

**Abstract:** Smoothed Particle Hydrodynamics (SPH) is a Lagrangian mesh-free method for Computational Fluid Dynamics (CFD) that is advancing in the field of lava simulations. Thanks to its properties, SPH copes very well with the issues that generally arise when dealing with lava flows, like the simulation of a free surface, the interaction with irregular topographies and the high dynamicity of the flow. A major aspect of simulating a lava eruption concerns the generation of the flow. Several approaches are possible in SPH, with different levels of implementation complexity, performance and quality of the generated flow, the latter affecting the entire emplacement process. Here we present a comparison of two different inlet designs simulated using GPUSPH, an implementation of weakly-compressible SPH running fully on Graphics Processing Units (GPUs) developed at the Tecnolab of INGV-Catania. A simple inlet, obtained as an injection of fluid by means of a piston, is compared to a more sophisticated model, where new fluid particles are generated during the simulation and open boundary conditions are imposed using Riemann invariants. A description of the two models will give a view of the pro and cons in terms of implementation, while a comparison of the two flows with the analytical results will highlight the effects of the inlet model in the emplacement. The overall model is validated, giving a first order convergence.

## Validation of GPUSPH for lava flows

The model has been validated according to several benchmark cases regarding the main features implemented in GPUSPH and the principal characteristics of lava flows. Both the thermal and mechanical models have been tested.

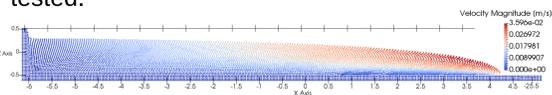


Fig. 8. Lateral view of a viscous dam-break. An amount of viscous fluid is suddenly left free to slump onto an horizontal plane. The validation is performed with respect to the evolution over time of front position and the thickness.

The main approach for the validation has been the convergence test with respect to the resolution, giving as result the order of convergence.

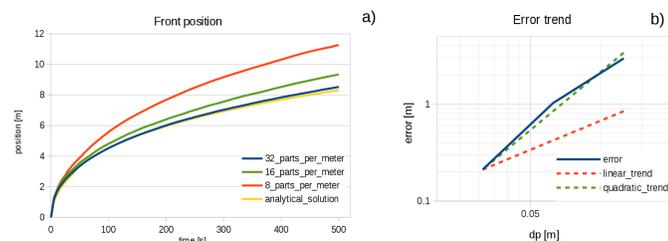


Fig. 9. a) front position for the viscous dam break. The different curves are obtained varying the resolution. b) convergence trend. The errors are plotted on a log-log reference, as a function of the discretization step.

The order of convergence is obtained looking at the variation of the errors with respect to the discretization interval. For the viscous dam break (Fig. 8 and 9) a second order convergence has been obtained, a good result in the SPH field. Other simulations involving inlets and other sources of disturbances gave convergence orders close to the linear one.

## Simulating a volcanic vent

A fundamental role in the simulation of lava flows is covered by the vent. Modeling a vent implies the management of two fundamental aspects: the input of new particles in the simulation domain and the establishment of a well defined flow rate. The first aspect can be easily treated using some holed reservoirs, as shown in Fig. 1, but this approach makes hard to prescribe a defined flow rate. This issue can be overcome using a piston structure. Anyway, a more sophisticated approach can involve open boundaries that allow a smarter management of flow and flow rate generation. Here we will show the details of these two approaches.

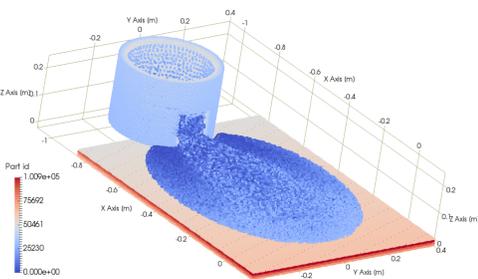


Fig. 1. Primitive version of vent mode: a holed crucible.

## Simulating an eruption using open boundaries

### Geometrical aspect

The vent is constituted by a buffer region, obtained as a portion of boundary, where new particles are generated. Unlike the case involving a piston, here no flow has to pass through any hole, then this kind of vent can have any desired shape. In this case we use a round geometry.

### Numerical aspect

There is no reservoir involved and the size of the inlet is smaller, therefore this implementation does not affect the speed of sound, then the performance.

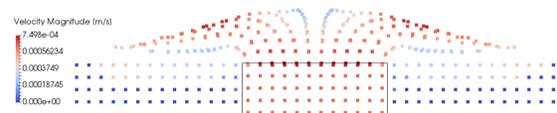


Fig. 5. Lateral view of an inlet implemented using open boundaries. The particles in the rectangular box constitute the buffer. Particles are colored by velocity magnitude.

### Implementation aspect

The picture on the left illustrates the constructive aspect of an inlet. The particles in the square constitute the buffer region. These particles are used to give continuity to the system domain. As first, they give continuity

to the domain, required by the SPH smoothing. Then the continuity to the pressure and velocity fields is also ensured; this last aspect is carried out by exploiting the Riemann invariants of the flow. The velocity and density of the flow are average in correspondence of a grid of points adjacent the inlet considering the Riemann invariants in the region of domain close to the inlet. These averaged quantities are then projected in the buffer region according to the nature of the open boundary. Here, we implement an inlet with imposed velocity, then the density is projected in the buffer region according to [4].

### Results

The plot on the right shows the convergence test for the viscous flow generated by means of an open boundary. Unlike the case involving a piston, thanks to the smaller speed of sound and the smaller number of particles, due to the absence of a reservoir and the piston structure, we can run simulations at higher resolution. In this way we avoid the artifacts appearing in the low resolution simulation involving the piston, where the flow front shows a trembling behavior due to its under resolved state.

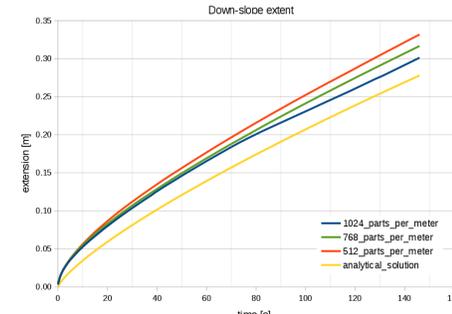


Fig. 6. Convergence test for the down-slope extent of an inclined viscous flow, obtained using open boundaries.

### Performance

The high resolution simulation involves 2'799'236 particles and a time step  $2.1 \cdot 10^{-5}$ s. Running on a TITAN X Maxwell GPU, one second of evolution is simulated in around 1'740s. The intermediate resolution simulation involves 1'536'444 particles and a time step  $2.67 \cdot 10^{-5}$ s, with one second of evolution being simulated in around 738 s. Finally, the low resolution simulation involves 696'308 particles and a time step  $4.1 \cdot 10^{-5}$ s, with one second of evolution being simulated in around 156 s.

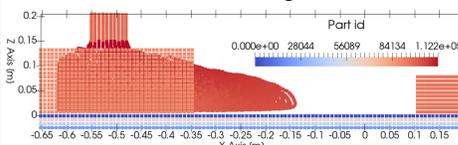


Fig. 7. Real lava flow simulated using the semi-implicit formulation and inlet simulated by means of open boundaries.

### Advantage at high viscosity

GPUSPH has recently included a semi-implicit integration scheme [2] to deal with high viscosities. This becomes particular convenient when the number of particles is reduced, thus taking great advantage from this kind of vent model.

## Simulating an eruption using pistons

### Geometrical aspect

The piston we implemented involves only squared geometries. This ensures that the spacing between all particles is the same, giving robustness to the simulation.

### Numerical aspect

The size and shape of the piston has an important impact on the performances and in the numerical stability of the simulations. In fact, we want a piston large enough to contain all the fluid volume that will be erupted during the flow, but we have some limits in both height and width of the chamber. Too high a chamber would imply a high fluid column, leading to a high artificial speed of sound, then to a small time step. This affects the simulation performance, increasing the simulation time, and can become a serious numerical problems when the time step drops to values that cause substantial rounding in the integration process.

On the other side, too large a chamber would determine small velocities of the piston, that, for very low effusion rates, could suffer for numerical precision issues during the integration process. Moreover, the more the ratio between the vent width and the piston width is bigger than 1, the more the flow will be disordered. This constitutes a drawback, since it is known that a disordered flows show lower convergence orders.

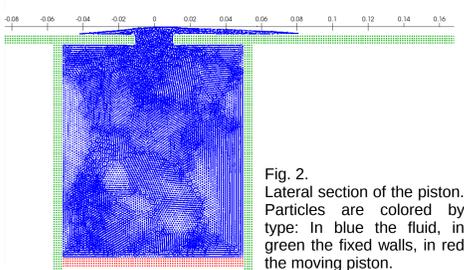


Fig. 2. Lateral section of the piston. Particles are colored by type: In blue the fluid, in green the fixed walls, in red the moving piston.

### Implementation aspect

The piston is constituted by a parallelepiped shaped chamber, realized with the same criterion used to implement the solid walls (in this case we use the Dummy boundary model). One of the walls is made moving, with a prescribed velocity. The opposite wall has a hole, in our case squared, for the sake of robustness, as said above.

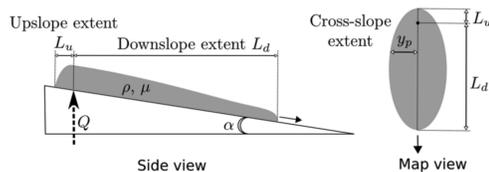


Fig. 3. Setup of the inclined viscous flow. (Image from B. Cordonnier et al., 2016)

### Performance

The high resolution simulation involves 2'761'836 particles and a time step  $4.15 \cdot 10^{-6}$ s. Running on a TITAN X Maxwell GPU, one second of evolution is simulated in around 7'165 s. The intermediate resolution simulation involves 1'171'807 particles and a time step  $6.22 \cdot 10^{-6}$ s, with one second of evolution being simulated in around 1'673 s. Finally, the low resolution simulation involves 645'444 particles and a time step  $8.27 \cdot 10^{-6}$ s, with one second of evolution being simulated in around 646 s.

### Results

The plot below (Fig.4) shows the results of a validation test for a flow generated by the piston. The problem is a viscous flow on an inclined plane, as depicted on the right (Fig.3). We performed a convergence test using several resolutions. The test shows convergence on the down slope extent of the flow: increasing the resolution, the simulated solution better reproduces the analytical solution.

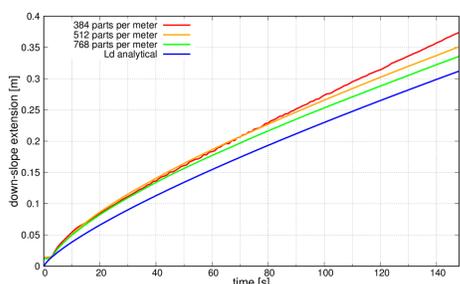


Fig. 4. Convergence test for the down-slope extent of an inclined viscous flow, obtained using an injection by piston.

## Conclusions and future work

- We have seen two possible ways to simulate a prescribed flow rate from a vent. Both approaches give a convergence in the results.
- Open boundaries reduce the number of particles involved in the simulation and the simulation time, offering also an enlargement of the time step and an improvement in terms of stability. The piston offers a higher simplicity in the implementation.
- The simulations involving a piston seem more accurate. In fact, they require more stringent conditions in parameters like the speed of sound, that give better results, paid in terms of simulation time. Higher value for the speed of sound can be used in the open boundary case to have better results.
- The validation process revealed convergence orders ranging around 1 and 2.

### References:

- [1] Zago V., G. Bilotta, A. Cappello, R. A. Dalrymple, L. Fortuna, G. Ganci, A. Héroult, C. Del Negro (2017). Simulating complex fluids with Smoothed Particle Hydrodynamics, *Annals of Geophysics*, 60(6), PH669, 2017. doi:10.4401/ag-7362.
- [2] Zago V., G. Bilotta, A. Héroult, R. A. Dalrymple, L. Fortuna, A. Cappello, G. Ganci, C. Del Negro (2018). Semi-implicit 3D SPH on GPU for lava flows. *Journal of Computational Physics*, in press.
- [3] Zago V., G. Bilotta, A. Cappello, R. A. Dalrymple, L. Fortuna, G. Ganci, A. Héroult, C. Del Negro. Preliminary validation of lava benchmark tests on the GPUSPH particle engine. *Annals of Geophysics*, 2018.
- [4] M. Ferrand et al. (2017). Unsteady open boundaries for SPH using semi-analytical conditions and Riemann solver in 2D, *Computer Physics Communications*. <http://dx.doi.org/10.1016/j.cpc.2016.09.009>