

STUDY OF DIFFERENT METHODS FOR THE SIMULATION OF VERTICAL WATER ENTRY OF RIGID WEDGE

KAMESWARA SRIDHAR VEPA ⁽¹⁾, ALEXANDRU NILA ⁽²⁾, JORIS DEGROOTE⁽³⁾, DIEDERIK VAN NUFFEL⁽⁴⁾, WIM VAN PAEPEGEM ⁽⁵⁾, STEVE VANLANDUIT⁽⁶⁾, JAN VIERENDEELS ⁽⁷⁾

⁽¹⁾ Doctoral researcher, Ghent University,
Department of Material Science and Engineering, Zwijnaarde, 9052, Belgium. KameswaraSridhar.Vepa@ugent.be

⁽²⁾ Doctoral researcher, Vrije Universiteit Brussels,
Department of Mechanical Engineering, Brussels, 1050, Belgium. alexandru.nila@vub.ac.be

⁽³⁾ Post doctoral researcher, Ghent University,
Department of Flow, Heat and Combustion Mechanics, Gent, 9000, Belgium. Joris.Degroote@ugent.be

⁽⁴⁾ Doctoral researcher, Ghent University,
Department of Material Science and Engineering, Zwijnaarde, 9052, Belgium. Diederik.VanNuffel@UGent.be

⁽⁵⁾ Professor, Ghent University,
Department of Material Science and Engineering, Zwijnaarde, 9052, Belgium. Wim.VanPaepegem@UGent.be

⁽⁶⁾ Professor, Vrije Universiteit Brussels,
Department of Mechanical Engineering, Brussels, 1050, Belgium. svlandui@vub.ac.be

⁽⁷⁾ Professor, Ghent University,
Department of Flow, Heat and Combustion Mechanics, Gent, 9000, Belgium. Jan.Vierendeels@UGent.be

1. Introduction

Wave impact or slamming is a phenomenon characterized by large local pressures (10 bar or more) for very short durations (order of milliseconds). Slamming loads can cause severe damage to the structure (Sebastiani *et al.*, 2001).

It is a computationally challenging task to model the slamming wave impact as the distribution of hydrodynamic pressures is highly localised in space and time (Oger *et al.*, 2006). The shape of the body subjected to slamming is crucial in calculating the incident slamming loads (Mei *et al.*, 1999). Different numerical approximation methods are available for simulating the fluid structure interaction problems. Traditional mesh techniques use nodes and elements for approximating the continuum equations whereas particle methods like smoothed particle hydrodynamics (SPH) approximates the continuum equations using the kernel approximation technique and hence can also be used for a wide range of fluid dynamics problems (Zhao and Faltinsen, 1993).

In this work, the effect of slamming on two-dimensional Wedge shaped structures is studied using three solvers i.e., 1) SPH solver, 2) Explicit solver and 3) Implicit solver in comparison to the experimental (Particle Image Velocimetry) PIV measurements.

2. Numerical Methods

In case of mesh-based explicit and implicit solvers, wave slamming is modelled as a fluid-structure interaction problem with multi-phase flow field. Hence, numerical modelling of the slamming wave impact includes: 1) multiphase modelling of the free surface flow subjected to gravity load, 2) fluid structure interaction between the body and the fluids (air and water), and 3) contact modelling between the fluids as well as fluid and structure.

Multiphase modelling of free surface flow is done numerically by solving the continuity equation and Navier-Stokes equations with kinematic and dynamic boundary conditions imposed on the flow field. Eq. [1] represents the continuity equation where V is the fluid velocity field. Eq. [2] shows the Navier-Stokes equations where ρ represents density, p represents pressure and ν represents viscosity. Kinematic boundary conditions are used to specify the conditions like no-slip and no-flux boundaries. Eq. [3] shows no slip boundary condition near the solid boundary which represents finite shear stress. Here, U is the velocity of the solid boundary. Eq. [4] shows the no-flux boundary condition at the solid boundary which represents a continuous flow. Dynamic boundary conditions are used to specify the dynamics of the boundary like the stress continuity. In case of a free surface, pressure outside the fluid is considered constant and is equal to atmospheric pressure. This is shown in eq. [5].

$$\nabla \cdot V = 0 \quad [1]$$

$$\frac{\partial v}{\partial t} + (\vec{v} \cdot \nabla) \vec{v} = -\frac{1}{\rho} \nabla p + \nu \nabla^2 \vec{v} \quad [2]$$

$$\vec{V} \cdot \vec{t} = \vec{U} \cdot \vec{t} \quad [3]$$

$$\vec{V} \cdot \vec{n} = \vec{U} \cdot \vec{n} \quad [4]$$

$$\frac{dp}{dt} = 0 \quad [5]$$

The SPH method has the characteristics of a lagrangian method and is successfully employed for modelling free surface flows. The Lagrangian characteristics of the SPH method help in accurately modelling the breaking of interface. Hence it is applied for modelling the free surface impact problems as in this case. The basic equation solved is the Navier-Stokes equation as given above in eq. [2]. An interpolation scheme as shown in eq. [6] is used for calculating the lagrangian derivatives at each point in the domain D. Finally, numerical integration is done using a quadrature formula like the one shown in eq. [7] at every integration point i.

$$f(\vec{r}) \approx \int_D f(\vec{x}) W(\vec{r} - \vec{x}, h) d\vec{x} \quad [6]$$

$$\int_D g(\vec{x}) d\vec{x} \approx \sum_i g_i w_i \quad [7]$$

3. Experimental setup

The PIV system used in the experiments is a Quantronix Darwin Duo Nd:YLF double pulsed laser up to 10kHz repetition rate, and a 12-bit Photron Fastcam SA1.1 camera, with a CMOS sensor and 1024 x 1024 pixel resolution. Experimental setup shown in Figure 1 is a Plexiglas water tank of 1m x 0.6m x 0.5m. A Copley Controls XM3804S ServoTube Module (with 12 micron repeatability and 350 micron absolute accuracy) motor is used to control the vertical movement of the wedge. Hollow glass spheres of 10 μm mean diameter are used for seeding.

4. Results

A commercial code, LS-DYNA is used for modelling the water entry of a wedge. Since this is a 2D simulation, nodal degrees of freedom are constrained in the third direction. The boundaries are also modelled using shell elements and a contact algorithm is employed for modelling the interaction between the container walls and the SPH particles (water). The timestep size is calculated based on the Courant-Friedrichs-Lewy (CFL) condition.

Figure 1 shows the experimental and numerical setups and corresponding velocity fields. The velocity fields shown here are for a 90° wedge compared for 0.037 seconds. Results show that mesh refinement/ particle size is very critical in getting a convergence of results. Refinement level of PIV is much coarser compared to numerical model. Other critical parameters effecting the convergence of numerical results include: 1) Equation of state for water and air, 2) Choice of kernel function for the SPH method, and 3) Volume fraction control algorithm.

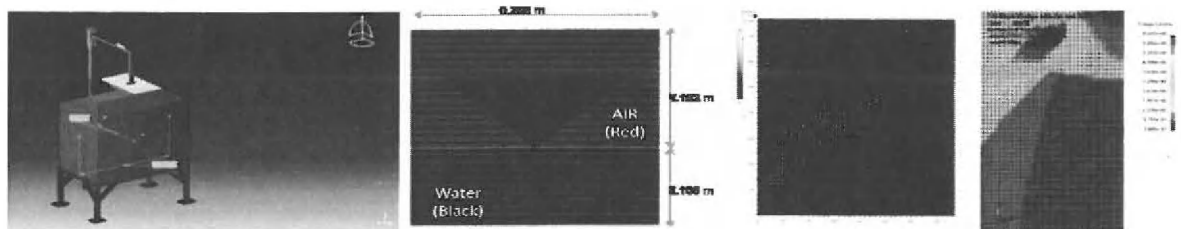


Figure 1. Comparison of PIV and numerical modeling of wedge

5. Acknowledgments

The authors are highly indebted to the university research fund FWO for sponsoring this research.

References

- Mei X., Y. Liu and K. P. Yue Dick 1999. 'On the water impact of general two-dimensional sections' *Applied Ocean Research* 21,1 1-15.
- Oger G., M. Doring, B. Alessandrini and P. Ferrant 2006. 'Two-dimensional SPH simulations of wedge water entries' *Journal of Computational Physics* 213,2 803-822.
- Sebastiani L., F. Valdenazzi, L. Grossi and G. K. Kapsenberg 2001. 'A theoretical / experimental investigation of the slamming pressures on fast monohull vessels' *FAST-2001*.
- Zhao R. and O. Faltinsen 1993. 'Water entry of 2-dimensional bodies' *Journal of Fluid Mechanics* 246, 593-612.