EFFICIENT SIMULATION OF COMPLEX NON-HYDROSTATIC THREE-DIMENSIONAL FREE SURFACE FLOWS USING SMOOTH-PARTICLE-HYDRODYNAMICS (SPH)

Angela Ferrari and Aronne Armanini

[1]Department of Civil and Environmental Engineering, University of Trento, Italy, Via Mesiano, 77, I-38123 Trento
E-mail: ferrari.angela@gmail.com

Abstract

We present a robust, efficient and accurate SPH scheme that is able to track complex three-dimensional non-hydrostatic free surface flows and which produces an accurate and little oscillatory pressure field. The scheme uses an explicit third order TVD Runge–Kutta time integrator, together with a monotone upwind flux for the density equation. For the discretization of the velocity equation, a non-diffusive central flux has been used. In our formulation, there are no parameters to tune and there is no need for any artificial viscosity term. To assess the accuracy of the new SPH scheme, a 3D mesh-convergence study is performed for the strongly deforming free surface in a 3D dam-break and wave-impact test problem. Moreover, the parallelization of the new 3D SPH scheme has been carried out using the message passing interface (MPI) standard, together with a dynamic load balancing strategy, which is needed to improve the computational efficiency of the scheme. Thus, simulations involving millions of particles can be run on modern massively parallel supercomputers, obtaining very good performance, as confirmed by a speed-up analysis.

The 3D applications consist of environmental flow problems, such as dam-break flows, overtopping flows over a sharp-crested weir and impact flows against a wall. The numerical solutions obtained with our new 3D SPH code have been compared with either experimental results or with other numerical reference solutions, obtaining in all cases a very satisfactory agreement.

Introduction

The governing equations of three-dimensional weakly compressible inviscid fluid motion are given by the compressible Euler equations which read as follows, using the total (material) derivative:

$$\frac{d\rho}{dt} + \rho\nabla\cdot\boldsymbol{v} = 0$$
$$\frac{d\boldsymbol{v}}{dt} + \frac{1}{\rho}\nabla p = \boldsymbol{g} \tag{1}$$

The position of each infinitesimal fluid particle is governed by the following system of ordinary differential equations:

$$\frac{d\boldsymbol{x}}{dt} = \boldsymbol{v} \tag{2}$$

The system of governing equations (1) and (2) is closed by the equation of state (EOS), which links the pressure to the density. Here, we use the classical Tait equation of state, also used in literature [5], which provides a valid model for weakly compressible flows:

$$\boldsymbol{p_1} = \boldsymbol{k_0}\left(\left(\frac{\rho}{\rho_0}\right)^{\gamma} - \boldsymbol{1}\right) \tag{3}$$

Here k_0 is a constant that governs the compressibility of the fluid and hence the speed of sound, ρ is the water density, ρ_0 is the liquid reference density at atmospheric standard conditions (p=0) and γ is a parameter that is used to fit the EOS with experimental data.

Numerical method

The semi-discrete form of our SPH discretization of the system (1) and (3) reads for each particle i as follows [1]:

$$\frac{d\rho_i}{dt} = -\sum_{j=1}^{N} m_j\left(\left(\boldsymbol{v}_j - \boldsymbol{v}_i\right)\cdot\nabla_{\mathrm{i}}W_{ij} - \boldsymbol{n}_{ij}\cdot\nabla_{\mathrm{i}}W_{ij}\left(\frac{c_{ij}}{\rho_j}\left(\rho_j - \rho_i\right)\right)\right)$$
$$\frac{d\boldsymbol{v}_i}{dt} = -\sum_{j=1}^{N} m_j\left(\frac{p_i}{\rho_i^2} + \frac{p_j}{\rho_j^2}\right)\nabla_{\mathrm{i}}W_{ij} + \boldsymbol{S}_i$$
$$\frac{d\boldsymbol{x}_i}{dt} = \boldsymbol{v}_i \tag{4}$$

In the first equation, a Rusanov-type flux has been used. Here, c_{ij} denotes the maximum sound speed between two particles i and j and W_{ij} is the usual smoothing kernel, which is characteristic for the SPH method. Here, we use the classical cubic B-spline kernel given by the expression

$$W_{ij} = \frac{K}{h_{ij}^d} \begin{cases} \frac{2}{3} - q_{ij}^2 + \frac{1}{2} q_{ij}^3 & \text{if } 0 \le q_{ij} < 1 \\ \frac{1}{6}(2 - q_{ij})^3 & \text{if } 1 \le q_{ij} \le 2 \\ 0 & \text{if } q_{ij} > 2 \end{cases} \tag{5}$$

Computational Results

In the following sections, some numerical results are shown to illustrate the capabilities of the method.

Elliptic drop

This test problem is taken from [5] and is used to assess the capability of the numerical method to simulate incompressible flows. The computational domain is initially at time t=0 a circle of radius R = 1 and the initial velocity field is $\mathbf{v}$ = (-100x,100y). A total number of 1858 particles has been used for this test problem and the coefficient k_0 is chosen such as to obtain a speed of sound of 1400 *m/s*. The exact solution of the problem is given at various times in [5] and is depicted in Figure 1 as thick solid line. The corresponding numerical solution given by the distribution of the fluid particles is shown as well. We note an excellent agreement for the free surface location between exact and numerical solution.

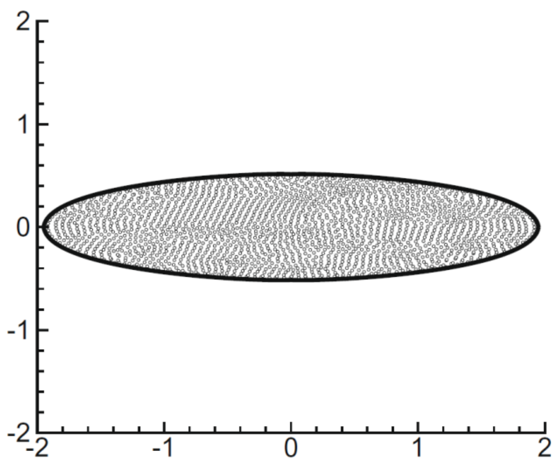

Figure 1: Exact (think solid line) and numerical solution (particles) for the elliptic drop test case.

Numerical convergence study

This test case has been originally proposed by Colagrossi and Landrini in two space dimensions [6]. In [1] it was extended to a three-dimensional dam break problem with following impact low against a rigid vertical wall. The initial conditions is given by an initial reservoir height as H = 0.6 m, the length and height of the channel are d = 5.366H and D = 3.0H, according to [6]. For our 3D version, we set the channel width to W = H = 0.6 m. The numerical solution has been computed with the new SPH formulation (4) for an ideal fluid with four different levels of refinement: 250,000, 500,000, 1,000,000 and 2,000,000 fluid particles. Fig. 2 shows our computational results using 2,000,000 points at various times (t = 0.6 s, 1.2 s, 1.5 s and 2 s) using 256 CPUs. In Figure 3 the solution is shown at various times for three different numbers of particles. One can observe that the method has reached convergence using about 500,000 particles.

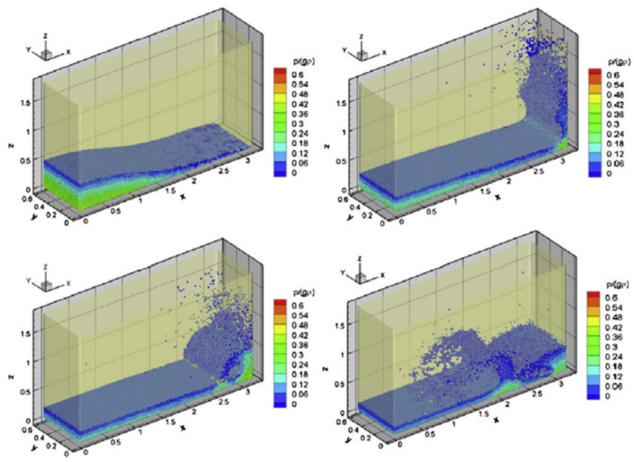

Figure 2: 3D view of the dambreak and wave-impact flow problem.

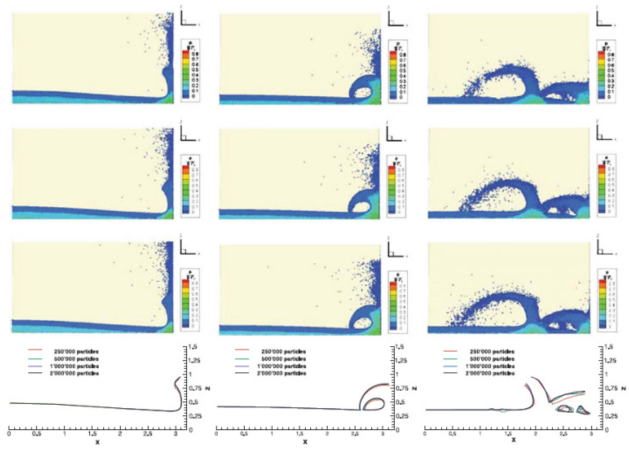

Figure 3: Mesh convergence study for the dambreak and wave-impact flow problem using 500,000, 1,000,000 and 2,000,000 particles.

CADAM Problem: Dambreak in a curved channel

This test problem is a classical test case to evaluate the accuracy of shallow water type models. Here we solve the fully three-dimensional version of the problem without assuming hydrostatic pressure. The channel geometry is depicted in Figure 4, while some 3D results are presented in Figure 5. In Figure 6 the 3D SPH solution is compared

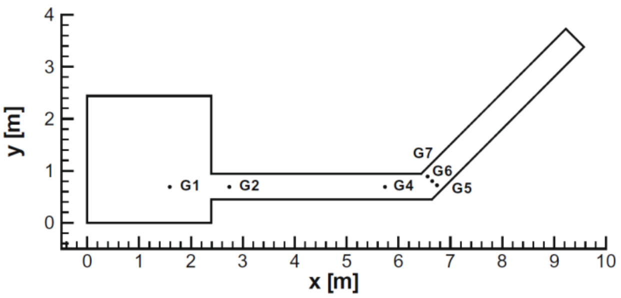

Figure 4: Channel geometry for the CADAM test case.

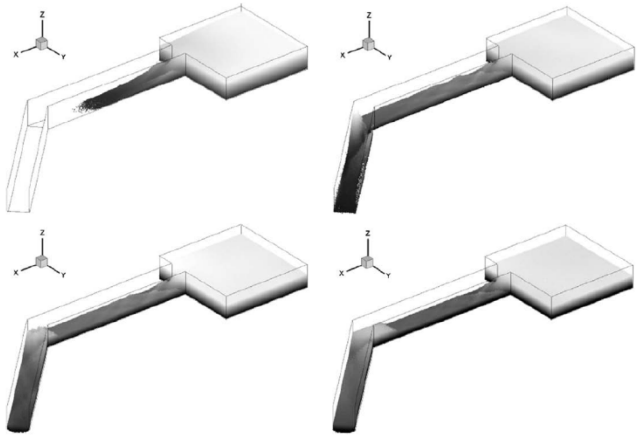

Figure 5: 3D view of the solution of the CADAM problem.

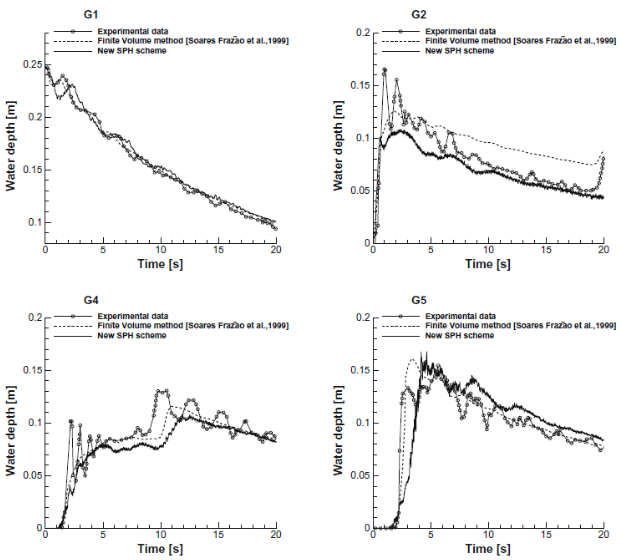

Figure 6: Comparison of 3D SPH solution, shallow water solution and experimental results.

against the solution of the two-dimensional shallow water equations using a finite volume scheme. The experimental water levels are also shown for comparison.

Three-dimensional Dambreak Problem

In this section, a truly three-dimensional dambreak flow problem is solved. The setup is taken from the popular paper by Fraccarollo and Toro [7], where also all the details about the geometry of the test problem are given. In reference [7] the authors compare the solution of the 2D shallow water equations with experimental measurements. In [2] the authors proposed for the first time a fully three-dimensional, non-hydrostatic computation of this test problem. In the following, some of the most important results are summarized. Figure 7 shows the 3D evolution of the free surface, both, computed with a 2D shallow water model and computed by solving the three-dimensional Euler equations, see systems (1) and (2).

The most important results are obtained by comparing the experimental water depths and the experimental velocity profiles with the results obtained for the shallow water model and the 3D Euler equations, see Figures 8 and 9. One finds that the 3D Euler model yields much better results compared to the shallow water model. This is due to the

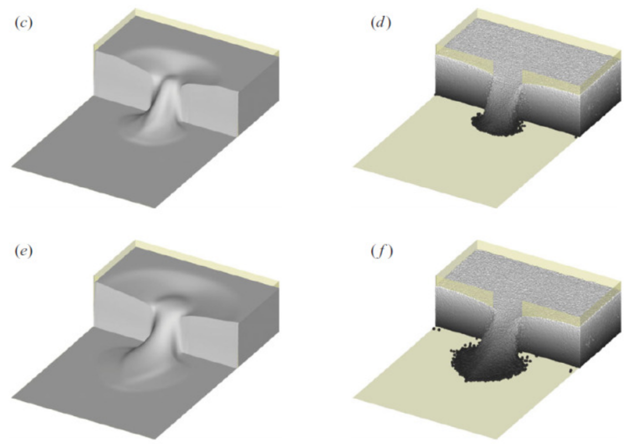

Figure 7a: 3D view of the solution of the 3D dambreak problem. Left: shallow water equations. Right: 3D Euler equations.

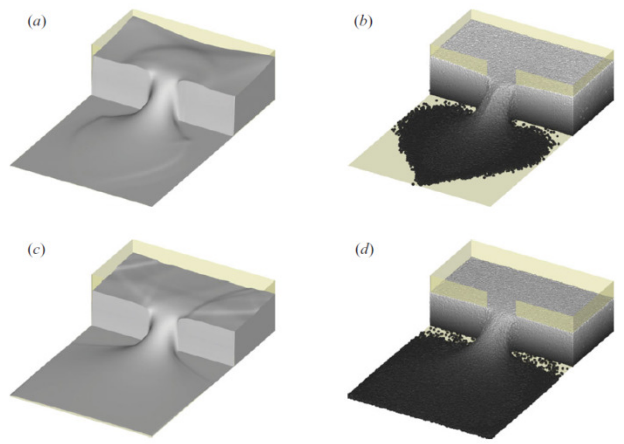

Figure 7b: 3D view of the solution of the 3D dambreak problem. Left: shallow water equations. Right: 3D Euler equations.

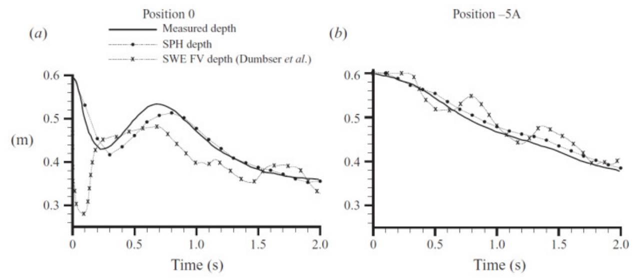

Figure 8: Comparison of experimental water depth, shallow water model and 3D Euler equations.

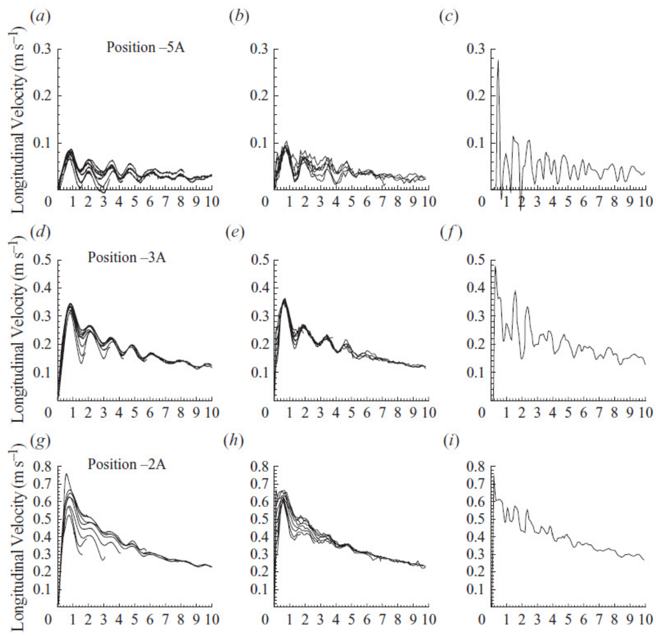

Figure 9: Comparison of experimental water depth, shallow water model and 3D Euler equations.

fact that at short times after the dambreak, the shallow water assumptions are **not** valid, since the vertical accelerations cannot be neglected in this case.

Flow Over a Sharp-Crested Weir

The last application concerns overtopping flow over a sharp-crested weir. This problem is solved in detail in [3] and [4], where also the empiric formula for the profile of the lower streamline is given. Here, only a short summary of the results is shown in Figures 10 and 11. One can see that the numerical solution for the lower streamline is in excellent agreement with the experimental measurements.

Conclusions

In this paper, a robust and parameter-free SPH method has been introduced. The numerical algorithm uses a monotone upwind flux for the density evolution and a central flux to compute the momentum equation. A classical explicit third order TVD Runge-Kutta is used for time integration.

The accuracy of the SPH scheme has been assessed over a wide set of test cases involving millions of SPH particles and strong free surface deformations. A good agreement has been obtained from the comparison with experimental reference solutions.

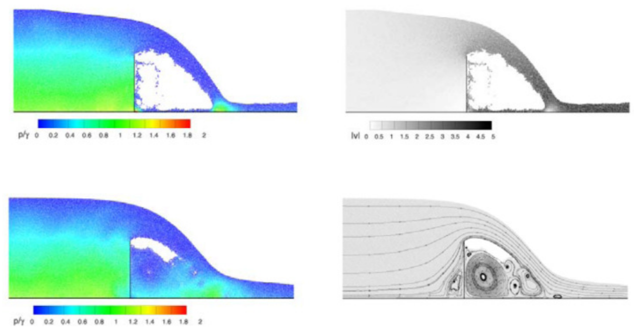

Figure 10: Instantaneous pressure distribution (left) and streamlines (right) for the flow over a sharp-crested weir.

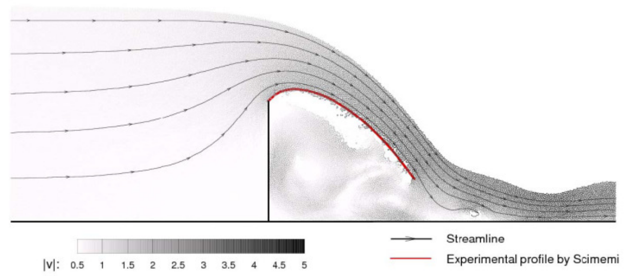

Figure 11: Comparison of the numerical results obtained for the lower streamline with the experimental measurements.

References

[1] A. Ferrari, M. Dumbser, E.F. Toro and A. Armanini (2009). A New 3D Parallel SPH scheme for Free Surface Flows. *Computers & Fluids*, **38**:1203-1217.

[2] A. Ferrari, L. Fraccarollo, M. Dumbser, E.F. Toro and A. Armanini (2010). Three-Dimensional Flow Evolution after a Dambreak. *Journal of Fluid Mechanics*, **663**:456-477.

[3] A. Ferrari (2010). SPH simulation of free surface flow over a sharp-crested weir, *Adv. Water Resour.* **33**:270–276.

[4] M. Dumbser (2011). A simple two-phase method for the simulation of complex free surface flows, *Comput. Methods Appl. Mech. Engrg.* **200**: 1204–1219.

[5] J.J. Monaghan (1994). Simulating free surface flows with SPH, *Journal of Computational Physics,* **110**:399–406.

[6] A. Colagrossi, M. Landrini (2003), Numerical simulation of interfacial flows by smoothed particle hydrodynamics. *J Comput Phys*;**191**:448–475.

[7] L. Fraccarollo and E.F. Toro (1995). Experimental and numerical assessment of the shallow water model for two-dimensional dam-break type problems. *J. Hydraul. Res.* **33**:843–864.