V European Conference on Computational Fluid Dynamics
ECCOMAS CFD 2010
J. C. F. Pereira and A. Sequeira (Eds)
Lisbon, Portugal, 14–17 June 2010

VALIDATION OF CFD CODES FOR SLAMMING

R. Marcer[1], C. Berhault[1], C. de Jouëtte[1], N. Moirod[2], L. Shen[3]

[1] Principia, Athelia 1, 13705 La Ciotat Cedex, France
[2] Bureau Veritas, France
[3] Strathclyde University – SSRC, UK

Key words: slamming, CFD, VOF, SPH

Abstract. *In the context of the European VIRTUE program, Principia, Bureau Veritas and SU-SSRC worked on the improvement of CFD codes for slamming modelling.*
For slamming, the objective is to use CFD for the computation of slamming loads induced by waves, which can be then introduced in a Finite Element model to predict the hull girder response (whipping).

Two free surface methods have been developed to simulate slamming problems: the Volume of Fluid (VOF) free surface method, implemented in URANS 2-phases codes, allowing large free surface deformation up to jet break-up, and the Smooth Particle Hydrodynamic method (SPH) mainly dedicated to high dynamic transient response of the free surface.

Benchmarks between the different methods and validations of the CFD codes are performed on the academic wedge impact problem.
Then, an example of slamming simulation for a real ship configuration is shown.

The results show that VOF and SPH methods give comparable and validated results. They can be used to predict accurate slamming loads on real ships and the induced whipping ship response after a coupling with a sea-keeping panel code.

Finally, the paper presents the methodology of coupling CFD with seakeeping panel codes (based on BEM theory) for computing slamming loads and hydro-elastic response of the ship.

1 INTRODUCTION

The problem of slamming is one of the most critical issues that have to be addressed by the research community in Naval shipbuilding.

Slamming is one of the leading hydro-elastic problems which keeps defying the designers understanding of ship behaviour at sea. It is just as difficult to measure this phenomenon in model tests as it is to simulate it properly.

The notion of slamming covers different physical aspects of water impact on the ship hull and equipments:

- The impact of the bow (bulb) and stern on water induced by the ship motions in waves. The impact is mainly vertical and is governed by the local vertical velocity.
- The horizontal impact of steep waves or breaking waves on the ship hull governed by relative ship/waves velocity
- The water impact on the ship equipments induced by water run-up and green water on deck

The majority of research and development in this field has concentrated on the classic water entry problem rather than wave impact or green water effects. Nevertheless, there is a substantial literature on both subjects, and Classification Society Rules that define empirical approaches that can be taken. There remains, however, a significant amount of uncertainty in the choice of impact coefficients, and CFD offers a potential route to understanding extreme design wave loads of this kind.

For the slamming and water entry problems, Kihara [1] used a mixed Euler and Lagrangian method to calculate the water entry of a wedge with constant vertical velocity. The hydrodynamic pressure on wedges agrees reasonably well with theoretical one.

Greenhow [2] simulated water entry into initially calm water by matched asymptotic expansions. The pressure distribution is encouraging. Zhao and Faltinsen [3] introduced the so-called “cut-off” model of jet flow to study the water entry of 2-D bodies. This model provides a good prediction of hydrodynamic pressure in the problems with various dead rise angles.

An application of VOF finite-volume RANS solver to the water impact problem has been reported by Azecuta et al [4]. The main advantage of this method is that the VOF technique can deal with complex free surface shapes which occur in these simulations. In 3D, Sames et al. [5] have demonstrated application of VOF techniques for full ship hull forms in regular waves, producing credible overall load distributions.

Applications of SPH method are more recent but the approach seems well adapted for dynamic phenomena. Some results are quite encouraging [6]. However the method is not as mature as VOF and numerical studies are still needed to reduce time consuming (3D) and to take into account viscosity and interaction in waves.

A lot of experimental works have been done on simple geometries and turns out efficient for CFD validation but few results exist for more realistic configurations.

The classical wedge impact on still water is a standard international benchmark [7]. Both experimental results and semi-analytical formulations are available for validation. 2D and 3D configurations have been studied in different conditions:

- Imposing impact velocity and inclination
- Free launching, which needs a specific set up to control attitude of the structure in air

Other simple geometries have been tested as the horizontal cylinder of circular section, the conical geometry impacting still water [8]. Some results exist for hydro-elastic structures as well [9].
More recently a French Joined Industrial Project (JIP) has been focused on the whipping prediction, including slamming loads prediction (Cyclope project, led by Principia). Tests have been done by ECN for a cruiser ship in different conditions: imposed impact of the ship bow on still water with variations of velocity and attitude, seakeeping response of a segmented model in regular seas with forward speed [10].
Tests have been also carried out by GIS-Hydro (in BGO First facilities) on a flexible segmented model of a barge of rectangular section. Decay tests, regular and irregular waves tests results are available [11].

So, the aim of this paper is to show CFD codes validation cases, based on different free surface methods, for slamming problem. And CFD results will finally used as inputs for Finite Element structural codes in order to calculate the structural elastic response.

Works focus on two different techniques:

- The Volume Of Fluid (VOF) method which permits to simulate large free surface deformation up to jets break-up. Cushioning effects could be predicted if fluid compressibility is included with this model.
- The Particle based method (including SPH) mainly dedicated to high dynamic transient response. This kind of method can be applied for further fluid and structure deformations which lead to hydro-elastic response.

The involved three CFD codes (and partners) are:

- In-house code EOLE for Principia (VOF)
- Commercial code Flow3D for Bureau Véritas (VOF)
- In-house code for Strathclyde (SPH)

The models are compared on the well-known wedge entry test case. An example of slamming simulation of a ship is given.

Finally, the paper presents the methodology of coupling CFD results with seakeeping panel codes (BEM) to compute slamming loads on one hand and hydro-elastic response of the ship on the other hand.

2 CFD MODELLING

2.1 Brief description of the involved CFD codes

The EOLE CFD code, developed by Principia since 1990, allows simulations of 3D multiphases flows, for incompressible or compressible fluid. The Reynolds averaged Navier equations are solved on curvilinear multi-block structured meshes which can move and deform. The numerical scheme is based on a finite volume scheme and a pseudo-compressibility method which uses two levels of iterations (dual time stepping) for unsteady simulations :

- a pseudo-time level based on an explicit Runge-Kutta algorithm
- and an implicit second order scheme in physical time.

Free surface motion is simulated by a fully implicit VOF (Volume Of Fluid) model which deals on complex free surface shape, without any CFL stability constraint.

Flow3d is a CFD commercial software that can simulate complex fluid flows and used in many industrials domains (Aerospace, Maritime, Microfluids …). The software is based on resolution of Reynolds Average Navier-Stokes equations including one or two phases flow models, compressible or incompressible models of inviscid flow or viscous laminar or turbulent flows.
Flow3d uses a structured finite difference grid in Cartesian or cylindrical coordinates and solve fluid interfaces with TruVOF method (improvements of the original Volume of Fluid – VOF- method to increase the accuracy of boundary conditions and interface tracking).

2.2 VOF method

The principles of the VOF technique are summed up below.

This is a technique commonly used in fluids dynamics software for its robustness for tracking free fluid surface in large deformations.

A function *F* is used to identify the volume fraction of the denser fluid in each cell. Then,

$F = 1$ means that the cell is filled in with liquid
$F = 0$ means that the cell is filled in with air
$0 < F < 1$ means cells containing an interface

The time evolution of *F* is governed by a conservation equation:

$$\frac{\partial F}{\partial t} + div\left[\rho\left(\vec{U} - \vec{W}\right)\right]F = 0$$

where $\vec{U}$ is the interface velocity
and $\vec{W}$ is the grid velocity

In the specific VOF model of EOLE, the free surface position can be computed on moving and deforming grids and in a non-galilean reference frame. The VOF algorithm is used to compute the convective fluxes while preserving the discontinuity at the interface, has been extended and adapted on arbitrary curvilinear grids. The time integration is fully implicit so without any CFL constraint.

In a 2-fluids model, the Navier-Stokes equations are solved in the 2 fluids (water and air), and for the cells containing the free surface as well, using a mean density defined as $\rho_{ijk} = \rho_w * VOF_{ijk} + (1 - VOF_{ijk})\rho_a$, where the subscripts (i,j,k) indicate the considered cell.
In this model, no boundary condition is needed on free surface.
Nevertheless, the atmospheric pressure is imposed on the upper boundary of the air domain.

For slamming, a 2-fluids model allows theoretically taking into account the resistance of the air flow, due to its rising pressure which occurs at the liquid impact on the wall. This resistance due to the air confinement will probably tend to limit the liquid pressure level at the impact.

2.3 SPH

A lot of R&D works are currently done on the SPH methods to model highly dynamic fluid flows and their coupling with structural vibrations. SPH is a pure Lagrangian method as the main principle to follow trajectories of particles in time. Each particle represents physical properties of the fluid.
The governing equations for fluid flow are the mass and momentum conservation for fluid particle.
The lagrangian nature of SPH (Smoothed Particles Hydrodynamics) means that changes in density and flow morphology are automatically accounted for without the need for mesh refinement or other complicated procedures.
In general, SPH calculation is very computationally demanding, both in memory and in CPU time. It usually involves a large number of particles to be geometrically enough to model the deformation of fluid body. In 3D cases, the SPH model may involve several millions of particles.

In this study, the flow is simulated with the SPH code by Strathclyde University. The main features of the calculations are:

- One-phase model: only the liquid phase is computed with Navier-Stokes system.
- Variable smooth length algorithm is used for slamming case.
- Mirror particles solid boundary condition is used to simulate entry wedge.

3 SIMULATION OF THE WEDGE ENTRY

3.1 Description of the test case

A sketch and the dimensions of the wedge are given hereafter:

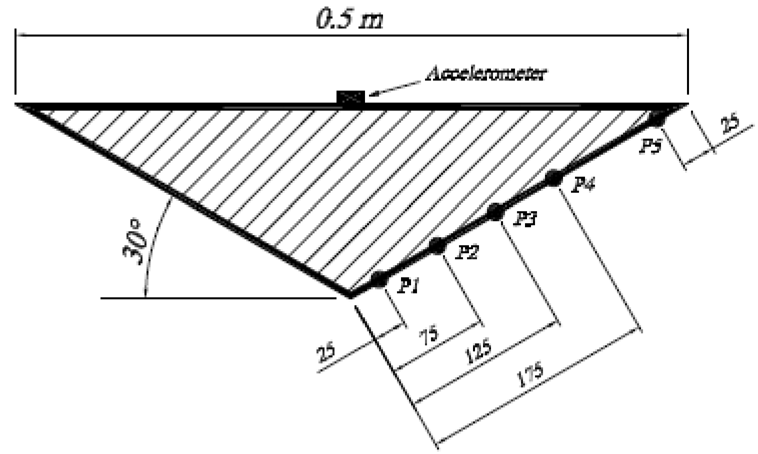

Figure 1: Sketch of the wedge geometry

- Width of the wedge: 0.5 m
- Dead rise angle: 30°
- Drop inclination with vertical : 0°
- Wedge dropped with vertical velocity -6.15m/s (at the instant of contact with water)
- Length of wedge: 0.5 m
- Length of the edge: 0.29 m
- Length of the height: 0.14 m

The flows conditions are:

- Water density = 1025 kg/m^3
- V(t) : drop velocity
- P0 : atmospheric pressure
- ρ : mass density of the fluid
- t : time variable
- Z : vertical coordinate on the body surface
- Zk : vertical coordinate of the keel
- Zd : draft of the body
- $C_p = \dfrac{P - P_0}{\frac{1}{2}\rho V(t)^2}$: pressure distribution along the edge
- Z/int(V(t)) $= \dfrac{z}{\int_0^t V(t).dt}$: Elevation of the free surface along the edge.

The imposed wedge velocity is given on Figure 2.

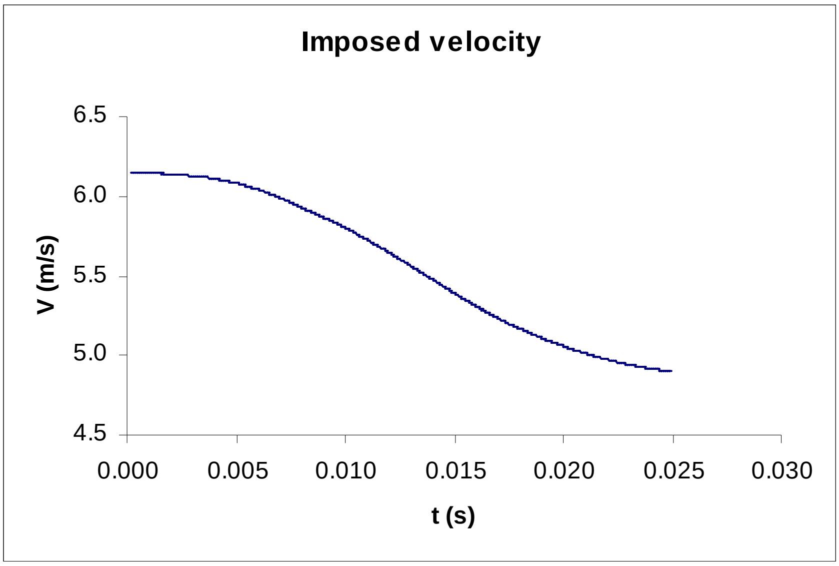

Figure 2: Drop velocity law

The instant t=0s corresponds to the initial time when the wedge impacts the free surface with 6.15 m/s velocity.

For the VOF codes, Figure 4 shows the mesh used for EOLE which is quite coarse (160.000 cells).
For Flow3D, the domain is made of 6 nested blocks. The finest one is centred on the zone of impact (Figure 4). The mesh includes 270.000 cells.

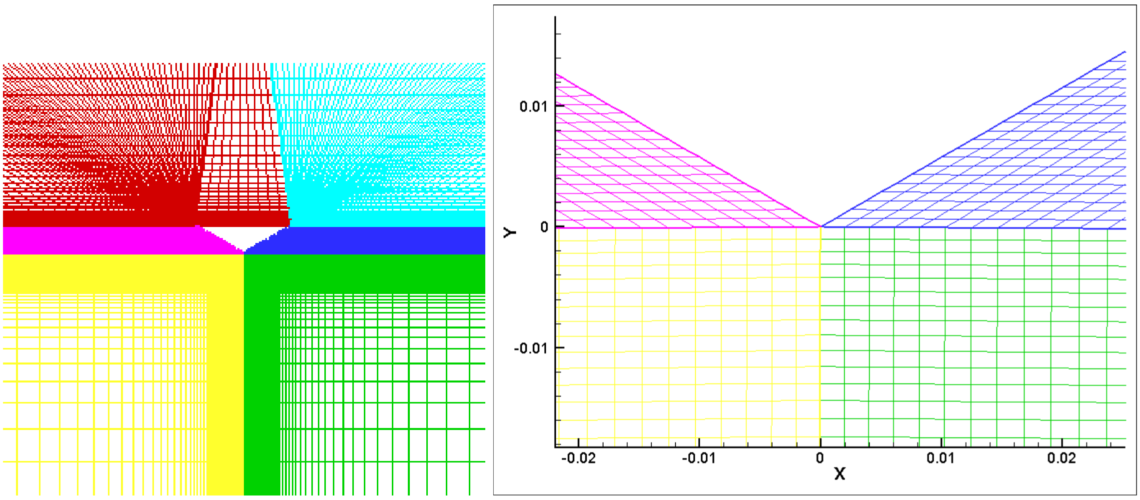

Figure 3: Multi-blocks mesh for EOLE

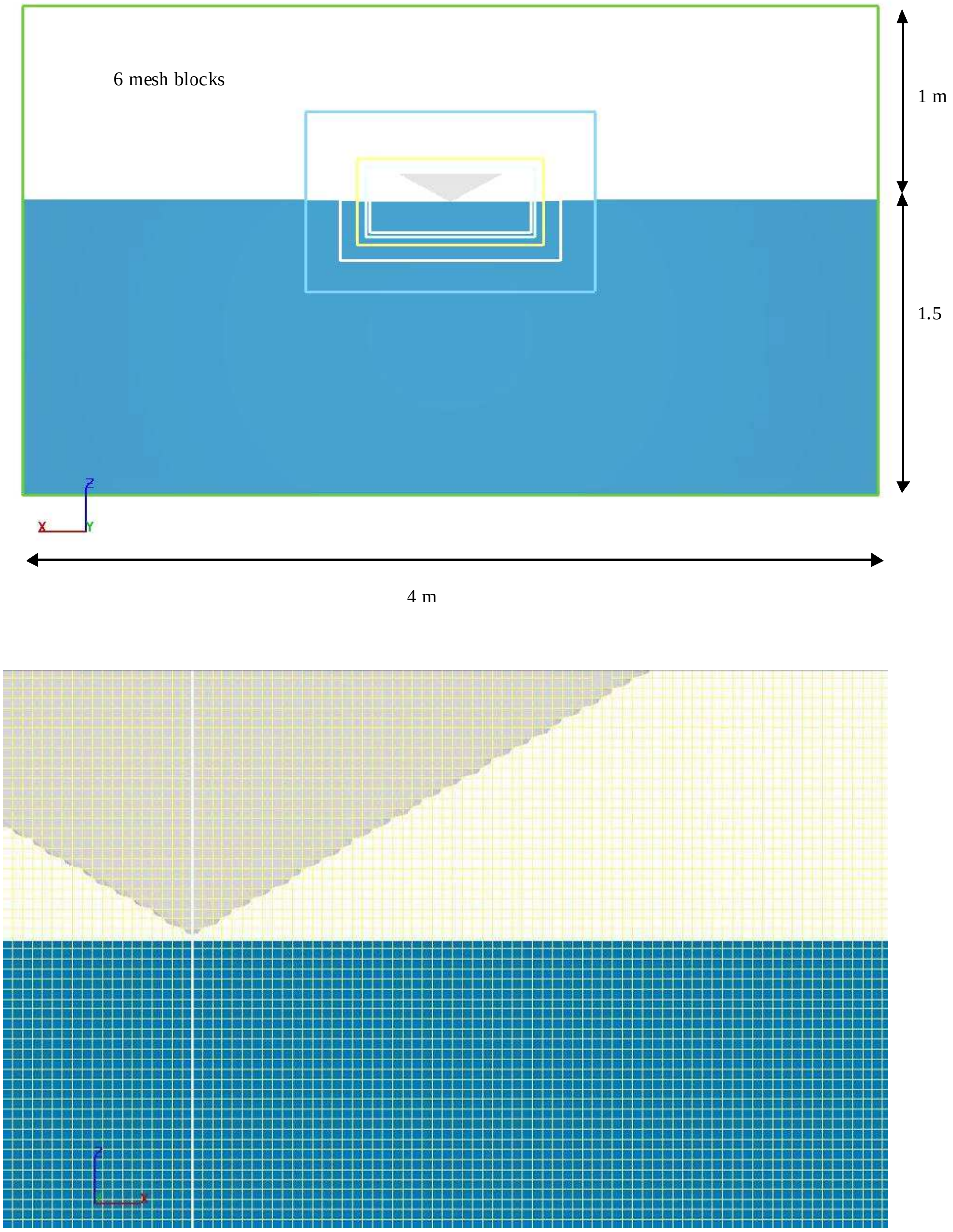

Figure 4: Mesh for Flow3D

3.2 Results

Figure 5 shows a snapshot of the flow at t=0.016s, computed with both VOF and SPH methods.
The results are quite comparable, especially regarding the jet thickness and dynamics.

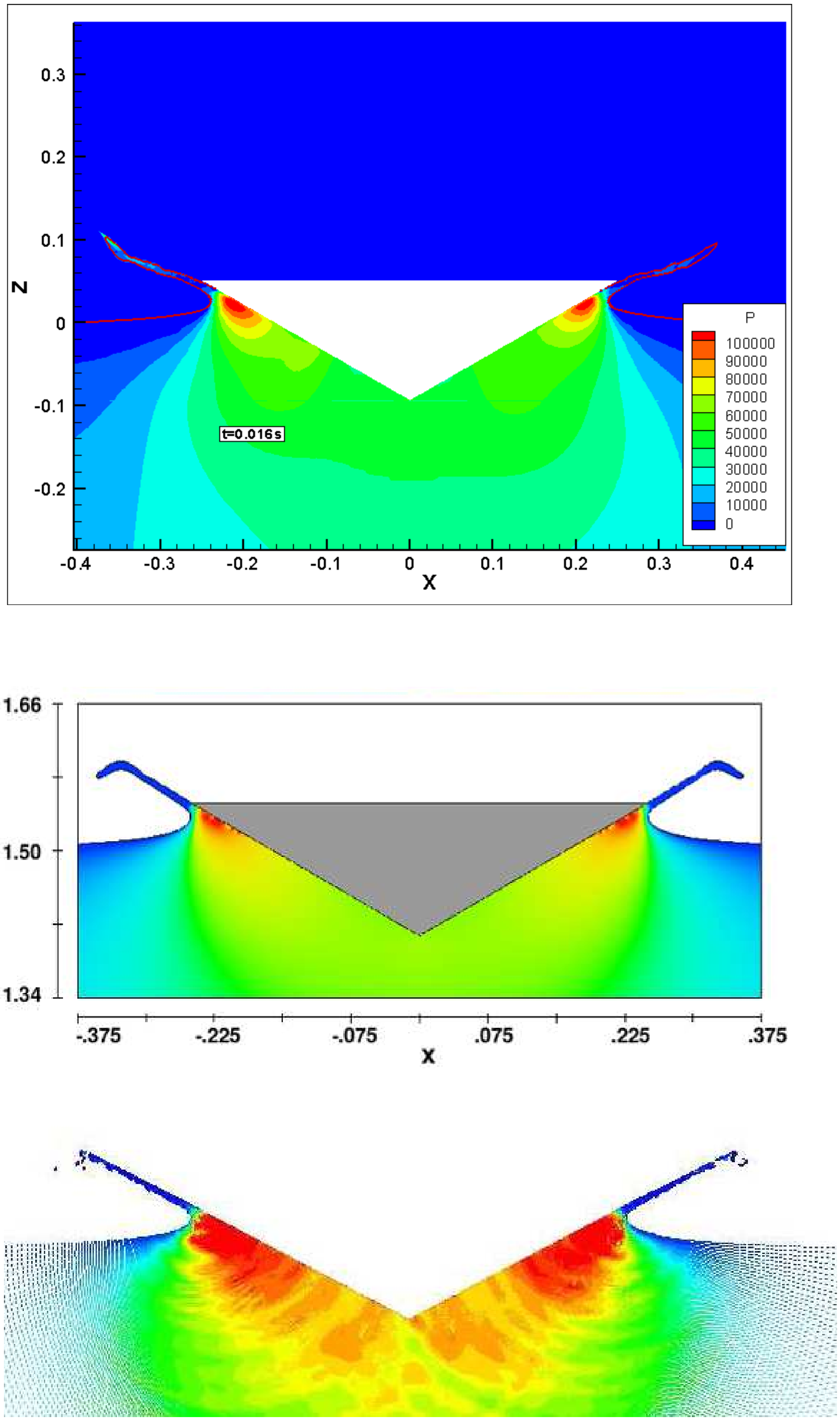

Figure 5 : Shape of jet at t=0.016s

VOF on the top (EOLE) and middle (Flow3D), and SPH at the bottom.

Figure 6 to Figure 8 show the pressure distribution along the wedge wall for three different instants. Comparisons have been done for VOF and SPH methods, and analytical and experimental reference results [7].

On the whole, the results obtained with the different codes (VOF, SPH) are comparable and quite satisfactory with respect to the reference (analytical values, experiments).

Looking further at the results, the SPH ones from SU-SSRC are close to the analytical solution except at the peak position. It may be explained by the hypothesis of perfect fluid. The VOF results (especially for EOLE) are more comparable with the experiments.

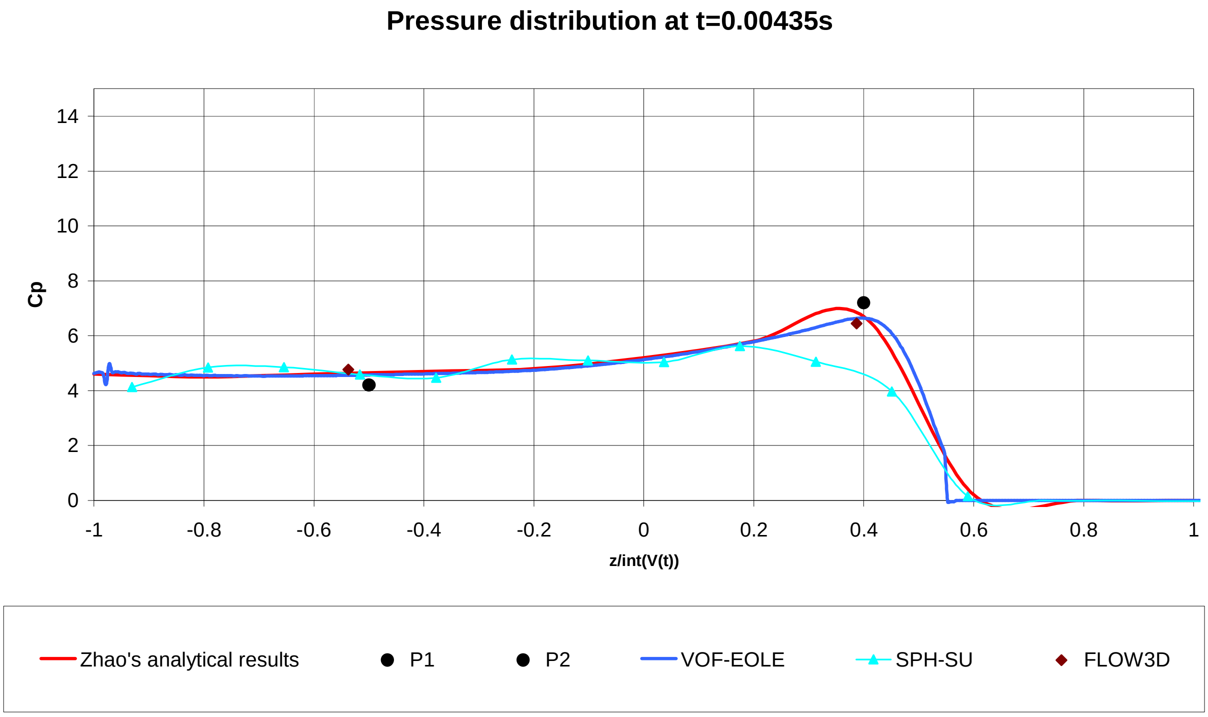

Figure 6 : Pressure distribution at t=0.00435 s - comparison between codes

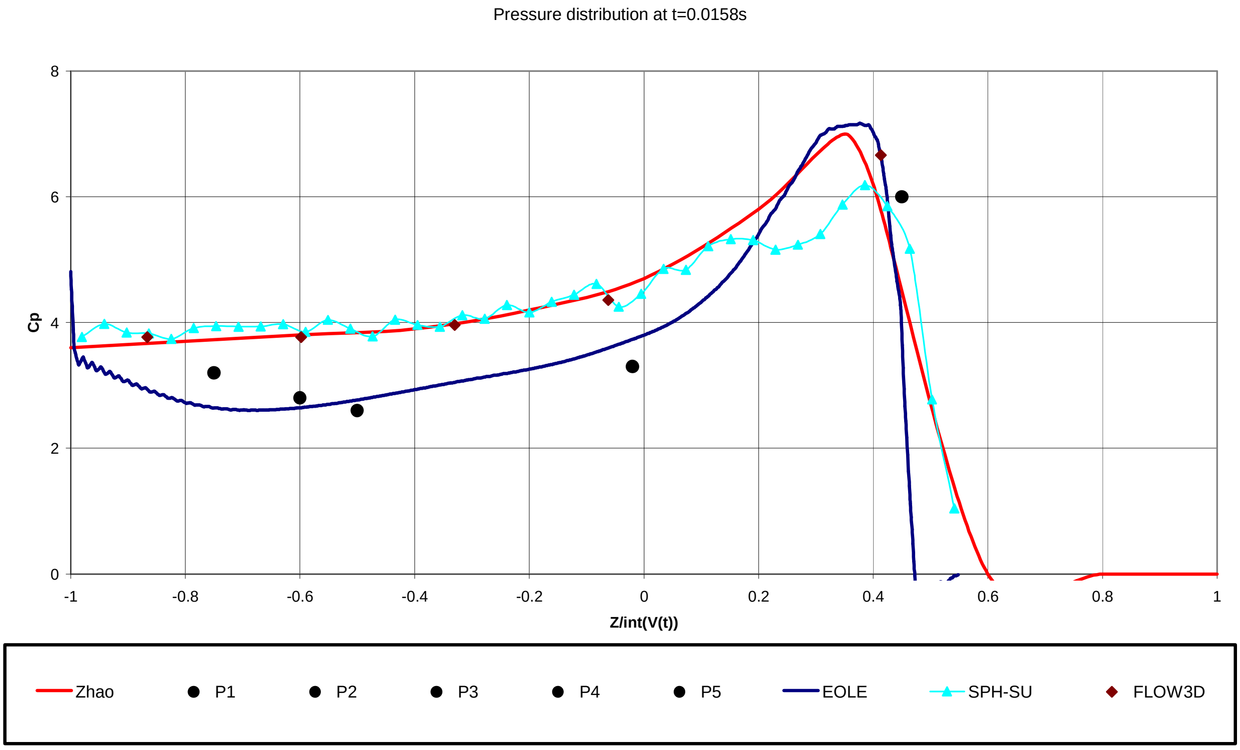

Figure 7 : Pressure distribution at t=0.0158 s - comparison between codes

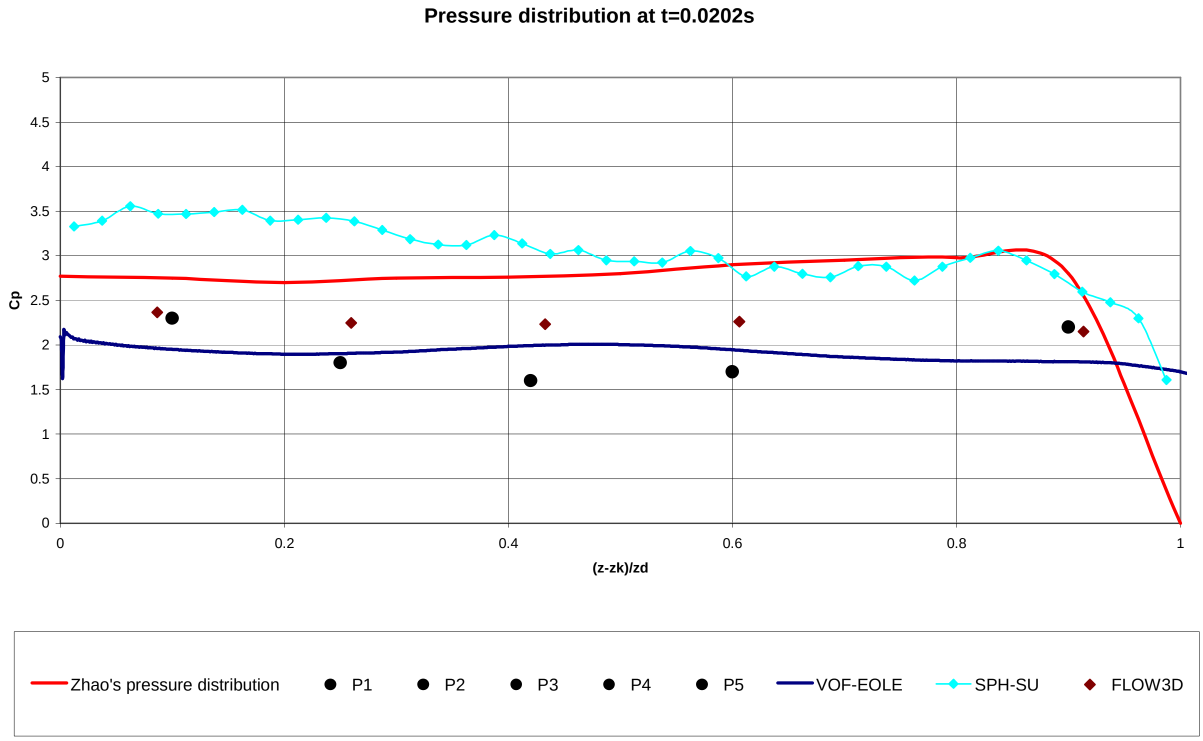

Figure 8 : Pressure distribution at t=0.0202 s - comparison between codes

Figure 9 shows comparisons of the computed slamming load with experiments [7]. The model gives a correct evolution of the load even if the corresponding maximal value is slightly higher than the experiment one. It can be explained by three dimensional effects observed in experiments, not taken into account with the 2D model, which represent approximately 20% reduction with respect to two dimensional results [7].

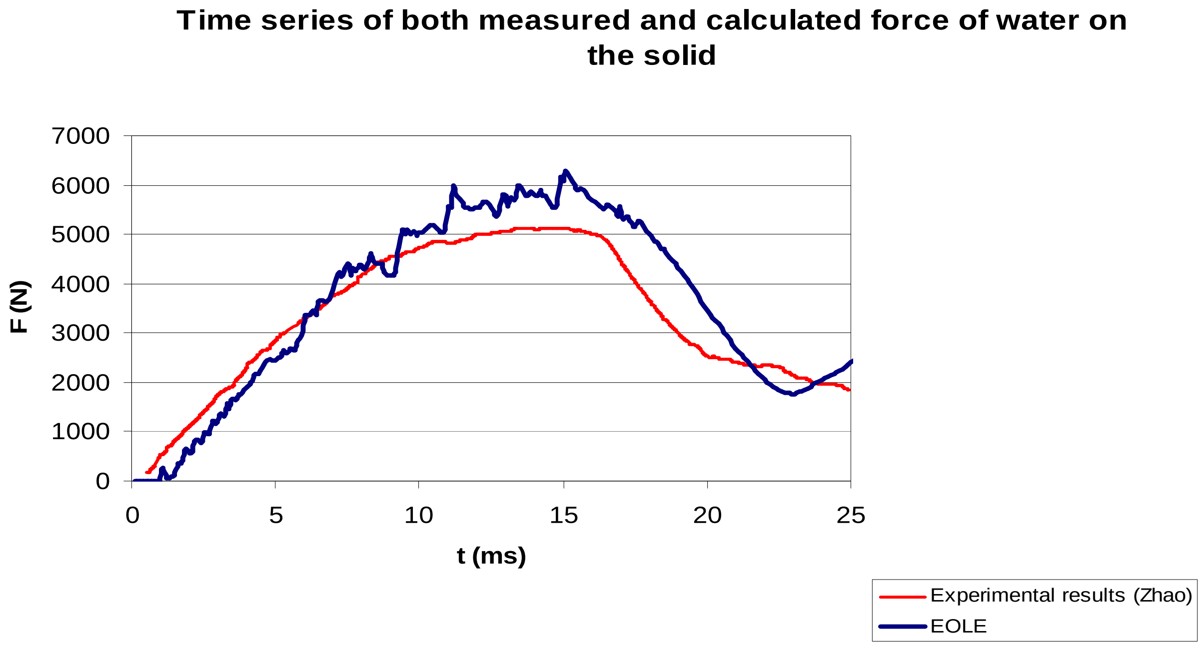

Figure 9 : Slamming load – EOLE (VOF)

4 SIMULATION OF SLAMMING OF A SHIP

An example of slamming simulation for a real ship configuration has been performed with the code EOLE using the VOF free surface method.

Qualitative results given on Figure 9 highlight the ability of that kind of model to simulate the slamming of the bow ship in the water.

Quantitative validations with experiments remain to carry out for this real ship application, for example passengers or containers ships

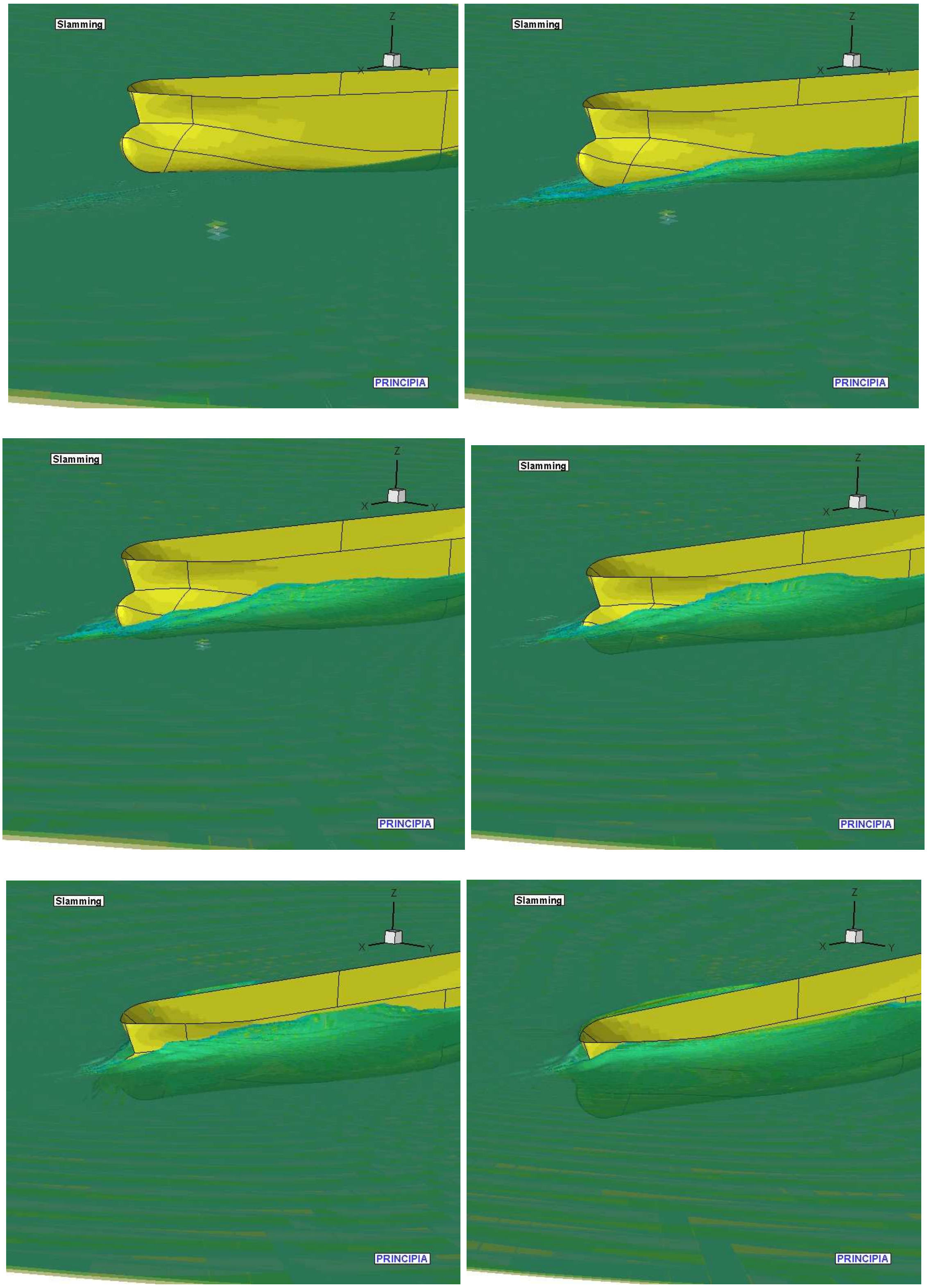

Figure 10: Example of slamming simulation of a ship – EOLE (VOF)

5 COUPLING WITH A SEAKEEPING PANEL CODE

Slamming loads influence the hull design in a number of different ways:

- Global forces inducing hull girder vibrations (whipping)
- Local loads and pressures inducing local deformation and vibrations of hull structure (hydro-elastic response)

Taking the ship bow as an example, the slamming phenomena can be described in the four stages:

- Slamming occurs when the vertical bow velocity at the bow water entry induced heave and pitch motions is greater than a specific value. The Ochi criteria is generally used:
  $V > 0.093\sqrt{gL}$
  As the ship motion is governed by inertia, water entry velocity is imposed. That means that slamming loads have a small influence on the ship motions.
- The first stage of slamming is the hull impact on water associated to a very small time scale (≈ 0.005 sec) corresponding to the occurrence of the maximum of local pressure:
  $p = 1/2\rho\, C_P V^2$
  Pressure is governed by local hull geometry, free surface deformation (jets), local hydro-elastic deformation
- The second stage of slamming is the hull water entry associated to a larger time scale (≈ 0.1 sec) corresponding to the occurrence of the maximum of the global force which induces whipping. Slamming force is mainly governed by the fluid inertia loads varying during water entry and is generally represented by a simple formulation:
  $F_S(t) = \rho C_S S_{wetted} V^2(t)$
  Where C_S is an impact coefficient derived from the force calculation.

C_P and C_S are not correlated, excepted when the maximum of pressure and global force occur at the same time. This is, for example, the case for a flat plate or shallow wedge impacting a flat water area, and where air cushioning is not involved. For the more practical ship hull impact problems, the flow field and geometries involved are complex. That is the reason why a methodology for studying such problems has been suggested within the VIRTUE Project.
A calculation process has been proposed in a similar way as for sloshing:

- Use a standard sea-keeping code to identify the occurrence and the conditions of slamming: long time series for specific cases of the sea-states scatter diagram. Selection of the most critical cases.
- Use a CFD code to simulate hull impact with the imposed water entry conditions selected in the sea-keeping screening study. Derivation of impact coefficients C_S associated to a slamming load formulation.
- Use the slamming loads time series in a FEM model to estimate stresses induced by whipping.

An alternative is to implement a hydro-elastic beam model coupled with the rigid body equations in a standard sea-keeping code. Then, the effect of imposing slamming loads can be combined with the wave frequency loads. This is the case for some sea-keeping codes used in the Virtue project. Whipping response could then directly be estimated.

The first step is to apply a methodology based of the standard approach used for design:

- Identification of the sea-sates conditions to consider,
- Estimation of the ship motions in waves in frequency domain to cover a large range of conditions. Time domain approach can be used to obtain more accurate results,
- Estimation of the slamming occurrence within the waves and the part of hull concerned with. The Occhi criteria, based on the local relative vertical velocity, is used,
- Estimation of the slamming conditions and calculations of the local slamming loads using an external routine based on CFD calculations instead of semi-analytical or empirical formulations,
- Loads are imposed on a FE model of the hull structure to provide dynamic response. Some sea-keeping models integrate directly a simple beam model for estimation of static and dynamic shear stress and bending moment, which is the case for Diodore.

In this methodology, the main assumption deals with:

- The estimation of the slamming conditions which are governed by the non linear local waves field and by the non linear ship response in case of severe seas.
- The estimation of the local slamming loads, modelling of the local free surface deformation, jets evolution and cushioning effects, vortex generation in the water entry phase.

The CFD application will be focused mainly on this key point to propose an alternative to existing approximations. Coupling with ship motions is immediate if the concerned wave time history is imposed in the CFD code. But ship hull kinematics provided by a standard sea-keeping code could be also imposed to model forced impact. Introducing these two kinds of input into CFD modelling, local loads are computed and imposed on a FE model of the structure.

Full hydro-elastic coupling does not seem necessary as high frequency vibrations of the ship do not influence the ship motions and the local kinematics. This is true only for standard ships (majority) but could be discussed for a “flexible” hull.

The methodology is to solve both global sea-keeping response and impact loads in the CFD codes, even if elastic hull guider is not modeled in the CFD code. The following steps are:

- Identification of the more critical sea states and the corresponding more critical waves sample using a standard sea-keeping code
- Simulation of sea-keeping response with the CFD code imposing the selected wave time histories and the hull velocities. CFD code will provide the time series of the global and local loads on the hull girder
- Calculation of whipping response using a FE model (simple beam or detailed hull model) on which loads time series issues from CFD calculations are imposed.

The following scheme illustrates the recommended methodology issued from Virtue analysis.

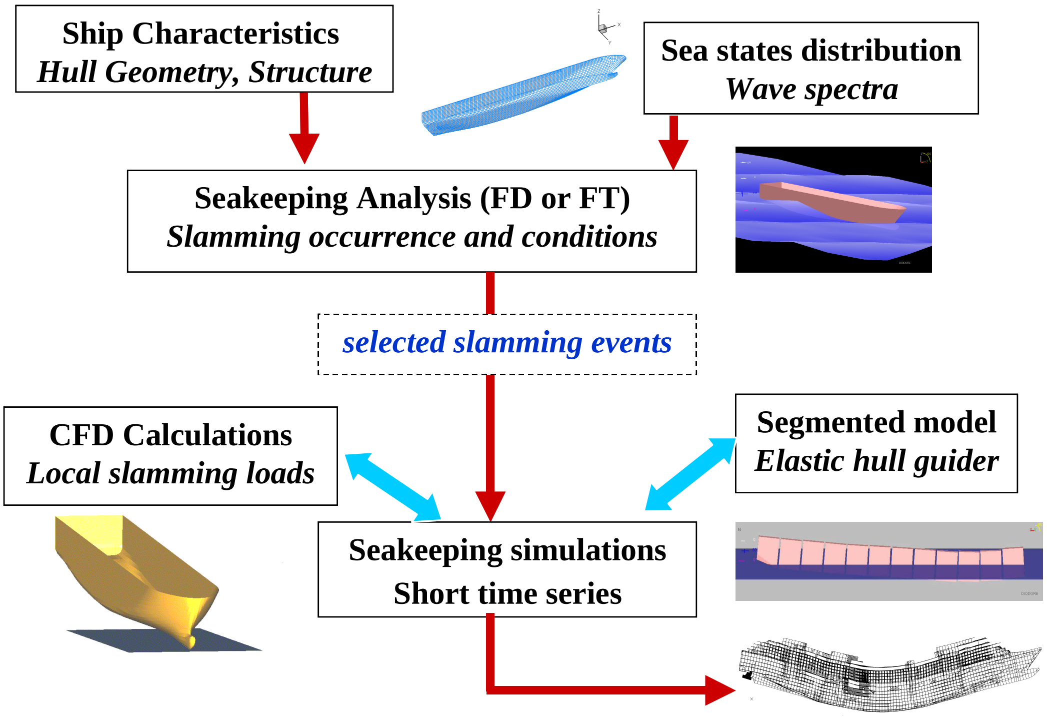

Figure 11: Methodology to predict slamming loads and induced whipping response

6 CONCLUSIONS

The results, shown in this report, point out that:

- Developments of VOF and SPH methods in CFD codes are validated,
- These free surface models, included in codes based on different numerical solver give comparable results for slamming problems regarding free surface evolution (especially the complex jet shape) and pressure field,
- These codes, validated on academic tests, can be used now to predict slamming loads on real ship geometry.

A methodology to couple standard sea-keeping analysis with the use of CFD calculations has been proposed to be included in the ship design process with realistic time consuming and cost.

Moreover, alternatives have been studied to progress on the way to the fully coupled fluid flow / hydro-elastic equations. Algorithms exist but time consuming remains still unrealistic today for a systematic use in the ship design process.

More accurate alternatives have been analysed, which could be used in the future with the benefit of new computer capabilities:

- Fully coupled approach between a time sea-keeping code and CFD code to impose at each time step slamming loads for the hydro-elastic response of the hull girder. This approach is necessary for hull having bending modes close to the wave frequency range
- Fully coupled approach by solving viscous fluid flow, free surface conditions and hydro-elastic response in the same code. The approach needs to be developed in EOLE using the same method than for the implementation of rigid body motions. Two different ways to proceed have been reviewed: use a modal decomposition of the ship structure or use FE coupling with the RANS algorithm.

The last approach is particularly attractive to analyse local hydro-elastic impact when structural deformation highly influences the fluid flow and the free surface evolution.

REFERENCES

[1] Kihara, H., Numerical Modeling of flow in water entry of a wedge, Proceeding of 4th international conference on numerical ship hydrodynamics pp 107-116, 1985.

[2] Greenhow, M., Wedge Entry into initially calm water, Applied Ocean Research, Vol. 9, No.4 214-223, 1987.

[3] R. Zhao, O. Faltinsen, Water entry of two-dimensional bodies, in J. Fluid Mech., 1993.

[4] Azcueta, R., Hadzic, I., Muzaferija, S. and Peric, M. Computation of flows with free surfaces, Marnet-CFD first annual workshop, 1999, Barcelona.

[5] Sames, P. C. and Schellin, T. E., Application of a two fluid finite volume method to ship slamming, Journal of Offshore Mechanics and arctic engineering (OMAE), Vol. 121, no. 1, 1999.

[6] Oger G., Doring M., Ferrant P., Allesandrini B., 2D SPH simulations of wedge water entry, YCPH, 2005.

[7] R. Zhao, O. Faltinsen, J. Aarnes, Water entry of arbitrary two-dimensional sections with and without flow separation, in 21th Symposium on Naval Hydrodynamics, 1997.

[8] Scolan Y.M., Hydroelastic behaviour of a conical shell impacting a quiescent free surface of incompressible liquid, Journ. of Sound and Vibrations, 2005.

[9] B. Peseux, L. Gornet, B. Donguy, Hydrodynamic impact: Numerical and experimental investigations, in Journal of Fluids and Structures 21, 2005.

[10] N. Couty, Three dimensional modelization of slamming impacts with an explicit finite element software. Application to fast ships, in ICCAS, 2002.

[11] Ledoux A., Molin B., Remy F.,Experimental and numerical study of the wave response of a flexible barge, 4th Int Conf. on Hydroelasticity in Marine engineering, 2006.

AKNOWLEDGMENTS

These works were carried out with a financial support from Europe within the European Sixth Framework Program (VIRTUE project).