Internal Seminar Series 2011, April 13th 2011
George Begg Building, Room B8, 15:00
School of MACE, The University of Manchester.

Multi-phase Modelling of Violent Hydrodynamics using SPH

Athanasios Mokos

athanasios.mokos@postgrad.manchester.ac.uk

Key words: Multi-phase flows, SPH, dam break

1 Abstract

SPH is a relatively new, rapidly evolving Lagrangian meshless method that is currently being applied in a large variety of cases, ranging from astrophysics to nuclear engineering. However, most of these cases have been treated so far as single phase cases, therefore limiting both the accuracy of the method and the range of cases that can be solved and importantly the physics captured. However, several models have been developed in recent years, allowing SPH to model multi-phase flows. This study is mainly focused in two-phase flows that include the interaction of two fluids: air and water which have a large density ratio of 1:1000. Understanding the interactions between them is essential for a large number of engineering cases, such as wave breaking as shown in Figure 1, fuel tank sloshing and dam break. Two distinct test cases have been selected for this project, both including interactions between water and air. In order to model them, we have selected the model of Colagrossi and Landrini (2003), which, despite being fairly simple has been extensively tested by Rogers et al. (2009), producing sufficiently accurate results. Modelling and testing has been done so far in Fortran, but the focus will soon shift to GPU programming, as this will allow us to model cases in a fraction of the time that CPU programming requires, enabling us to target larger scale cases.

Figure 1: Wave Over-topping (University of Plymouth 2011)

2 Results

The still water case was extensively tested for the errors both in the particle position and in their pressure. Several modelling improvements, such as a zeroth-order density filter and a variable time step were also used. A convergence study of the pressure errors was performed for different particle resolutions, shown in Figure 2, while the water has converged for a fairly coarse distribution, an increase in the number of particles is needed in order for the air phase to converge. In order to check whether the particles remain still we are using a relative error the global relative error for both coordinates is also calculated. In order for the computation to be correct, the value of this error should be less than 10^{-3} for any particle at the computational field. As we can see from Figure 3 this condition is fulfilled.

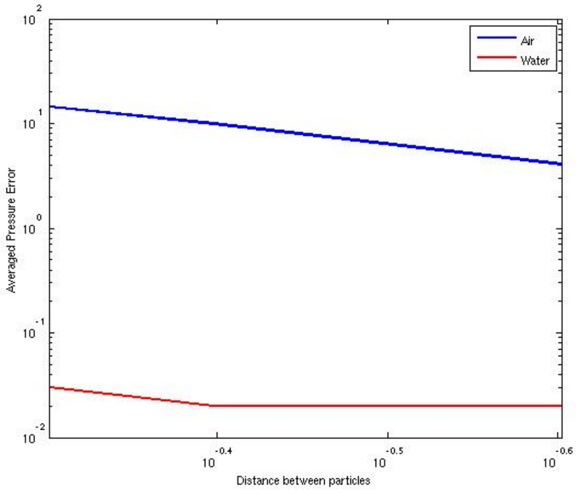

Figure 2: Convergence of the two phases for different particle resolutions.

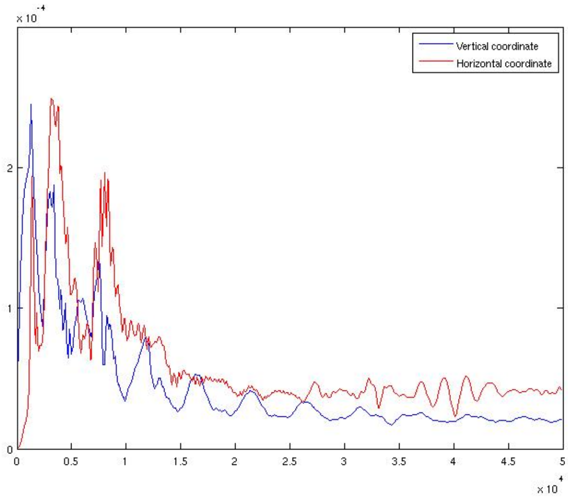

Figure 3: Maximum relative error for each time-step

The modelling of the dam break case is based on the still water case. It is a much more dynamic test case as it involves water movement, large deformations and high-order phenomena. Phase interaction is also very important as, due to the water flow, there is mixing between the two phases, which significantly alters the pressure field and the behaviour of the flow. Several test runs with different particle resolutions (see Figs. 4 and 5) were performed and the simulations were compared to the experimental results of Koshizuka and Oka (1996).

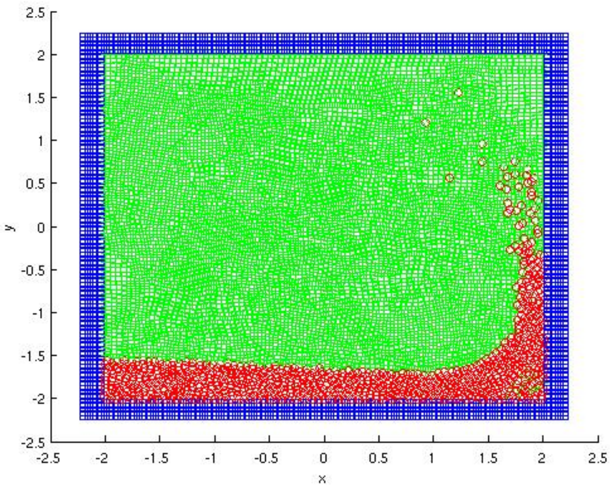

Figure 4: Particle arrangement after 190000 time steps

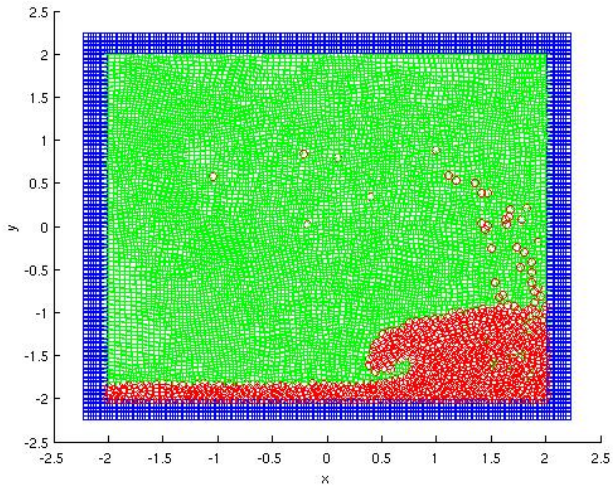

Figure 5: Particle arrangement after 330000 time steps

3 Ongoing and Future Work

Work is currently being done on improving the calculation of the boundary particles and their interaction with the fluid particles. Building on the work of Ferrand et al. (2010), a re-normalization term of the density field near the solid wall boundaries will be used, in conjunction with a multi-phase model. This term will correct the density with respect to the missing kernel and it will be computed with regard only to the position of the particles. Work is also being done on modelling the enrgy equation. The Colagrossi and Landrini (2003) model is using an incompressible equation of state with some additional terms in order to model air particles. However, if we change this equation to one suited for compressible fluids (such as the Ideal Gas Law), we will obtain more accurate results, but, in order to make this change, the energy equation must also be solved.

References

Colagrossi, A. and M. Landrini (2003). Numerical simulation of interfacial flows by smoothed particle hydrodynamics. *Journal of Computational Physics 191*(2), 448 – 475.

Ferrand, M., D. Laurence, B. Rogers, and D. Violeau (2010). Improved time scheme integration approach for dealing with semi analytical boundary conditions in SPARTACUS2D. In *5th SPHERIC Workshop, Manchester 2010*.

Koshizuka, S. and Y. Oka (1996, Jul). Moving-particle semi-implicit method for fragmentation of incompressible fluid. *Nuclear Science and Engineering 123*(3), 421–434.

Rogers, B., J. Leduc, J. Marongiu, and F. Leboeuf (2009). Comparison and Evaluation of Multi-phase and Surface Tension models. In *4th SPHERIC Workshop, Nantes 2009*.