



**HAL**  
open science

## Smoothed Particle Hydrodynamics in OpenFOAM for jet application

Nicolas Hecht, Felix Dabonneville, Grégory Pinon, Julien Reveillon, F.X. Demoulin, Elie Rivoalen, Jean-Marc Cherfils

► **To cite this version:**

Nicolas Hecht, Felix Dabonneville, Grégory Pinon, Julien Reveillon, F.X. Demoulin, et al.. Smoothed Particle Hydrodynamics in OpenFOAM for jet application. 12th OpenFOAM Workshop, Jul 2017, Exeter, United Kingdom. hal-02144964

**HAL Id: hal-02144964**

**<https://normandie-univ.hal.science/hal-02144964>**

Submitted on 27 May 2024

**HAL** is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.

## Smoothed Particle Hydrodynamics in OpenFOAM for jet application

Nicolas HECHT<sup>1</sup>, Felix DABONNEVILLE<sup>2</sup>, Gregory PINON<sup>3</sup>, Julien REVEILLON<sup>4</sup>, Francois-Xavier DEMOULIN<sup>5</sup>,  
Elie RIVOALEN<sup>6</sup>, Jean Marc CHERFILS<sup>7</sup>

<sup>1</sup>LOMC, UMR 6294 CNRS-Normandie Université, France, nicolas.hecht@univ-lehavre.fr

<sup>2</sup>CORIA, UMR 6614, Normandie Université, France, felix.dabonneville@coria.fr

<sup>3</sup>LOMC, UMR 6294 Normandie Université, France, gregory.pinon@univ-lehavre.fr

<sup>4</sup>CORIA, UMR 6614, Normandie Université, France, julien.reveillon@coria.fr

<sup>5</sup>CORIA, UMR 6614, Normandie Université, France, demoulin@coria.fr

<sup>6</sup>LOFIMS, EA 3828, INSA de Rouen, France, elie.Rivoalen@insa-rouen.fr

<sup>7</sup>LOFIMS, EA 3828, INSA de Rouen, France, jean-marc.cherfils@insa-rouen.fr

**Keywords:** Lagrangian methods, Smoothed Particle Hydrodynamics, Sprays and Injection

This project is part of labex EMC3 also call Energy Materials and Clean Combustion Center . The purpose is to reduce the emissions of pollutants, for this it is necessary to understand all the phenomena involved in the combustion chambers. It is in this component that fuel is consumed and pollutants are generated. Its mastering is thus essential to optimize motors and to minimize the cost of external depollution components. The physical principles occurring in the combustion chamber can be summarized as follow : the injection system atomize the fuel which will evaporate and be mixed inside the chamber. Once the mixing is sufficiently homogeneous, combustion can takes place.

The key and starting point is the atomization process. If it is wrongly predicted, it will influence evaporation and mixing inside the chamber. Numerical simulation has become today one essential tool for designing and optimizing injecteur since the difficulty to instrument them for experiments.

Simulate primary atomization and mixing area in combustion chamber is not feasible in actual conditions of LES and DNS developments. Constraints come from the Eulerian formalism which needs, in the region close to the nozzle, a sufficiently enough fine mesh to describe the injection gradients. The goal of the project is to used a meshless Lagrangian method in the primary atomisation. And in the futur realized a coupling between the lagrangian method with the Eulerian formalism inside the combustion chamber.

The first point was to developed a Smoothed Particle Hydrodynamics method (SPH) into OpenFOAM-3.0.1. The SPH method is a Lagrangian method Weakly Compressible. This method is inspired from the code JOSEPHINE developed by J. M. Cherfils et al [1]. The SPH method was validated on dambreak test case.

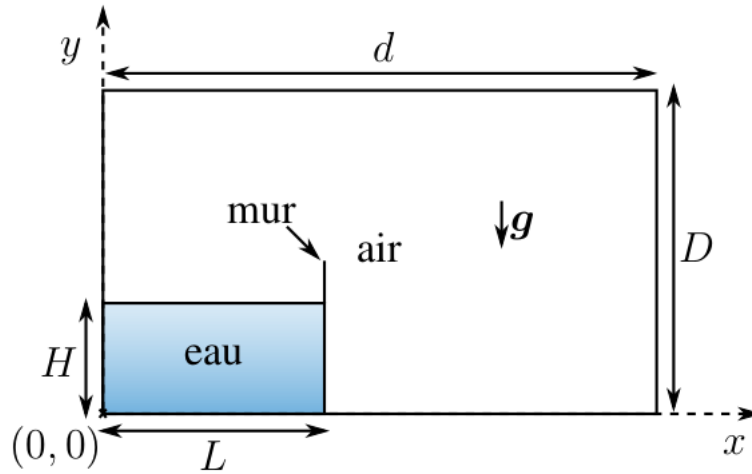
### DamBreak test case

In two dimensions of space, this case, widely studied within the community, plans to let collapse a column of water, initially retained by a vertical wall. Under the effect of gravity, the fluid moves and strikes a wall placed at the opposite end of the physical domain. This test case validates SPH developments realized.

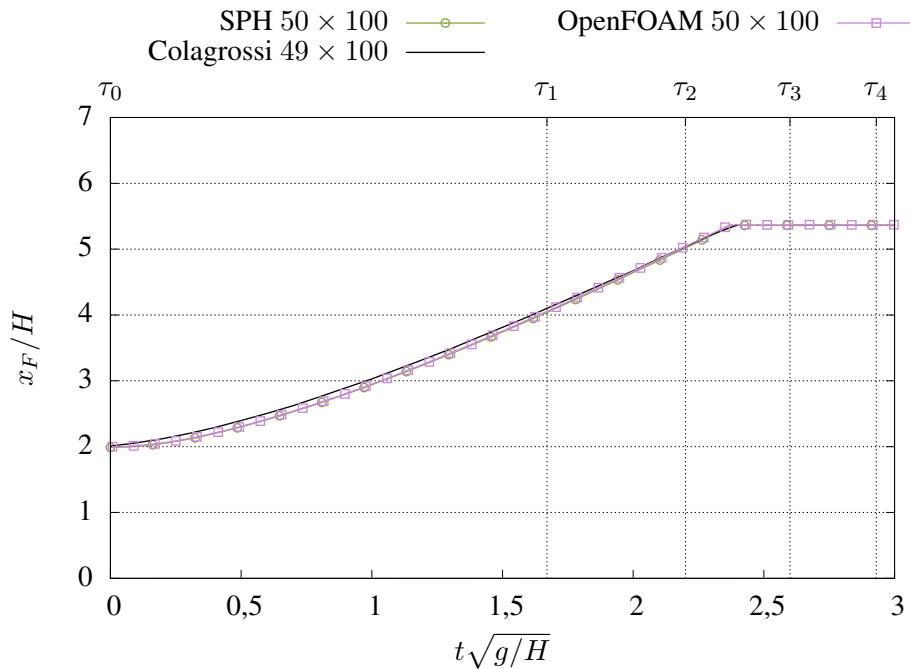
We consider the flow of a fluid mass, initially retained by a vertical wall (Figure 1). Under the effect of gravity  $g = (0, -g)$ , the water column collapses, forming a front that moves rapidly along the ground. The impact of the water front on the opposite wall forms a vertical jet, then this jet will collapse under the effect of gravity. The free surface thus undergoes several successive fragmentations and complex reconnections.

The position of the water front can be approximated by determining, at each step of the simulation, the particle whose component  $x$  of the position is maximum. The error between this approximation and the real position of the front is less than  $h$ . This approach is very simple and fast.

No experimental data are available in the case of  $L/H = 2$ . The results (Figure 2) given by the model show a slight delay compared to the Colagrossi [2] reference solution but is identical to the Cherfils [3] reference solution. More validation was done on pressure impact and water level.



**Figure 1: Initial configuration of collapse of a water column.**  $L$  represents the width of the fluid volume initially retained,  $H$  is its height. A vertical wall is placed at a distance  $d$  from the origin. The computational domain is limited vertically to a distance  $D$  from the origin.



**Figure 2: Evolution of the position of the waterfront.** Comparison the reference SPH solution provided by Colagrossi (black), the reference solution provided by Cherfils (green) and the SPH (purple) model for the same resolution

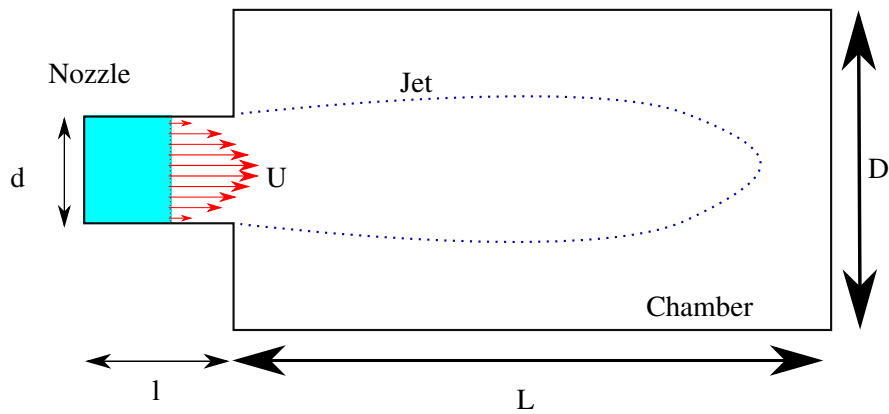
### Jet configuration

Still in two dimensions of space, this case, provides for the injection of water into a combustion chamber. Under the effect of pression, the fluid on the nozzle moves and form a jet on the chamber.

SPH method and finite volume method (FVM) will be use for simulate this test case. A comparison between a FVM simulation and a SPH simulation of jet will be presented on the configuration presented on Figure 3. Angle and length of the jet will be compared together with velocity profiles.

### Acknowledgments

The authors thank all those involved in the organisation of OFW12 and to all the contributors that will enrich this event [4].



**Figure 3: Initial configuration of the jet. Water in the nozzle are initialized with poiseuille velocity and in the chamber water are not moving.**

### References

- [1] J. Cherfils, G. Pinon, and E. Rivoalen, "Josephine: A parallel sph code for free-surface flows," *Computer Physics Communications*, vol. 183, no. 7, pp. 1468–1480, 2012.
- [2] A. Colagrossi and M. Landrini, "Numerical simulation of interfacial flows by smoothed particle hydrodynamics," *Journal of Computational Physics*, vol. 191, no. 2, pp. 448–475, 2003.
- [3] J.-M. Cherfils, "Développements et applications de la méthode sph aux écoulements visqueux à surface libre." Ph.D. dissertation, Université du Havre, 2011.
- [4] OpenCFD, *OpenFOAM: The Open Source CFD Toolbox. User Guide Version 1.4*, OpenCFD Limited. Reading UK, Apr. 2007.