Approach for Coupling Volume-Of-Fluid and Lagrangian Particle Tracking in OpenFOAM

Dr. Martin Heinrich

Institute of Mechanics and Fluid Mechanics, Lampadiusstraße 4, Technical University Bergakademie Freiberg, 09599 Freiberg, Tel: +49 3731 39 3053, Email: martin.heinrich@imfd.tu-freiberg.de

Simulation of atomization processes involves large spatial scales due to the production of droplets with a wide range of diameters. CFD simulation based on a Eulerian approach, e.g. Volume-of-Fluid, are mainly used to predict the primary breakup. However, they require a very high grid resolution to capture the free surface of a large number of smaller droplets. In contrast, the Lagrangian particle tracking method can be used to track a cloud of smaller droplets and estimate their secondary breakup with little computational cost.

This talk will present a method to couple Volume-of-Fluid phase tracking and Lagrangian particle tracking. The basic idea is to track the phase fraction α provided by the VoF method for the primary breakup and larger droplets. Smaller droplets are converted to Lagrangian particles and tracked as such. This significantly reduces the computational cost if coupled with an adoptive mesh algorithm since only VoF regions have to be resolved with a high grid resolution. The presented algorithm works as following:

1. Based on a given phase fraction threshold, all cells containing a fluid phase are marked.
2. Connected regions of fluid are identified and classified based on dimensions, sphericity, volume, and position of the enclosed fluid.
3. If a connected region of fluid fulfils a given criteria of diameter, sphericity, or position, it gets removed and a droplet with equal mass and momentum is inserted at its former position.
4. Droplet movement is tracked in a particle cloud with external forces, such as drag or gravitation. Furthermore, droplet breakup and droplet-droplet collision are taken into account using corresponding models.

This model is implemented in the framework of the Lagrangian library of OpenFOAM v1812 and coupled with the multiphase solver interIsoFoam. A validation study is performed using the benchmark case of Gopala et al. [1] and Sekar et al. [2]. They investigated a liquid jet in cross flow, its trajectory and droplet size distribution at two locations downstream the nozzle. Geometry and boundary conditions were adopted for the CFD simulation with a momentum ratio of 10 and a Weber number of 1500.

Figure 1 shows the cross jet, droplets converted to Lagrangian particles, and the dynamically adopted mesh in the background. Figure 2 compares the experimental and the simulated droplet size distribution at z/d = 30 and z/d = 60 downstream the nozzle. The agreement is quite well with some error close to the wall at x < 2 mm. This could result from a poorly resolved boundary layer around the nozzle.

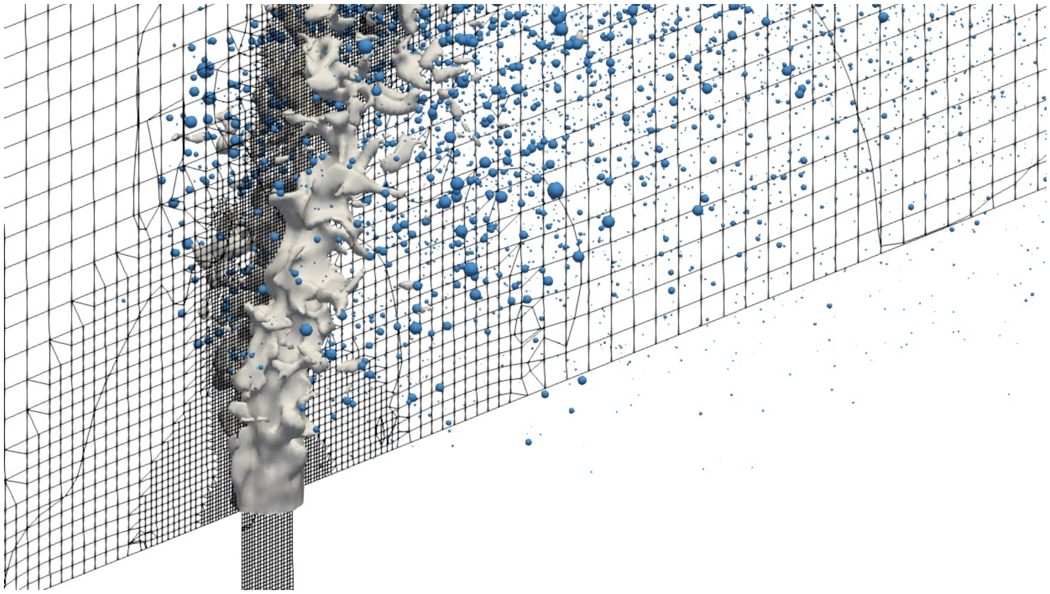

Figure 1: Free surface of jet in cross flow (grey) and Lagrangian particles (blue) with dynamic mesh.

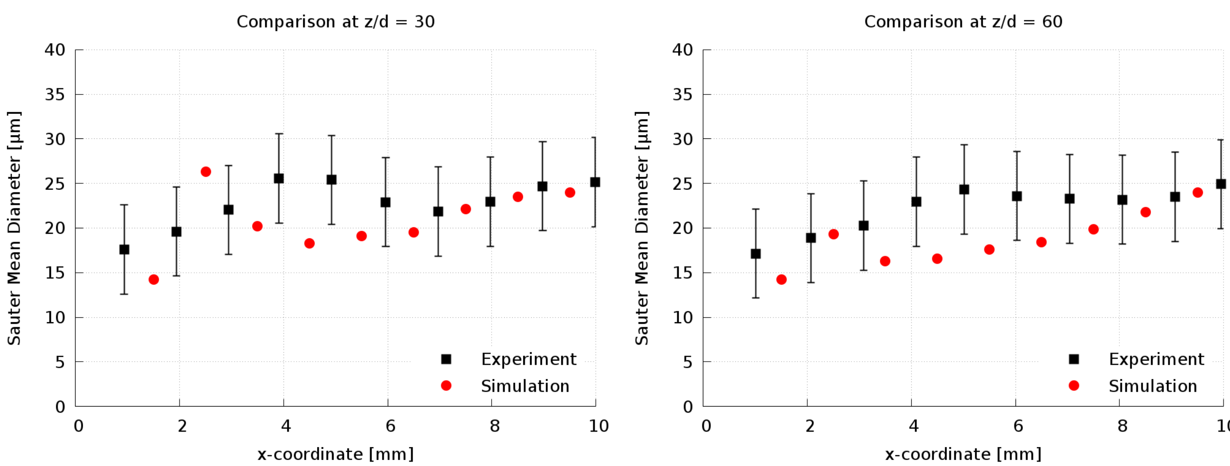

Figure 2: Droplet size distribution at z/d = 30 and z/d = 60 downstream the nozzle exit.

References

[1] Gopala Y., Zhang P., Bibik O., Lubarsky E., and Zinn B.T.: *Liquid Fuel Jet in Crossflow – Trajectory Correlations based on the Column Breakup Point*, 48th AIAA Aerospace Sciences Meeting, 4 – 7 January 2010, Orlando, Florida.

[2] Sekar J., Rao A., Pillutla S., Danis A., and Hsieh S.-Y.: Liquid Jet in Cross Flow Modeling, Proceedings of ASME Turbo Expo 2014: Turbine Technical Conference and Exhibition, 16 – 20 June 2014, Düsseldorf, Germany.