drawn for the breaking wave case (Figure 9). With interfaces being different from one mesh to the other, the pressure peak can not be the same. For this particular case we not only found differences in the average angle between interface and wall, but we also observed smaller scale differences of the interface shape; the most resolved interface (Figure 9b) being affected by Kelvin-Helmoltz instabilities due to the strong shear flow generated by upward air expulsion. [Note that a similar phenomenon has been already observed in a flip through impact by Brosset *et al.* (2011)].



**a)** **b)**

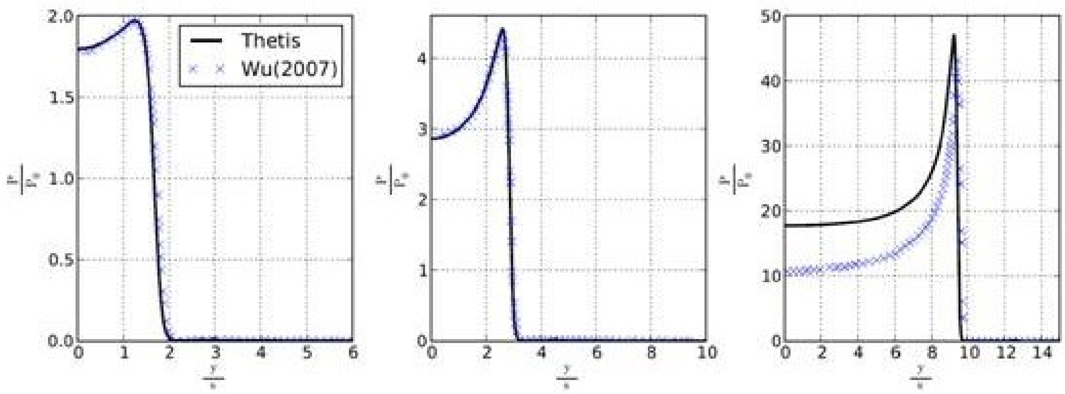
**Figure 9 : Local water air interface right before impact for two meshes in the breaking wave case. a) ΔX=ΔZ = 0,05m, b) ΔX=ΔZ =0,0125m**

So it appears very difficult to obtain exactly the same interface shape just before impact when simulating the same case with two different meshes. As the process of impulsive pressure is very sensitive to this local shape, simulation of the same flow can generate significantly different pressure values.

**Triangular jet**

However to interpret the model results, we still have to demonstrate its ability to accurately reproduce a pressure signal for a given interface at impact. To verify this point, we considered the triangle jet case shown Figure 1. Wu (2007) solved this self similar problem analytically and numerically using a Boundary Element Method. Consistency between both methods is shown in his paper.

Figure 10 shows the numerical results obtained by Wu (2007) (in blue). The signal is plotted in a self similar abscissa (with $s=V_0t$) for which only one solution exists for a given angle α. The values obtained by Wu (2007) show the strong increase in the pressure maximum generated by the jet flow with increasing incidence angles consistent with the explanations given in the third section. [For example, for α=80°, the pressure peak reaches more than forty times the reference pressure ($\rho V_0^2$)].




a) b) c)

**Figure 10 : Normalized pressure distribution on the wall for different initial angle in the triangular jet case. a)α=45°; b) α=60°; α=80°. $P_0=\rho V_0^2$. ΔX=ΔZ=0,02m**

The triangular jet impact is simulated with THETIS. At t=0, an inclined interface is imposed just upstream the wall with initial velocity $V_0$ like in Figure 1 (dotted line). Free slip boundary conditions are imposed on the bottom and on the wall. Some of THETIS results are plotted and compared with Wu (2007) in Figure 10. For α=45° and α=60° the agreement between the Navier-Stokes (THETIS) and the BEM solutions is very good. Actually, Mokrani (2012) shows that THETIS results respect the self similarity properties of the flow and matches Wu (2007)' solutions up to 70° (not shown here). From this limit on (i.e., 80°<α<90°), pressure signal calculated with THETIS differs from Wu's solution and is no longer self similar. The same problem appeared first for α=70° but was successfully fixed by adjusting the interface thickness, which seems to be a key parameter in the numerical model. Unfortunately, the solution that worked for 70°, didn't fix the problem at 80°. Obviously, beyond 80°, the violence of the flow (i.e., strong changes in a short time scale) makes it more difficult for THETIS to reach an accurate solution. So far, no solution has been found to overcome this difficulty.

## CONCLUSIONS

In this study, we have shown the strong link that exists between local free surface shape at impact and instantaneous pressure signal. This link is of course physical as highlighted by the triangular jet flow.

For numerical simulations, this implies that numerical methods unable to ensure very reliable interface tracking over a long duration, won't be able to ensure convergence of the results and as for instance demonstrated here, two meshes will induce different pressure signal in the same physical case simulated. So far, we are not aware of any numerical study related to impact in which the stability of the pressure peak results has been really demonstrated.

The model, which was tested here, was however found to be reliable for assessing the pressure signal when the interface was imposed just before impact (i.e., skipping the impact jet formation stage). In that case, the results are accurate for say, "moderate" impact case. When the angle between the interface and the wall is small (i.e., <10°), the flow is more violent and Navier-Stokes simulations show errors that become significant. For such flows, we think that the numerical method used to perform the interface tracking should play an important role in this problematic behavior.

**REFERENCES**


Abadie S. , Caltagirone J. P. & Watremez P. 1998. Splash-up generation in a plunging breaker. *Comptes Rendus de l'Académie des Sciences - Series IIB - Mechanics-Physics-Astronomy*, 326, 9, 553-559.

Abadie, S., Morichon, D., Grilli, S., Glockner, S. 2010. Numerical simulation of waves generated by landslides using a multiple-fluid Navier-Stokes model. *Coastal engineering*, 57, 779-794.

Arai M., Cheng L., Kunamo A., Miyamoto T. 2002. A technique for stable numerical computation of hydrodynamic impact pressure in sloshing simulation. *Journal of the society of naval architects of Japan*, 191, pp 299–307.

Brosset , L,. Lafeber , W., Bogaert , H., Marhem , M., Carden , P., Maguire, J. 2011. A Mark III panel subjected to a flip-through wave impact: results from the Sloshel project. Proc. 21st International Offshore and Polar Engineering Conference. 84-96.

Chen Y., Djidjeli, K., Price W. 2009. Numerical simulation of liquid sloshing phenomena in partially filled containers. *Computers & fluids*, 38, 830–842.

Hu, C. & Kashiwagi, M. 2004. C.I.P. method for violent free-surface flows, *Journal of Marine Science and Technology*, 9, 143–157.

Lubin, P., Vincent, S., Abadie, S. & Caltagirone, J.P., 2006. Three-dimensional Large Eddy Simulation of air entrainment under plunging breaking waves, *Coastal Engineering*, 53, 8, 631-655.

Lugni C., Miozzi M., Brocchini M., Faltinsen O.M. 2010. Evolution of the air cavity during a depressurized wave impact. I. The Kinematic Flow Field. *Physics of fluids*, 22, 56-101.

Kleefman K., Fekken, G., Veldman A., Iwanoski B., Buchner B. 2005. A volume-of-fluid based simulation method for wave impact problems. *Journal of computational physics*, 206, 363–393.

Mokrani C. 2012. Impacts de vagues déferlantes sur un obstacle vertical. Modèle théorique et estimation numérique des pics de pression. *PhD Thesis*, Université de Pau et des Pays de l'Adour. (in french)

Mory, M., Abadie, S, Mauriet, S. and Lubin, P. 2011. Run-up flows of collapsing bores over a beach, European Journal of Mechanics – B/Fluids, 30, 6, 565-576.

Plumerault L-R., Astruc, D., Maron, P., 2012. The influence of air on the impact of a plunging breaking wave on a vertical wall using a multi-fluid model. Coastal Engineering, 62, 62-74

Richardson L.F. 1911. The approximate arithmetical solution by finite differences of physical problems including differential equations with an application to the stresses in a masonry dam. *Philo Trans of Royal Soc of London*, Series A, 210, pp 307-357.

Vinje, T. and Brevig P., 1981. Numerical Calculation of Breaking Waves", J. Adv. Water Res., 4, 77-82.

Witte H.H. 1988. Wave induced impact loading in deterministic and stochastic reflection. *Mitt Leichtweiss Inst. Wasserbau*, vol. 102, Tech. Univ. Braunschweig, Braunschweig, Germany.

Wu G. 2007. Fluid impact on a solid boundary. *Journal of fluids and structures*, Vol. 23, pp 755–765.

Youngs D.L. 1982. Time-dependent multimaterial flow with large fluid distorsion. K.W. Morton and M.J. Baines, *Numerical methods for fluids dynamics*, Academic Press, New York.

Yettou El-M., Desrochers A., Champoux Y. 2007. A new analytical model for pressure estimation of symmetrical water impact of a rigid wedge at variable velocities. *Journal of fluids and structures*, 23, pp 501-522.