# EVALUATION OF SLAMMING LOADS ON SHIP BOW SECTION ADOPTING SPH AND RANSE METHOD

Viviani Michele[(1)], Savio Luca[(2)], Brizzolara Stefano[(3)]

[(1)]Department of Marine Technology (DINAV), University of Genoa, Via Montallegro 1, 16148 Genova, Italy, phone: +39 010 3532547; fax: +39 010 3532127; e-mail: viviani@dinav.unige.it
[(2)]Department of Marine Technology (DINAV), University of Genoa, Via Montallegro 1, 16148 Genova, Italy, phone: +39 010 3532438; fax: +39 010 3532127; e-mail: luce.lux@tiscali.it
[(3)]Department of Marine Technology (DINAV), University of Genoa, Via Montallegro 1, 16148 Genova, Italy, phone: +39 010 3532386; fax: +39 010 3532127; e-mail: brizzolara@dinav.unige.it

## ABSTRACT

Different numerical methods have been applied to the evaluation of slamming loads on a typical ship bow section, for which experimental data of drop tests has been made available within MARSTRUCT thematic network. Experimental data cover different drop cases, with variable speed and different heel angles. Numerical techniques adopted are a commercial CFD VOF program and Smoothed Particles Hydrodynamics Method (SPH) currently under development at DINAV; results from both methods are presented and compared with available experimental data, allowing to make an assessment of these different techniques and to stress their relative merits and shortcomings.

Regarding SPH, most of calculations are performed using a conventional treatment of boundaries with repulsive forces, while a few tests have been performed with a novel boundary treatment which adopts variable smoothing length ghost particles (VSLGP), with promising results in terms of computational efficiency.

*Keywords*: impact load, slamming, SPH, RANSE

## 1 INTRODUCTION

This work deals with calculations performed in order to analyse slamming phenomenon on ship bow sections; slamming is a typical phenomenon in naval architecture field, and in particular it is related to violent impacts of ship's bottom on the sea surface after emersion due to motions in waves; this results both in high local dynamic pressures and in possible transient vibratory stresses (whipping) in the hull. All these effects strongly influence structural design of ships, especially for high-speed vessels which are more likely to experience high motions and accelerations and therefore are more prone to slamming and subsequent high pressures.

The problem of slamming impact on ship has been investigated for a long time (see for instance Wagner 1932 , Chuang 1967, Stavoy and Chuang 1976, Zhao and Faltinsen 1993), both theoretically and experimentally; nevertheless, this problem is still under investigation, due to the intrinsic difficulties which arise from the high non-linearities involved and to the complex interactions between free-surface, hull sections and air; difficulties are further increased in correspondence to discontinuous hull shapes (hard-chines, spray-rails and so on).

One of the most important problems related to slamming impact is linked to the strong variations of free surface, with development of sprays and consequent high non-linearity, which make free surface treatment more difficult with conventional methods which require a mesh or a grid. Meshless methods, like SPH, with their Lagrangian approach allow to overcome this problem, with a much faster generation of the input data for the problem and their intrinsic ability to treat free surfaces.

The present paper specifically addresses pressures calculation on 2D sections falling in smooth water; this simplified approach is frequently adopted for the evaluation of pressures

due to slamming, both numerically and experimentally; calculation results are compared with experimental "drop tests", in which cylindrical bodies are let free to fall into water.

## 2 EXPERIMENTAL RESULTS

Calculations reported in the present work have been performed in the context of MARSTRUCT thematic network in order to compare with experimental results of free-falling drop tests on a typical bow section, whose data have been made available from Marintek (Aarsnes 1996 and Zhao et al. 1997). In following figures 1 and 2, details of the experimental setup are reported; during tests, pressures time histories in correspondence of measuring points indicated as P1 – P4 have been recorded by means of pressure cells having a diameter of 3 mm; moreover, vertical and lateral forces on the central measuring section (length 100 mm, see figure 2) have been recorded.

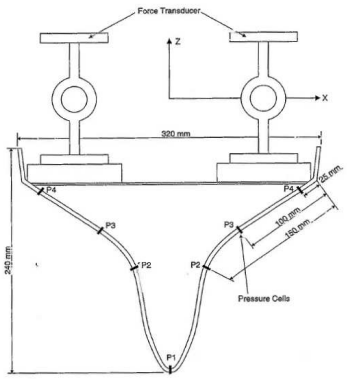



Figure 1: detail of ship bow section and experimental setup (Aarsnes 1996)

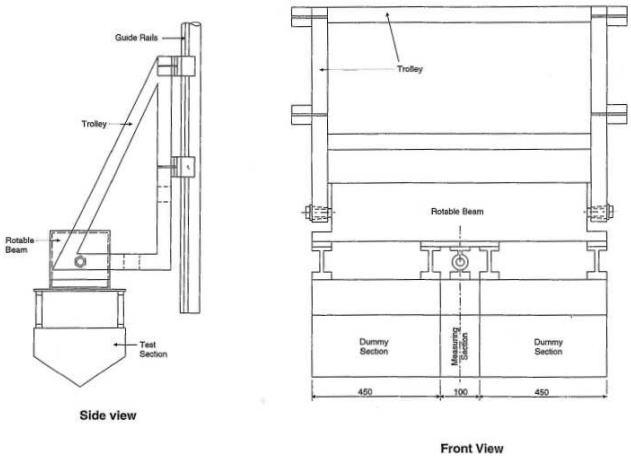



Figure 2: sketch of the free falling rig (Aarsnes 1996)

Tests were conducted in order to analyse the effect of drop speed and transversal (fixed) heel angle; conditions analysed in present work are reported in table 1; drop speed reported is recorded at impact instant, while during the free fall into water it evolves in time.

| Test n° | Drop speed [m/s] | Heel angle [deg] |
|---|---|---|
| 216 | 0.58 | 0 |
| 225 | 0.61 | 9.8 |
| 268 | 0.61 | 28.3 |
| 261 | 2.43 | 0 |
| 223 | 2.43 | 9.8 |
| 265 | 2.43 | 28.3 |

Table 1: tests analysed

As an example, in following figures 3 and 4 drop speed time history recorded during tests 216 and 261 is reported. It can be seen clearly that drop speed presents significant variations during experiment especially in correspondence to tests with lower initial value, while in correspondence to the higher initial value variations are more limited.

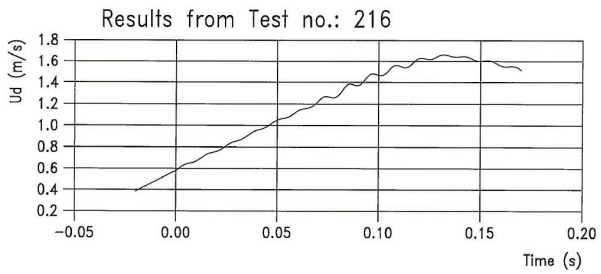



Figure 3: Drop speed time history in test n°216 (Aarsnes 1996)

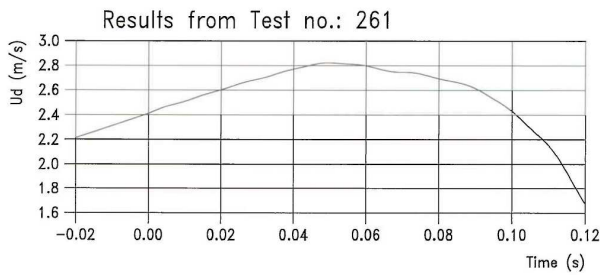



Figure 4: Drop speed time history in test n°261 (Aarsnes 1996)

## 3 DESCRIPTION OF METHODS ADOPTED

In following paragraphs 3.1 and 3.2 a brief description of methods adopted is reported.

### 3.1 SPH METHOD

Smoothed Particle Hydrodynamics is a relatively novel Lagrangian meshless CFD method. The method, initially developed for compressible fluids (Lucy, 1977; Gingold and Monaghan, 1977), must be adapted and 'corrected' for hydrodynamic problems (Monaghan et al, 1994, Liu and Liu, 2003). The continuum is discretised in a number of particles, each one representing a certain finite volume of fluid, which are followed (in a Lagrangian way) during their motions induced by internal forces between nearby interacting particles and external mass forces or boundary forces. Internal forces derive from the usual Navier-Stokes and continuity equations made discrete in space by means of a kernel formulation. Moreover, an equation of state (reported hereunder) which relate density to pressure for each particle is adopted, thus considering the fluid flow as compressible:

$$p = \frac{c_0^2 \rho_0}{\gamma}\left[\left(\frac{\rho}{\rho_0}\right)^{\gamma} - 1\right]$$

where the value of sound speed $c_0$ has been set in order to limit the Mach number to a value below 0.1 (Monaghan 1994) and consequently density variations to values less than 1%.

Space integrals of a hydrodynamic variable over the fluid domain are discretised by means of a 'kernel estimate' of the field variable (Libersky et al, 1993, Liu and Liu 2003); in the present work, Gaussian kernel has been chosen among those adopted by different authors.

Moreover, since SPH can be affected by a lack of stability, artificial viscosity (Monaghan 1992) and XSPH (Monaghan 1992) have been adopted.

For what regards boundary treatment, use of virtual particles and a modified repulsive force with respect to (Monaghan 1994) has been considered. In particular, in the original formulation, repulsive force is dependent to inverse of the distance between fluid and boundary particles according to Lennard-Jones formulation for molecular force

This formulation has been modified (Viviani et al. 2006) in order to make the force act in the direction normal to boundary surface and to avoid forces oscillations (in magnitude and/or direction) along the surface itself. In particular, for each real particle in proximity of the boundary the nearest virtual particle is searched, the repulsive force is evaluated according to their distance in direction normal to the boundary with the modified formulation reported (and not according to the radial distance), and the force itself is applied in the same normal direction, as represented in figure 5.

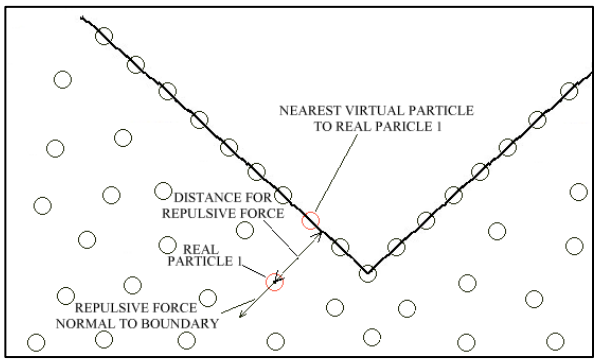



Figure 5: Scheme for repulsive force calculation

$$f(r) = \begin{cases} D\dfrac{\vec{n}}{r_n}\left(\left(\dfrac{r_0}{r_n}\right)^{p_1} - \left(\dfrac{r_0}{r_n}\right)^{p_2}\right), & when\ \ r_n < r_0 \\ 0, & elsewhere \end{cases}$$

where $p_1$=12, $p_2$=6, $r_0$ is the cutting off distance, approximately equal to the smoothing length, $r_n$ is distance in radial direction and $\vec{n}$ is normal vector

Finally, in order to evaluate pressure an approach similar to the one presented in (Oger et al. 2005) is adopted, somehow simplified since pressure is evaluated as a mean of values on real particles in proximity of each boundary particle (namely in a rectangular region with

width and height parallel and perpendicular to boundary surface).

In the present calculations, a “pool” 1.2 m wide and 0.6 m high is used, with a particle radius equal to 1.75 mm, resulting in about 250000 particles equally spaced; fixed boundary particles (pool walls and bottom) are spaced in the same way, while moving particles of the 2D falling section have a finer spacing (1 mm); time step adopted is different for various tests, according to Courant condition.

### 3.2 RANSE METHOD

The RANSE solver used is a commercial finite-volume-finite-differences method that solves the non stationary Navier-Stokes equation in a fixed Eulerian rectangular grid, representing objects by means of an original technique, the Fractional-Area-Volume-Obstacle-Representation (FAVOR™) method (Hirt and Sicilian, 1985). Such technique allows for the definition of solid boundaries within the Eulerian grid and determines the fractions of areas and volumes (open to flow) in partially blocked cells, for the computation of flows correspondent to those boundaries. In this way, the process of defining boundaries and obstacles is done independently of grid generation, avoiding saw-tooth representation typical of Eulerian grid solvers or the use of body fitted grids used in most of the other CFD codes. In this way, the body geometry can be defined in its most general shape by use of an STL surface discretized description.

An example of the simulated domain used for the calculation presented is given in figure 6. It spans 3.0 m in width and has a fixed bottom at 1.1 m from the free surface at rest. The grid has been refined along the transversal and vertical directions close to the section bottom in order to accurately capture the development of inner and outer domain of the flow, (local rising up of the free surface and the jet-flow). Except for the body in upright condition, for which only a half symmetric part has been described, the complete section has been modelled, as per figure 7. A final structured Cartesian grid of 500x350=175,000 cells, refined in the proximity of the wedge, has been used and is presented in figure 6 and 7, for the case of an heeling angle of 9.8 degrees. Minor modifications have been applied to the mesh for the case with an inclination of 28.3 degrees.

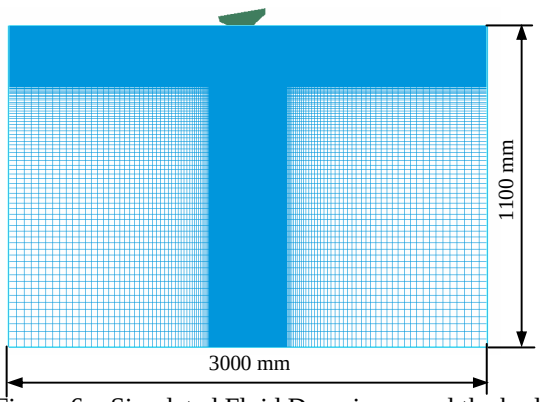


Figure 6 – Simulated Fluid Domain around the body

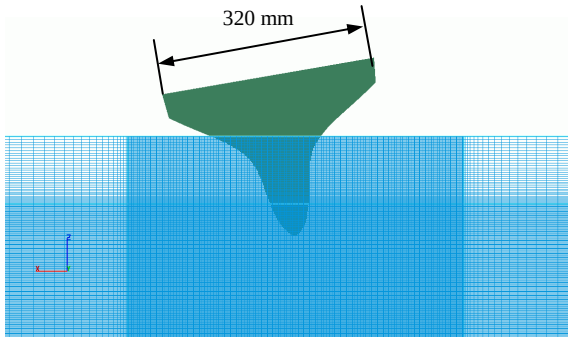


Figure 7: Detail of mesh used for RANSE simulations

The following conditions at the boundaries of the parallelepipedal mesh have been assigned:

- the body section has been set as a moving object with a constant vertical velocity;
- the wedge bottom has been defined as a non-slipping wall condition;
- the bottom has been assigned as a symmetry boundary condition
- a continuative (zero gradient with suppression of reflection) condition has been assigned to both sides of the fluid domain

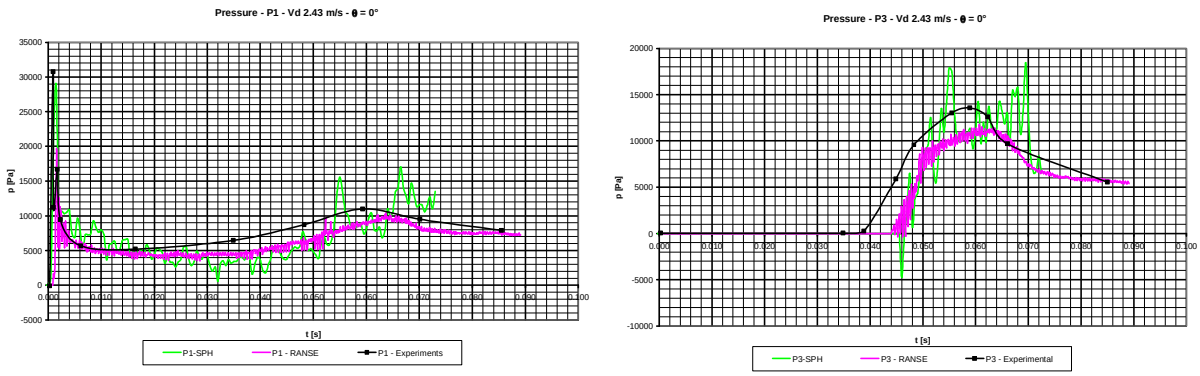
All the calculations were performed in a non-stationary mode, up to a time of about 0.9s, corresponding to the section chine deeply submerged under the undisturbed free surface. The method has been used with a variable time step, automatically refined by the code to obtain a good convergence of the residuals and satisfy a set of stability criteria. All simulations used an incompressible viscous flow with a standard k-ε turbulence model. The solver uses a new optimized implicit pressure solver (Yao, 2006) and a first order approximation of advection terms in the momentum equations. Free surface tracking method uses a VOF with internal void packing method.

## 4 RESULTS FROM CALCULATIONS

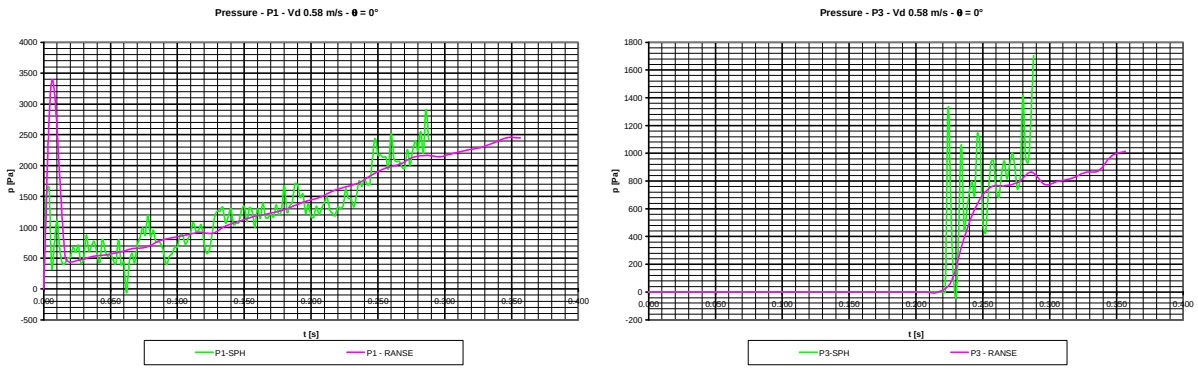
In following paragraphs 4.1 and 4.2 results from calculations with both methods are reported, together with experimental results. In particular, a first series of tests was made at constant drop speed, in order to simplify calculation and to compare results from SPH and RANSE; after these preliminary tests additional runs have been performed with "forced" drop speed, i.e. drop speed time history as recorded during experimental tests has been imposed. The actual dynamics of the free fall, following the Newton equation with gravity and hydrodynamic forces acting on the section, will be included in future applications. In the whole, CPU times for SPH are longer than RANSE for the presented cases by a 1.5 ratio.

### 4.1 CONSTANT DROP SPEED

Preliminary results with constant drop speed are reported in following figures 8-9 and 10-11 regarding calculations for the section in the upright position and drop speed equal to 2.43 and 0.58 m/s respectively (P1 and P3). It has to be noted that in the case of the lower drop speed experimental results are not reported having a large variations in speed during free fall.



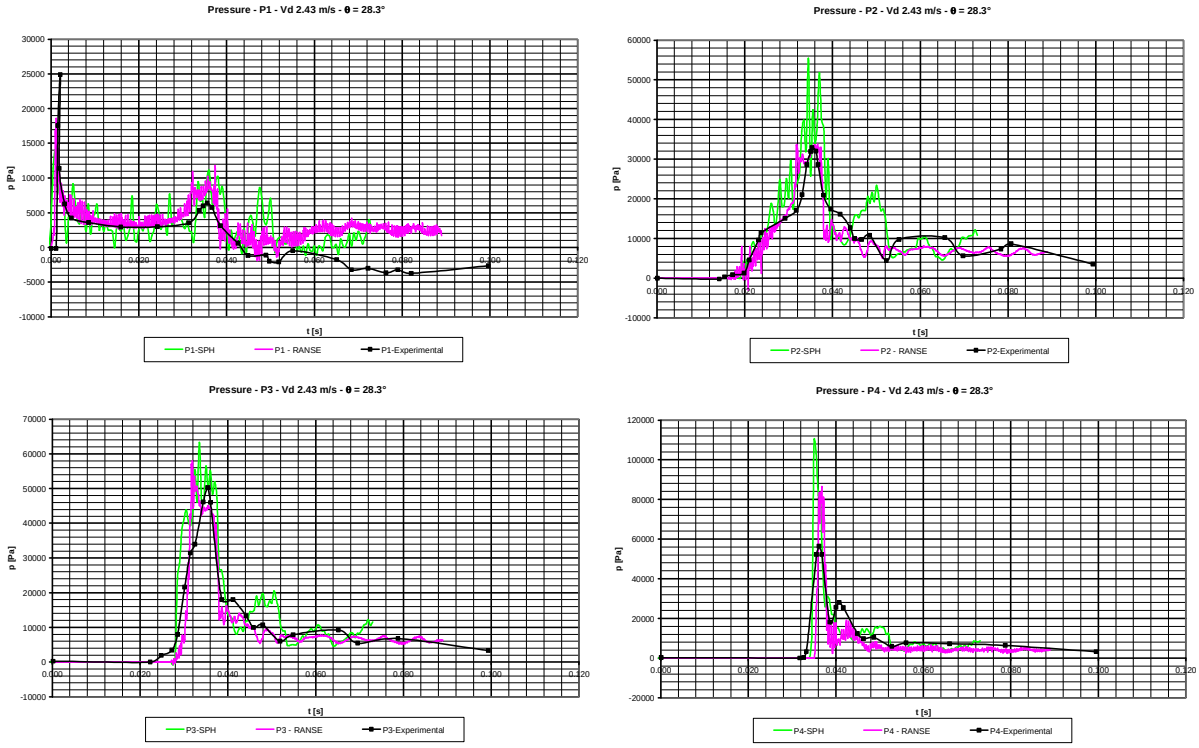

Figures 8-9 – Results of calculations with SPH and RANSE – Vd = 2.43 m/s (constant) – Heel = 0°




Figures 10-11 – Results of calculations with SPH and RANSE – Vd = 0.58 m/s (constant) – Heel = 0°

Results from SPH and RANSE method are comparable, providing same qualitative behaviour and pressures order of magnitude; in particular, SPH method seems capable of capturing the main effects, even if pressure time histories still present significant oscillations,

especially after a large number of time steps has been computed; it is believed that this effect can be partly due to reflected pressures inside the “pool”, which currently are not damped in any way. For what regards comparisons with experimental data in case of higher drop speed, there is in general a rather good correlation, with similar peak pressures and qualitative behaviour; SPH method captures better P1 peak pressure, while RANSE behaves better for other points, for which SPH starts oscillating, even if qualitative behaviour is still captured; existing differences (delay in impact instant) are mainly due to drop speed time history. Since for this speed differences are not too pronounced, it has been decided to analyse also the two more complex tests with 9.8° and 28.3° heel angles. Results for 9.8° are not reported for space reasons, but are qualitatively similar to those obtained for the upright condition, while in next figures 12-15 results for P1-P4 are reported in case of 28.3° heel angle.




Figures 12-15 – Results of calculations with SPH and RANSE – Vd = 2.43 m/s (constant) – Heel = 28.3°

Also results for this highly asymmetrical test present a good correlation, demonstrating the capability of both methods to treat more complex configurations; both methods underpredict the first peak value for P1, but provide very good results for remaining time history and for whole time history of P2 and P3, while P4 peak pressure is overestimated in both cases. It is believed that in this case better results could be obtained by letting section fall free with drop speed governed by equation of motion in the vertical DOF, thus considering better the violent deceleration in correspondence to impact of the inclined section into water (figure 16).

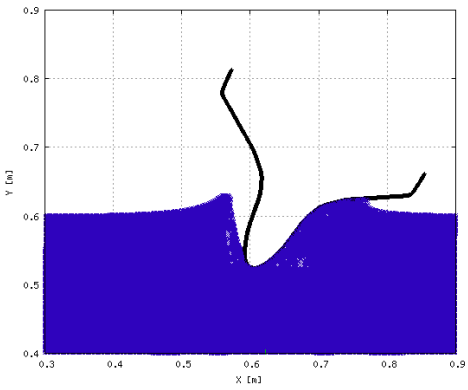
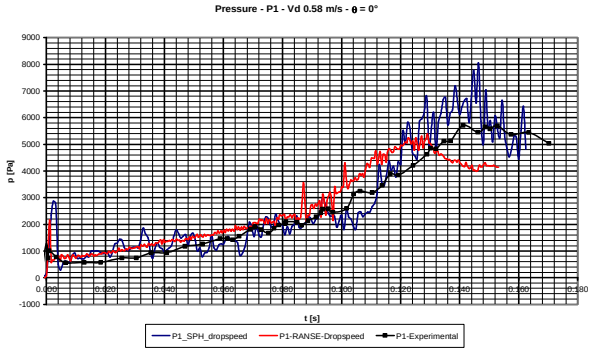
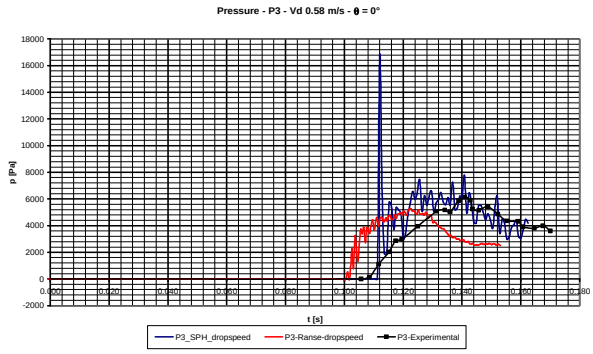



Figure 16 – Impact at 28.3° heel angle

## 4.2 FORCED EXPERIMENTAL DROP SPEED

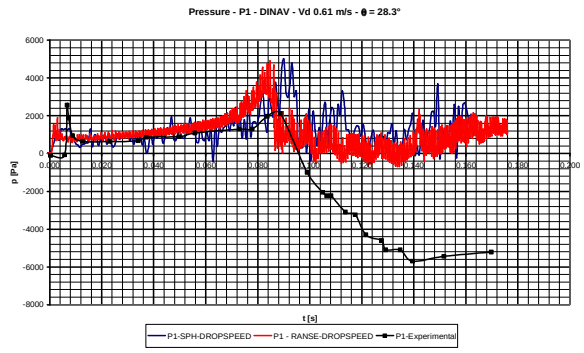
In following figures 17-18 and 19-22 results for calculations with forced drop speed (lower initial speed) and heel angles of 0° and 28.3° are reported, while in figures 23-24 results for calculations with 0° heel angle and higher initial drop speed are reported.

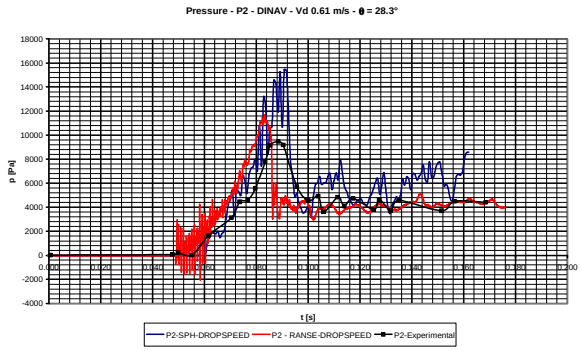


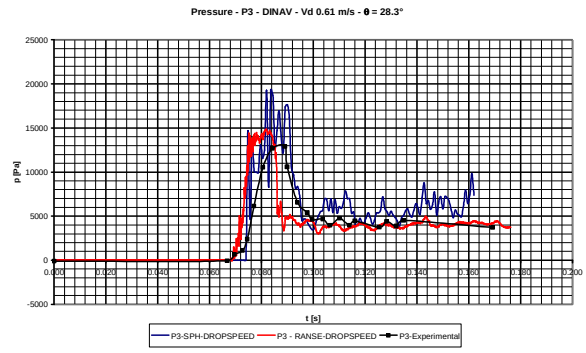



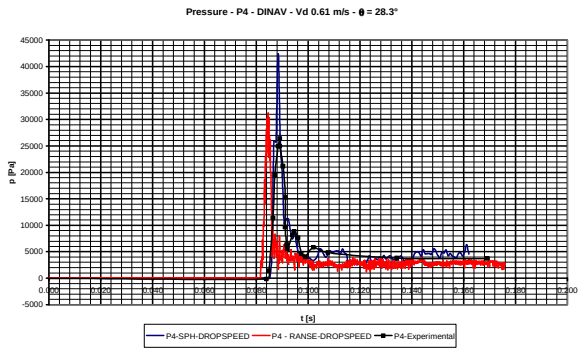

Figures 17-18 – Results of calculations with SPH and RANSE – Vd = 0.58 m/s (initial) – Heel = 0°

It can be seen that a good correspondence between experimental results and numerical calculations can be achieved; for what regards 0° heel angle, despite presenting oscillations slightly higher than those described previously, SPH seems to estimate correctly peaks and timing for pressures (not considering spurious point in P3 time history with lower velocity), RANSE has a good agreement even if it fails in estimating peak pressure of P1 for higher drop speed and timing in lower drop speed. In correspondence to 28.3° heel angle P1 time history presents significant differences (currently not explained), while remaining points are captured with sufficient correlation, even if SPH tends to overestimate pressure peak for P4.

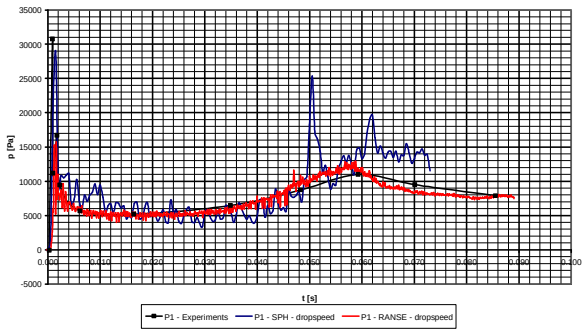


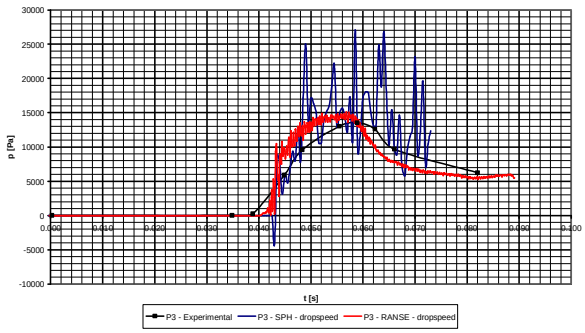




Figures 19-22 – Results of calculations with SPH and RANSE – Vd = 0.61 m/s (initial) – Heel = 28.3°

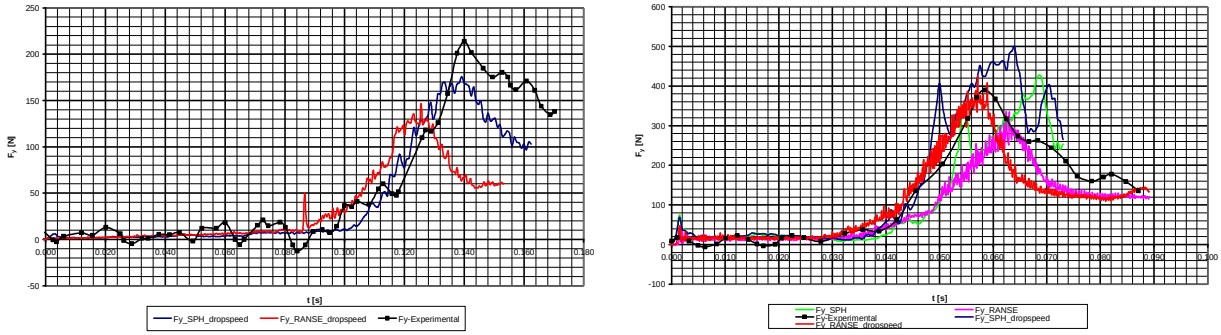


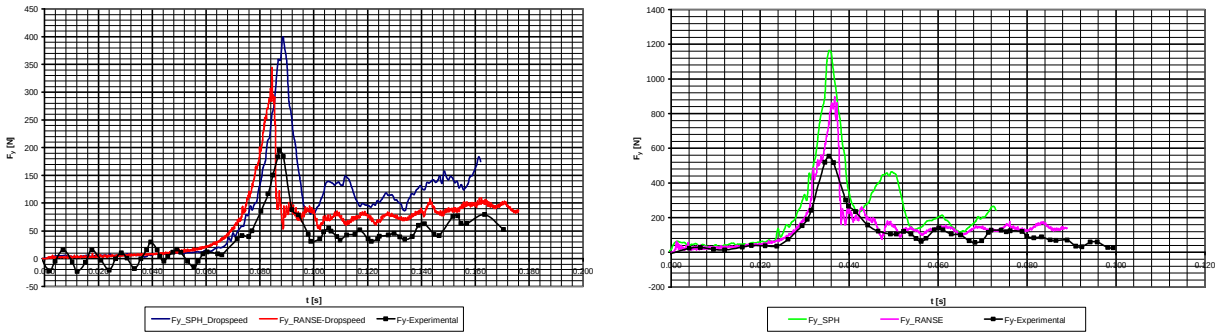

Figures 23-24 – Results of calculations with SPH and RANSE – Vd = 2.43 m/s (initial) – Heel = 0°

## 4.3 LATERAL AND VERTICAL FORCES

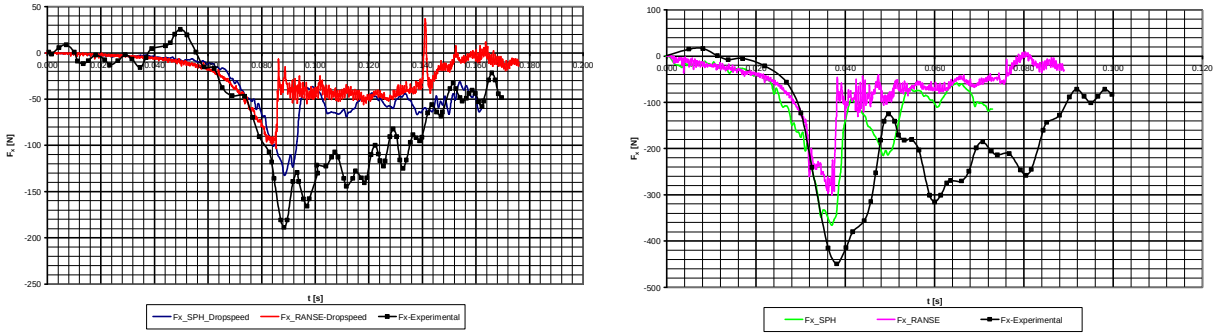
In order to have a better insight on the capabilities of the two methods, vertical and lateral forces time histories have been computed and reported in following figures 25-30.




Figures 25-26 – Results of vertical forces calculations – Heel = 0° - Vd = 0.58 (initial) -2.43 (initial / constant)




Figures 27-28 – Results of vertical forces calculations – Heel = 28.3° - Vd = 0.61 (initial) and -2.43 (constant)




Figures 29-30 – Results of lateral forces calculations – Heel = 28.3° - Vd = 0.61 (initial) and -2.43 (constant)

Forces calculation tends to smooth oscillations in case of SPH; in general, it can be seen that there is a qualitative agreement between results, which however are not completely satisfactory. In particular, regarding the upright condition, SPH presents a 20% error in peak force estimation while RANSE method captures the peak force value in correspondence to higher drop speed but underestimates in correspondence to the lower one; regarding heel angle tests, both methods tend to underestimate lateral force and overestimate vertical forces; it is believed that in this case it is not sufficient to simply impose drop speed time history (as it has been done for 0.61 initial drop speed), but it is necessary to let velocity evolve under the effect of forces acting on the sections, which would probably allow to reduce vertical force peaks in correspondence to slams of the horizontal parts.

## 5 ALTERNATIVE BOUNDARY TREATMENT WITH SPH

In the present paragraph, a novel boundary treatment which is being tested currently at DINAV is presented; in particular, the method is an “improved ghost particles” approach, which is being studied in order to better solve the problem of the boundary truncation effect.

The boundary truncation effect is due to the region of the particle support domain which falls outside the fluid domain in correspondence to boundaries. In two dimensions, as in the present work, this region corresponds to the area marked in red on the left side of figure 31. Main idea of this approach is to use ghost particles in order to “fill in” the lost area, leading to the concept of variable smoothing length ghost particles (VSLGP). When a particle moves near a boundary a VSLGP is created, having a smoothing length related to the area of the support domain of the real particle that extend outside the boundary and placed in its centre; density of the ghost particle is equal to the one of the real particle. Ghost particle velocity can be treated as in usual ghost particle approach, but only a slip condition was tested presently.

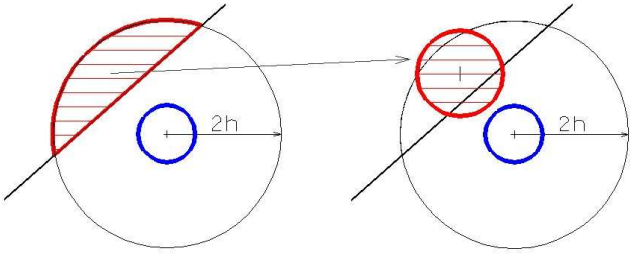
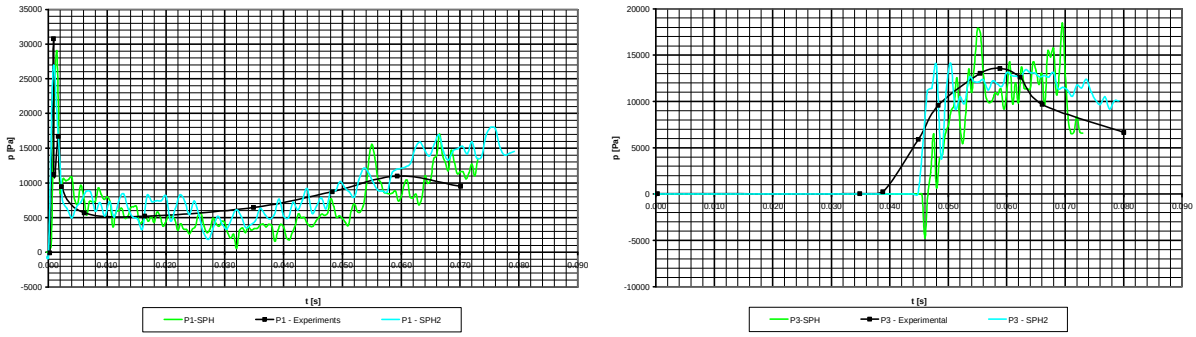


Figure 31: Alternative boundary treatment scheme

Preliminary calculations showed that this treatment is able to avoid particles penetration of solid boundary and to provide good results also with a lower number of particles than those utilised with the repulsive forces approach. As an example, in following figures 32-33 results for 2.43 m/s constant drop speed are reported and compared with previous results. As it can be seen, results are qualitatively comparable, even if number of particles has been reduced by a factor of 2.5, with the subsequent speed-up of calculation time with respect to usual ghost particles approach (see Viviani et al. 2006). Moreover, application of this new approach seems capable of reducing part of the oscillations.



Figures 32-33 – Results of calculations with different boundary treatment - Vd = 2.43 m/s (constant) – Heel = 0°

## 6 CONCLUSIONS

A series of calculations have been made in order to simulate slamming phenomenon on a typical 2D ship bow section, adopting lagrangian SPH method and eulerian VOF RANSE.

In general, both methods seem to be capable of capturing the physics of this phenomenon, even if SPH calculations still present significant oscillations, while RANSE calculations seem to have a higher stability. Nevertheless, SPH method has proven to be suitable also for this kind of calculation with complex shapes, variable drop speed and heel angles, and not only for simpler (and more usual) prismatic wegde calculations already presented in (Viviani et al. 2006).

Compared to RANSE, SPH methods present significant advantages, like the inherent capacity to capture free surfaces, sprays and complex kinematics, and the simpler implementation of algorithms, which make them more suitable for “ad hoc” adaptations.

It is believed that with some further development efforts the main existing problem of pressure oscillations can be reduced, by means of a better damping of pressure waves inside the pool and by application of smoothing functions in time to reduce fast oscillations; moreover, it is believed that the introduction of real dynamics for the free fall rather than an imposed speed can result in a certain damping of oscillations, especially in presence of violent

impacts. The implementation of effective forces will allow also to study different cases, such as floating bodies, for which however a parallel version of the code will be needed in order to avoid too long computational times.

Finally, an alternative boundary treatment currently under development has been presented; results computed adopting this treatment seems promising and it is believed that it could allow to reduce significantly number of particles with similar results, thus contributing to the reduction of computational burden; moreover, this method can be used also in case viscosity has to be considered, since implementation of no-slip condition is straightforward.

**AKNOWLEDGMENTS**

This work has been developed with the support of the European Union MARSTRUCT Network of Excellence.

**REFERENCES**

Aarsnes, J.V., 1996, "Drop test with ship sections – effect of roll angle" Marintek Report 603834.00.01"

Chuang, S.L., 1967, "Experiments on Slamming of Wedge-Shaped Bodies", Journal of Ship Research, 190-198

Gingold, R.A. and Monaghan, J.J., 1977, "Smoothed particle hydrodynamics: theory and application to non-spherical stars", Royal Astronautical Society, 181, 375-389

Hirt, C. W. and Sicilian, J. M.: 1985, 'A Porosity Technique for the Definition of Obstacles in Rectangular Cell Meshes', Proc. Fourth Int. Conf. Ship Hydrodyn., National Academy of Science, Washington, DC.

Libersky, L.D., Petschek, A.G., Carney, T.C., Hipp, J.R. and Allahdadi, F.A., 1993, "High strain Lagrangian hydrodynamics: a 3D SPH code for dynamic material response", J. Computational Physics, 109, 67-75.

Liu, G.R., Liu M.B., 2003, "Smoothed Particle Hydrodynamics – A Meshfree Particle Method", World Scientific Publishing Co. Pte. Ltd.

Lucy, L.B., 1977, "A numerical approach to the testing of the fission hypothesis [close binary star formation]", Astronomical Journal, 82, 1013-1024.

Monaghan, J.J., 1992 "Smoothed Particle Hydrodynamics" Annual Rev. Astron. Astrophysics 30, 543-574

Monaghan, J.J, Thompson, M.C. and Hourigan, K., 1994, "Simulation of Free-Surface Flows with SPH", ASME Advances in Comp. Methods in Fluid Dynamics, 196, 375-380.

Monaghan, J.J. , 1994, "Simulating free surface flows with SPH, Journal of computational Physics, 110, 399-406

Oger, G., Doring, M., Alessandrini, B., Ferrant, P., 2006, "Two-dimensional SPH simulations of wedge water entries", Journal of Computational Physics 213 (2006) 803-822

Stavoy, A.B. and Chuang, S.L., 1976, "Analytical Determination of Slamming Pressures for High Speed Vehicles in Waves", Journal of Ship Research, 20-4, 190-198

Viviani, M., Savio, L., Brizzolara, S., 2006 "Evaluation of slamming loads on V-shape ship sections with different numerical methods", 9th Num. Towing Tank Symp. (NUTTS 2006)

Wagner, V.H., 1932, "Über Stosz und Gleitvorgange an der Oberflache von Flüssigkeiten", Zeitschrift für Angewandte Mathematik und Mechanik, 193-215

G. Wei, "An Implicit method to Solve Problems if Rigid Body Motion Coupled with Fluid Flow", FSI-05-TN76

Zhao, R. and Faltinsen, O.M., 1993, "Water entry of two-dimensional bodies", Journal of Fluid Mechanics 246, 593-612

Zhao, R. Faltinsen, O.M., Aarsnes, J.V., "Water entry of Arbitrary Two-Dimensional Sections with and without Flow Separation", Proc. 21st Symposium on Naval Hydrodynamics