MULTIPHASE VOF SIMULATION OF GASOLINE INJECTORS WITH TOPOLOGICALLY CHANGING GRIDS

FEDERICO PISCAGLIA[1†], ANDREA MONTORFANO[1], FILIPPO GIUSSANI[1], JEROME HÉLIE[2], SHASHIKANT M. AITHAL[3]

[1]*Energy Department, Politecnico di Milano, Milano, ITALY*

[2] *Continental Automotive SAS, Tolouse, FRANCE*

[3] *Argonne National Laboratory, Lemont (IL), USA*

[†] *Corresponding author: federico.piscaglia@polimi.it*

Keywords: *VOF, multi-phase flow, cavitating spray, LES, GDI injector, dynamic mesh handling, topology changes*

In this work, a multi-phase method based on a Volume-of-Fluid (VOF) solver supporting mesh motion based on topological changes and scale-adaptive turbulence modeling is presented. This method is applied to simulate atomizing jets under conditions which approximate those found in high-pressure GDI engines. The objectives are to study the physics of jet atomization with cavitation and upstream nozzle geometry induced disturbances, during the injector opening and closing events.

Simulations of the injector opening event have been performed on a computational grid with 12 M cells at the injector closure. The corresponding cell size is about 1 μm in the nozzle region, that should allow a complete resolution of near-wall turbulence dynamics and flow cavitation. A scale-adaptive filtering technique developed by the authors [1] has been used to perform hybrid URANS/LES modeling of turbulence and to handle the transition between URANS and LES in the computational domain.

The mesh motion strategy used to simulate the needle opening (and closure) event is depicted in Fig. 1; a prescribed vertical motion is set for the boundary `needleHead`, while the injector opening/closure event is simulated by dynamically attaching and detaching the conformal interface represented by the set of faces `detachFaces` in Fig. 1. Because of the very small gap between the injector needle and the injector body at closure ($\mathcal{O}(\delta) = 10\ \mu$m), the mesh handling looks particularly difficult: cell sizes change by orders of magnitude during needle movement and a dynamic cell is required to maintain the cell quality. For this reason, dynamic cell layering is applied in two regions of the injector volume over the face sets named as `topFaces` and `bottomFaces`.

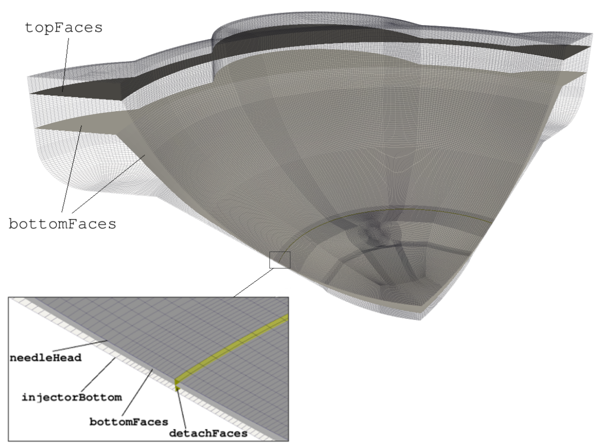

Figure 1: Detail of injector mesh. Layer A/R is performed by face sets "topFaces" and "bottomFace"; dynamic attach/detach is done on "detachFaces" set. "needleHead" is the moving patch.

To run parallel simulations, the FV mesh has to be decomposed into a set of subdomains, each to be assigned to a single core for processing. In simulations involving mesh motion with topological changes, new constraints in domain decomposition arise. In OpenFOAM®, as well as in most of the CFD codes, topological changes cannot occur across inter-processor patches between neighboring subdomains, so some constraints have to be set on the decomposition algorithm:

1. in regions where layer A/R occurs, inter-processor faces cannot be parallel to layer A/R surface;

2. both sides of an attach/detach (or sliding interface) modifier must be included in the same subdomain.

Nevertheless, it is important for the decomposition to remain balanced despite the aforementioned restrictions. In addition, the algorithm should require minimal user intervention, allowing the automation of case setup for large simulation campaigns (i.e. validation/optimization). A new strategy for mesh decomposition, which considers all the aforementioned aspects, has been developed by the authors in OpenFOAM® and applied to the GDI injector. The decomposition strategy outlined above is very flexible, and a very good balancing has been achieved on the present case. The geometrical and/or topological complexity of some configurations prevent the use of a completely automatic algorithm on large meshes. On the other hand, given a precise geometry family, the steps for domain decomposition can easily be automated, thus allowing for a completely automatic case setup for optimization/validation simulations.
Simulations have been carried out on the high performance cluster BLUES at LCRC – Laboratory Computer Resource Center, Argonne National Lab (USA) under project PETSc-FOAM.

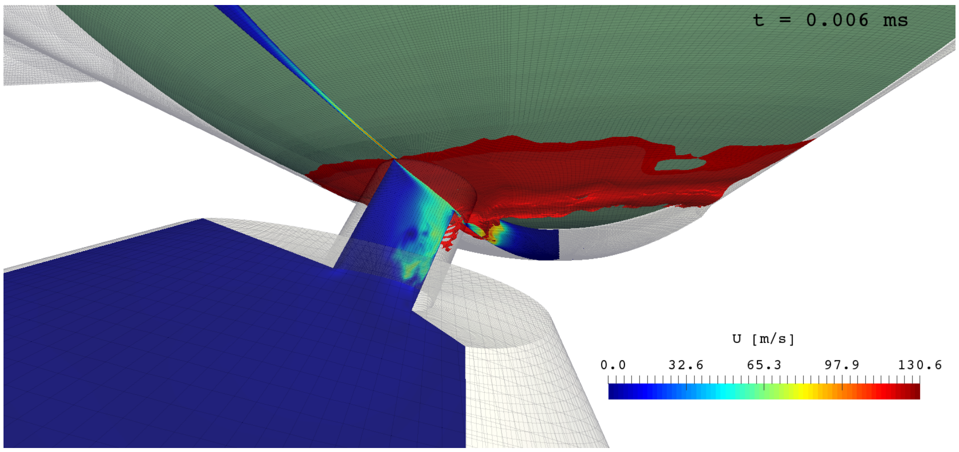

Figure 2: Snapshot of velocity field and fuel/vapor interface at $t = 6$ μs ASOI in the nozzle region of the injector.

References

[1] F. Piscaglia, A. Montorfano, and A. Onorati, "A Scale Adaptive Filtering Technique for Turbulence Modeling of Unsteady Flows in IC Engines," *SAE Int. J. Engines, Paper n. 2015-01-0395*, 2015.

[2] A. Montorfano, F. Piscaglia, and A. Onorati, "An Extension of the Dynamic Mesh Handling with Topological Changes for LES of ICE in OpenFOAM," *SAE paper 2015-01-0384*, 2015, SAE World Congress & Exhibition, Detroit, Michigan (USA).

[3] A. Montorfano, F. Piscaglia, and A. Onorati, "A framework for compressible hybrid RANS/LES simulations in topologically changing grids," in *10th OpenFOAM workshop*, Michigan University. Ann Arbor (MI), USA, July 2015.

[4] F. Piscaglia, A. Montorfano, J. Hélie, and F. X. Demoulin, "Development of a Multi-Phase VOF Dynamic Solver in OpenFOAM: an Application to the Simulation of the Opening and Closure Events in High Pressure GDI Injectors," in *ICMF-2016 9th International Conference on Multiphase Flow, Firenze, Italy*, 2016.

[5] A. Montorfano, F. Piscaglia, M. Schmitt, Y. M. Wright, C. E. Frouzakis, A. G. Tomboulides, K. Boulouchos, and A. Onorati, "Comparison of direct and large eddy simulations of the turbulent flow in a valve/piston assembly," *Flow, Turbulence and Combustion*, vol. 95, no. 2, pp. 461–480, 2015. http://dx.doi.org/10.1007/s10494-015-9620-6