



HAL
open science

A Zonal Grid Method For Liquid Atomization Modeling Using OpenFOAM

Felix Dabonneville, Julien Reveillon, Nicolas Hecht, Grégory Pinon, Elie Rivoalen, F.X. Demoulin

► **To cite this version:**

Felix Dabonneville, Julien Reveillon, Nicolas Hecht, Grégory Pinon, Elie Rivoalen, et al.. A Zonal Grid Method For Liquid Atomization Modeling Using OpenFOAM. 12th OpenFOAM Workshop, Jul 2017, Exeter, United Kingdom. hal-02144968

HAL Id: hal-02144968

<https://normandie-univ.hal.science/hal-02144968v1>

Submitted on 18 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.

A ZONAL GRID METHOD FOR LIQUID ATOMIZATION MODELING USING OPENFOAM

F. DABONNEVILLE^{1*}, J. REVEILLON¹, N. HECHT², G. PINON², E. RIVOALEN³, F.X. DEMOULIN¹

* *Corresponding author : felix.dabonneville@coria.fr*

¹*CORIA Rouen, France,*

²*LOMC Le Havre, France*

³*INSA Rouen, France*

Keywords: *Atomization, Sprays and Injection, Interface, Coupling*

Automotive industry is facing many constraints coming from a strong international competitiveness and a more restrictful legislation about the emission of pollutant gases. Respect of environmental rules goes through technical improvement and control of motor elements. Among them, the combustion chamber plays a crucial role. It is in this part that fuel is consumed and pollutants are generated. Its mastering is thus essential to optimize motors and to minimize emission of pollutants. The physical principles occurring in the combustion chamber can be summarized as follow : the injection system atomize the liquid fuel that evaporates inside the chamber while being mixed with the oxidizer. Once the mixing is sufficiently homogeneous, combustion can takes place. These processes are generally dissociated and studied separately because a complete study of the whole event is currently too complex and hardly achievable.

The key and starting point is the fuel atomization. If it is wrongly predicted, it will notably influence evaporation and mixing inside the chamber. At the present time, it is possible to say that primary atomization modeling is at a preliminary stage of development. Three approaches are generally considered : the industrial approach uses RANS models [1] with a Lagrangian solver "reproducing" the presence of physical particles inside the domain. Despite rough results of this method, this approach has been widely adopted because of its ability to model the whole spray, from the nozzle outlet down to the mixing area inside the combustion chamber, even if the flow is inaccurate at the nozzle outlet. A second approach uses Direct Numerical Simulation (DNS) to capture all the scales of the flow [2, 3]. However, the computational cost is prohibitive and it excludes simulations with a domain larger than ten times the nozzle diameter. The third method avoids the fine mesh constraint by developing subgrid methods both for Eulerian terms [4] and smallest spray droplets [5] by using LES instead of DNS (thus allowing a wider computational domain). However, this approach is limited and computational costs are still too high to be able to treat the spray both in the nozzle area and in the mixing area. Thus, modeling both primary atomization and mixing 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 highly refined mesh to capture the strong injection gradients. For example, an injection velocity of 200 m/s through a 0.1 mm diameter nozzle is usual. Consequently the computational time becomes unrealistic due to the mesh size.

To help alleviate the problem with mesh dependence, adaptive mesh refinement (AMR) adapted to liquid/gas interface is an appropriate option. AMR is a "intelligent meshing strategy" method that refine regions only where it is necessary. This strategy has been utilized already for the modeling of diesel sprays [6, 7, 8, 9, 10] using jet breakup models. It shows that the spray structure and liquid penetration are accurately predicted with a reasonable computational cost but by using spray breakup models. Furthermore, the type of AMR that is used employs a single refined grid with a global time step. Thus, the time step is penalized by the refinement process. To gain more computer resources, refined regions should be independent blocks with a specific time step that does not impact the global time step. This type of AMR is called AMR with subcycling in time. Refinement is performed in time as well as in space so that the ratio of the time step to the grid spacing is kept constant. This approach is based on that of Berger and Olinger [11]. It has been recently adapted by Martin et al. [12, 13] and Almgren et al. [14] for incompressible Euler and Navier-Stokes equations and applied to primary atomization by injection by Lebas et al. [15] using DNS.

Similarly to what is done in AMR, the use of zonal grids or grid nesting allows to deal with spatially varying resolutions, at the difference that the region where refinement is necessary is known in preprocessing, there is no cell tagging algorithm based on an precise criteria as in AMR. Zonal grid method has been used in meteorological applications [16, 17, 18, 19] for reducing the subgrid scale motions in LES simulations near boundaries. However, it has not been applied in our knowledge to liquid atomization.

We decided to use this method. Indeed, as for the zonal grid, we know where the region of primary atomization is located and thus where to place a zonal refined region. In perspective, a refinement criteria could be developed to allow

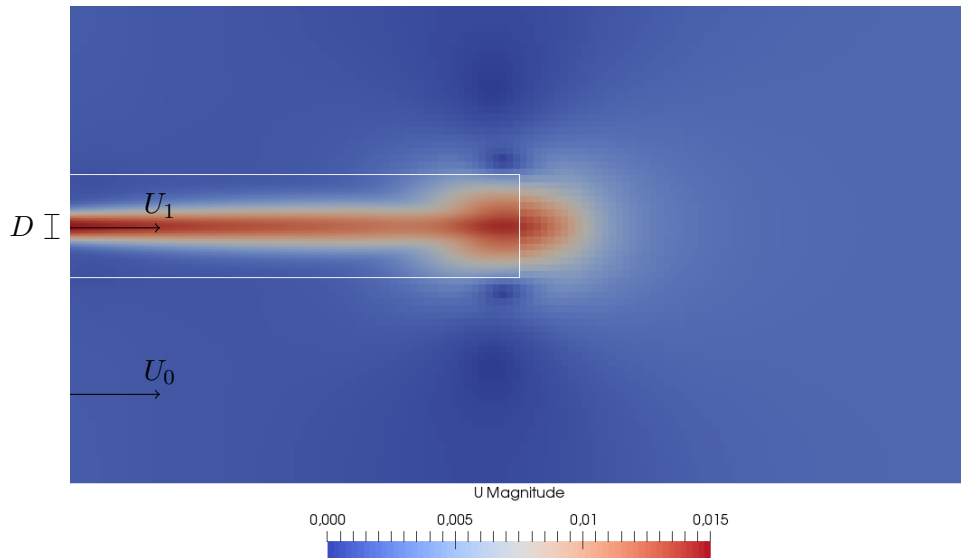


Figure 1: Velocity field of a jet in a coflow by using our Zonal Grid method, at $t(U_1/D) = 17$. $U_1/U_0 = 1.5$. Zonal Grid is outlined in white

adapting automatically the location of the interface between Zonal Domain (ZD) and Global Domain (GD). Another advantage of such method, in addition with refinement, is the possibility to use two different numerical methods in each region. This is similar to the Chimera method used in aerospace [20, 21], where Navier-Stokes equations are solved close to the walls while Euler equations are solved in the rest of the domain. In this optic, we wish to use a VOF method with Interface Capturing Method (ICM) in the zonal refined region and a VOF method with diffuse interface model in the global "coarse" mesh. Diffuse interface models have been developed where the interface is considered as a mixing zone such that the two phases coexist at the same macroscopic position with an occupied portion of volume defined by the liquid volume fraction.

As a conclusion, we shall retain two objectives of the zonal grid method we develop : subcycling in time between the two finite volume regions and the possibility to use two different VOF solvers : one with ICM and the other with diffuse interface model, without impacting the correct physical behavior of the flow. The question that arises is : how to transfer information from one domain to another ?

The plan of our presentation is as follow : i) Description of our new zonal grid method for a single phase incompressible laminar solver in OpenFOAM[®] and presentation of validation test case results. A visual of what we can get with our method is shown in figure 1. It shows a laminar single phase in a coflow. The refinement ratio between the Global grid and the Zonal Grid is 5. ii) Description of the extension of this method to two-phase turbulent solvers with diffuse interface model and presentation of validation test case results. In OpenFOAM[®], `twoLiquidMixingFoam` solver is the dedicated solver. iii) Description of a further extension to two phase turbulent solver with ICM and presentation of the first results. `interFoam` solver is the dedicated solver.

Acknowledgments

The authors acknowledge the financial support of the french Agence Nationale de la Recherche (ANR), through the program Investissements dAvenir(ANR-10-LABX-09-01) , LabEx EMC3.

References

- [1] J. K. Dukowicz, "A particle-fluid numerical model for liquid sprays," *Journal of Computational Physics*, vol. 35, no. 2, pp. 229–253, 1980.
- [2] T. Ménard, S. Tanguy, and A. Berlemont, "Coupling level set/vof/ghost fluid methods: Validation and application to 3d simulation of the primary break-up of a liquid jet," *International Journal of Multiphase Flow*, vol. 33, no. 5, pp. 510–524, 2007.
- [3] O. Desjardins, G. Blanquart, G. Balarac, and H. Pitsch, "High order conservative finite difference scheme for variable density low mach number turbulent flows," *Journal of Computational Physics*, vol. 227, no. 15, pp. 7125–7159, 2008.
- [4] J. Chesnel, T. Menard, J. Reveillon, and F.-X. Demoulin, "Subgrid analysis of liquid jet atomization," *Atomization and Sprays*, vol. 21, no. 1, 2011.

- [5] M. Herrmann, "A parallel eulerian interface tracking/lagrangian point particle multi-scale coupling procedure," *Journal of Computational Physics*, vol. 229, no. 3, pp. 745–759, 2010.
- [6] S. Tonini, M. Gavaises, and A. Theodorakakos, "Modelling of high-pressure dense diesel sprays with adaptive local grid refinement," *International Journal of Heat and Fluid Flow*, vol. 29, no. 2, pp. 427–448, 2008.
- [7] H. M. Ismail, H. K. Ng, S. Gan, and T. Lucchini, "Application of adaptive local mesh refinement (almr) approach for the modeling of reacting biodiesel fuel spray using openfoam," SAE Technical Paper, Tech. Rep., 2014.
- [8] T. Lucchini, G. D'Errico, and D. Ettore, "Numerical investigation of the spray–mesh–turbulence interactions for high-pressure, evaporating sprays at engine conditions," *International Journal of Heat and Fluid Flow*, vol. 32, no. 1, pp. 285–297, 2011.
- [9] R. Kolakaluri, Y. Li, and S.-C. Kong, "A unified spray model for engine spray simulation using dynamic mesh refinement," *International Journal of Multiphase Flow*, vol. 36, no. 11, pp. 858–869, 2010.
- [10] Q. Xue and S.-C. Kong, "Development of adaptive mesh refinement scheme for engine spray simulations," *Computers & Fluids*, vol. 38, no. 4, pp. 939–949, 2009.
- [11] M. J. Berger, "On conservation at grid interfaces," *SIAM Journal on Numerical Analysis*, vol. 24, no. 5, pp. 967–984, 1987.
- [12] D. F. Martin and P. Colella, "A cell-centered adaptive projection method for the incompressible euler equations," *Journal of computational Physics*, vol. 163, no. 2, pp. 271–312, 2000.
- [13] D. F. Martin, P. Colella, and D. Graves, "A cell-centered adaptive projection method for the incompressible navier–stokes equations in three dimensions," *Journal of Computational Physics*, vol. 227, no. 3, pp. 1863–1886, 2008.
- [14] A. S. Almgren, J. B. Bell, P. Colella, L. H. Howell, and M. L. Welcome, "A conservative adaptive projection method for the variable density incompressible navier–stokes equations," *Journal of computational Physics*, vol. 142, no. 1, pp. 1–46, 1998.
- [15] R. Lebas, T. Menard, P.-A. Beau, A. Berlemont, and F.-X. Demoulin, "Numerical simulation of primary break-up and atomization: Dns and modelling study," *International Journal of Multiphase Flow*, vol. 35, no. 3, pp. 247–260, 2009.
- [16] G. W. Ley and R. L. Elsberry, "Forecasts of typhoon irma using a nestedgrid model," *Monthly Weather Review*, vol. 104, no. 9, pp. 1154–1161, 1976.
- [17] Y. Kurihara, G. J. Tripoli, and M. A. Bender, "Design of a movable nested-mesh primitive equation model," *Monthly Weather Review*, vol. 107, no. 3, pp. 239–249, 1979.
- [18] P. P. Sullivan, J. C. McWilliams, and C.-H. Moeng, "A grid nesting method for large-eddy simulation of planetary boundary-layer flows," *Boundary-Layer Meteorology*, vol. 80, no. 1, pp. 167–202, 1996. [Online]. Available: <http://dx.doi.org/10.1007/BF00119016>
- [19] M. Manhart, "A zonal grid algorithm for {DNS} of turbulent boundary layers," *Computers & fluids*, vol. 33, no. 3, pp. 435 – 461, 2004. [Online]. Available: <http://www.sciencedirect.com/science/article/pii/S0045793003000616>
- [20] P. Buningt, "A 3-d chimera grid embedding technique," 1985.
- [21] J. Benek, J. Steger, and F. Dougherty, "A flexible grid embedding technique," *AIAA paper*, pp. 85–1523, 1985.