CFD Modelling of Fuel-Air mixture formation in a GDI engine using OpenFOAM

Andrea Pati, Federica Ferraro, Christian Hasse

TU Darmstadt – STFS, Otto-Berndt-Str.2, D-64287 Darmstadt
+49 6151 16-24147, pati@stfs.tu-darmstadt.de

Introduction

In the automotive sector gasoline engines represent a large market share, so they are the main responsible for the pollution and greenhouse gas emissions due to road transport. This prioritizes the necessity to develop engines according to the most strict rules like EURO-6 and to increase the efficiency of the engine reducing CO, $NO_{x,}$ and particulate matter emissions.

In this context gasoline, direct injection (GDI) represents the state of the art for spark-ignition engines, due to its ability to generate non-uniform mixture formation [1]. These engines are characterized by global in-cylinder flow structure, like tumble and swirl, used to redirect the injected fuel near the spark-plug and to increase the turbulence of the in-cylinder charge. This makes it possible to ignite a lean mixture and to increase the flame speed.

Computer fluid dynamics (CFD) is a powerful tool to investigate gasoline engines because of its capability to discretize complex 3D moving geometry. Moreover, in combination with multidimensional models, also highly turbulent, multiphase and multicomponent flows can be simulated. GDI engine simulation is a challenge requiring all these capabilities.

In this work the numerical framework for simulating GDI engines developed OpenFOAM is first presented. Subsequently, results obtained for the Darmstadt Optical Engine [2] are discussed.

Numerical method

In this work, a full set of models implemented in OpenFOAM framework is used to simulate the Darmstadt Optical Engine (DOE). It is a 4-valve pent-roof engine with full optical access through the liner and the piston, for further information the reader is invited to refer to [2].

The injector coupled to DOE is the Spray G, an 8-hole injector manufactured by Delphi and used in the context of the Engine Combustion Network, providing a common platform for industries and academia for the validation and the study of gasoline injection. The operating point used is the G3, characterized by injection in ambient pressure, typical of early injection mode in the GDI engine.

More in detail, a Reynolds-averaged Navier-Stokes (RANS) CFD approach was used to simulate the engine in motored condition. The comparison of the results against experimental results was used to validate the methodology. Then a fully coupled Eulerian-Lagrangian simulation was used to study the spray evolution and the subsequent fuel-air mixture formation. The effect of spray impingement was considered including a Finite-Area Method to predict the evolution of the wall film.

A set of solvers, libraries, and Lagrangian sub-models was implemented in a comprehensive library named TFMotion [3] [4], heavily based on the well-validated LibICE code, developed by Politecnico di Milano [5] [6]. Aside from the C++ code, a set of python libraries and utilities was developed to script the meshing and the calculation process. This allows decoupling the mesh generation from the calculation, to parallelize the meshing phase of different engine phase, easily switching between different models during the calculation, targeting a *hit-and-miss* strategy. All calculations are performed in OpenFOAM® 2.4.x framework, except for mesh generation, achieved with ESI OpenFOAM® v1812.

The approach for the mesh management in engines is the *key-grid* approach: an initial hexahedral mesh for a given crank angle is generated, then the cells of the mesh are moved according to prescribed boundary motion using a Laplacian equation. When the mesh quality is no more satisfied the motion is stopped and a new mesh is generated for the latest valid valve and piston position. This ensures a high quality of the grid before the calculation. The entire methodology was validated in previous work [6] [3]. When the calculation is started, the results at the end of a certain mesh are mapped on the next one, proceeding with the calculation, a set of python scripts are used to automatize the job.

The calculations were performed using the standard k-ε turbulence model including the Pope correction for the transport equation of the dissipation rate ε, to improve the prediction of the spray morphology and the air entrainment [7] [8]. The limitedLinear scheme was used in every calculation, except the convective transport of the velocity where linearUpwind was used, to guarantee consistency between different cases. PISO algorithm was used without any field under-relaxation. Double level Courant number was developed in this context. This method, unlike the classic variable timestep method, not only reduces the timestep when a threshold Courant is reached but also increase again the timestep only if Courant number drop below a certain threshold; this reduces significantly the timestep changing during the simulation, reducing the numerical diffusion and increasing the stability of the solver.

For the spray, a Lagrangian methodology was employed. A set of Lagrangian parcels, that statistically represent the entire spray are introduced in the system and undergo into drag, breakup, and evaporation. Parcels are injected in the domain according to the *blob*-injection mechanism, with a velocity coefficient $C_v = 0.73$. The model used for the breakup is based on the classic KHRT approach [9]. Kelvin-Helmholtz *wave* model was used for the primary breakup, while Rayleigh-Taylor *catastrophic* model was used for the secondary breakup, beyond the core length region, identified as $L_c = CD\sqrt{\rho_l/\rho_g}$, where C is a parameter constant influenced by the injector geometry, D is the parcel diameter ρ_l and ρ_g are the density of liquid and gas respectively. A mass-based approach was used for modeling the liquid evaporation [10], while heat transfer was modeled through the Ranz-Marshall correlation.

The wall film was considered of this work, it was treated as a thin film discretized by the Finite Area Method. It assumes that the film is thin enough to neglect the gradient of the properties perpendicular to the surface, this makes it possible to discretize the film on a two-dimensional surface developed in the space. The liquid film is fully coupled with the Eulerian fields and with the Lagrangian spray, according to the Bai-Gosman impingement model [11].

In future work, the flexible TFMotion numerical framework will be extended to account for combustion in GDI engines. A tabulated flamelet progress variable approach will be integrated to describe premixed flame combustion, characterizing spark-ignition engines. This approach decouples the complex chemistry from the turbulent flow. The Artificial Thickened Flame (ATF) approach will be used to account for the turbulence-chemistry interaction. In the flamelet model, laminar flames are calculated in a pre-processing step and stored in the flamelet database. The flamelet look-up table is then accessed as a function of a reduced set of variables, e.g. mixture fraction and a progress variable, providing high computational efficiency [12]. Pollutant emissions as well as soot particle distribution can be accounted for with proper submodels, that have been developed at STFS (TU Darmstadt) in the OpenFOAM framework. In particular, to characterize the soot particle formation and growth, the Quadrature-based Method of Moments (QMOM), has been developed [13]. The method reconstructs the particle size distribution by solving only for its low-order moments. A numerically robust formulation, the split- based EQMOM, provides the ability to account for soot particle oxidation, which is one of the challenging processes in soot modeling [14] [15].

Finally, the TFMotion numerical framework, developed in OpenFOAM in a modular and extendable structure, includes the main physical models, from spray injection, evaporation, combustion, and pollutant emissions, needed for predictive numerical simulations of real engine configurations.

Results

The numerical setup for the simulation was investigated and validated in this section. A full-cycle simulation of a complete engine cycle was performed. Bore and stroke are equal to 86 mm, engine speed equal to 800 rpm. The adopted valve timing is shown in Table 1, with a minimum valve lift imposed to 0.5mm. It is possible to observe that intake and exhaust ducts are attached and detached when the corresponding valve lift in the real engine is equal to half of the minimum lift (0.25 mm). This methodology was used to consider the mass/momentum exchanged when the valves are lifted to a value below the minimum valve lift.

Table 1 Valve timing in the Darmstadt Engine

IVO@0.25 mm	IVC@0.25 mm	EVO@0.25 mm	EVC@0.25 mm
347° aTDC	132 °bTDC	127 °aTDC	351 °bTDC

Table 2 Operating condition in the Darmstadt Engine

		Regime [rpm]	
		800	1500
Intake pressure [bar]	0.95	A	C
	0.4	B	D

The fluid dynamics into the duct was considering using the measured time-dependent pressure at intake as a boundary condition. Wall temperature is set up to 333.15 K according to experimental measure. Working fluid is air with a prescribed intake temperature of 307.85K according to experimental data. Different operating conditions are available and shown in Table 2.

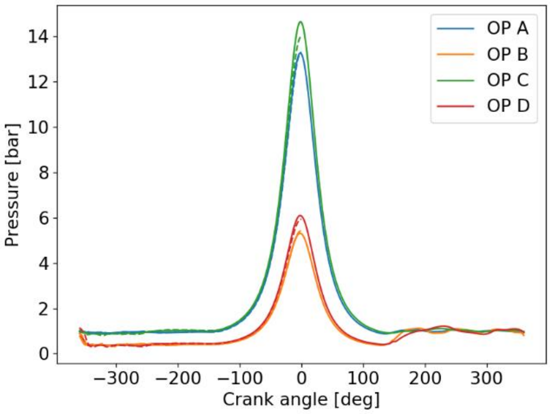

Figure 1 Comparison between experimental and numerical cylinder pressure of the Darmstadt engine, dashed line represents the simulation results, solid line experimental data

To validate the methodology the in-cylinder pressure was compared with experimental data, as shown in Figure 1. It is possible to observe a perfect match between experiments and numerical results. This is evidence, that both the mass trapped and the thermal exchange in the cylinder was properly captured.

As a second step, the velocities field on the tumble plane was compared against experimental measurement for operating point A at a crank angle equal to 90° aTDCc.

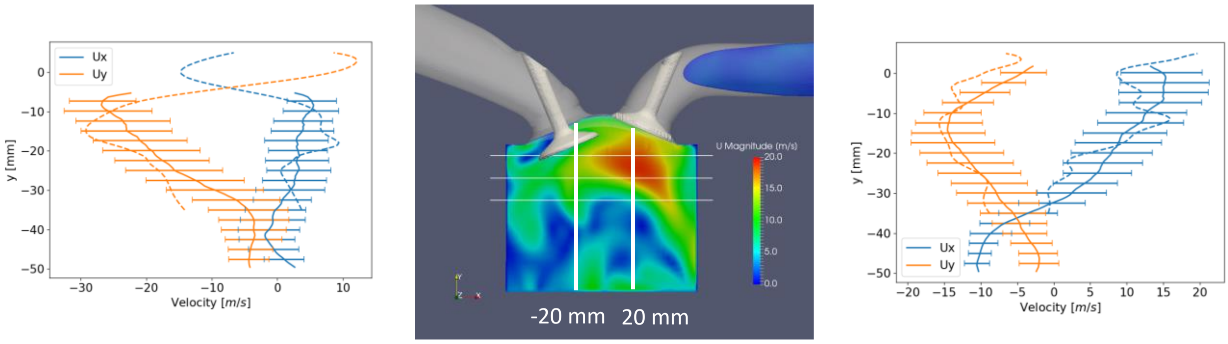

Figure 2 Comparison between computed and experimental velocities profile on the tumble plane on two vertical lines for the Darmstadt Engine under the operating point A at 90° aTDCc

In Figure 2 the velocities along vertical lines are compared. It is possible to observe a good match to the experimental measurements, indicating the main tumble flow is properly reproduced. A similar comparison (not shown here) was performed on horizontal lines, yielding to the same conclusion.

As the next step, the effect of spray was studied on the flow evolution and fuel-air mixture distribution. Within this context, the nominal ECN Spray G3 was introduced into the Darmstadt engine with an injector axis of 8° towards intake valves [16]. The electric start of injection was imposed at 270 °bTDC. The morphology of the spray 3.5 °aESOI is visible in Figure 3. The impact of the spray on the valves can be observed, as well as the liquid core. Beyond the liquid core, the catastrophic breakup reduces immediately the diameter of the spray. The wall impingement is visible with more detail in Figure 4 at 10° aESOI. Toward the exhaust valve, the spray impingement develops a wall film on the liner, while the plume oriented toward the intake valve impacts the piston developing a large fuel film on it.

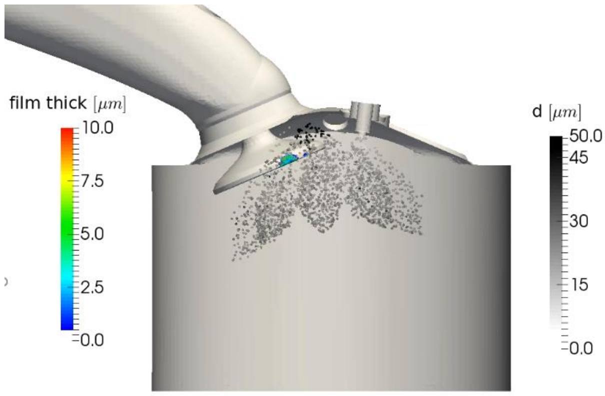

Figure 3 Spray overview, operating point A, 3.5 °aESOI

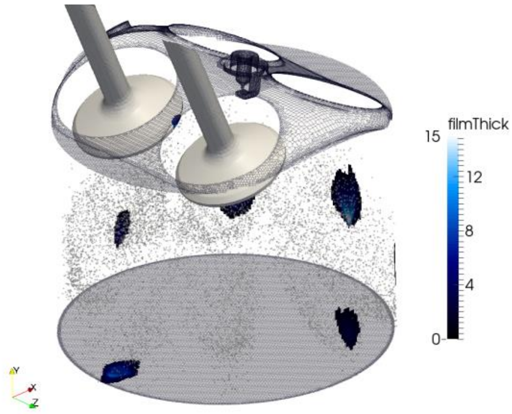

Figure 4 Wall film overview 10° aESOI

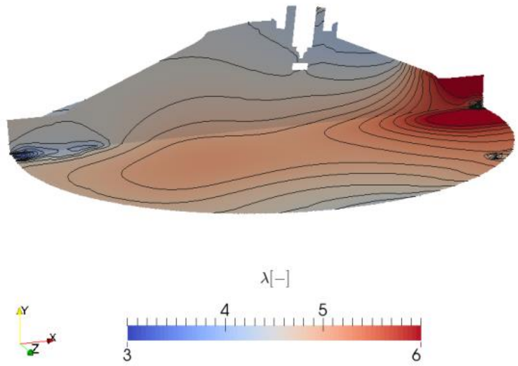

Figure 5 Air/fuel distribution at 20° bTDCf

The impact of the spray on the fuel-air mixture formation can be observed in Figure 5. It can be seen that the amount of fuel injected by the ECN spray G injector is too low for the optical engine.

The formation of a stratified mixture can be identified around the spark plug. The effect of the evaporation of the wall film can be also be observed. The large wall film on the piston evaporates enriching the mixture between the piston and intake valve. On the other side, the wall films on the liner get into the crevice (not shown in the picture) during the compression phase leading to a really small and concentrate rich mixture surrounded by a large volume of lean mixture.

Conclusion

In this work, a set of libraries and utilities developed in the OpenFOAM® framework was introduced specifically for simulation of the internal combustion engines. The key-grid approach was used in this work and validated on an optically accessible engine comparing the pressure in the cylinder and the velocities on the tumble plane. Both comparisons show a good match with experimental data, meaning that trapped mass, thermal exchange, and tumble motion are properly captured.

As a second point, the effect of spray G injection is shown on wall film formation and air-fuel distribution. The spray impacts largely on the wall forming a consistent amount of fuel film, on liner, piston, and valves. The evaporation of fuel film largely impacts the fuel distribution, enriching the mixture between piston and intake valve and leading a strong stratification. This distribution can impact the pollutant formation, e.g. the rich region near the wall can promote the soot formation.

Finally, it has been proven that OpenFOAM® can be used as a platform to predict the fuel-air mixture in GDI engines and, as a consequence, as a design and optimization tool.

Bibliography

[1] P. Koch, M. G. Löffler, M. Wensing and A. Leipertz, “Study of the mixture formation processes inside a modern direct-injection gasoline engine,” *International Journal of Engine Research,* vol. 11, pp. 455-471, 2010.

[2] E. Baum, B. Peterson, B. Böhm and A. Dreizler, “On The Validation of LES Applied to Internal Combustion Engine Flows: Part 1: Comprehensive Experimental Database,” *Flow, Turbulence and Combustion,* vol. 92, p. 269–297, 2014.

[3] A. Pati, D. Paredi, T. Lucchini and C. Hasse, “CFD Modeling of Gas-Fuel Interaction and Mixture Formation in a Gasoline Direct-Injection Engine Coupled With the ECN Spray G Injector,” in *SAE Technical Paper*, 2020.

[4] A. Pati and C. Hasse, “Simulation of fuel/air mixture formation in optical Gasoline-Direct Injection engine,” in *Fourth Two-day Meet. Intern. Combust. Engine Simulations Using OpenFOAM® Technol.*, 2020.

[5] T. Lucchini, G. D'Errico, F. Brusiani, G. M. Bianchi, Z. Tukovic and H. Jasak, “Multi-dimensional modeling of the air/fuel mixture formation process in a PFI engine for motorcycle applications,” Vols. SAE paper, 2009-24-0015, 2009.

[6] T. Lucchini, G. D'Errico and M. Fiocco, “Multi-dimensional modeling of gas exchange and fuel-air mixing processes in a direct-injection, gas fueled engine,” Vols. SAE paper, 2011-24-0036, 2011.

[7] S. B. Pope, “An explanation of the turbulent round-jet/plane-jet anomaly,” *AIAA Journal,* vol. 16, pp. 279-281, 2 1978.

[8] D. Paredi, T. Lucchini, G. D'Errico, A. Onorati, A. Montanaro, L. Allocca and R. Ianniello, “Combined Experimental and Numerical Investigation of the ECN Spray G under Different Engine-Like Conditions,” *SAE Technical Paper,* Vols. 2018-01-0281, 2018.

[9] R. D. Reitz, “Modeling Atomization Processes In High Pressure Vaporizing Sprays,” *Atomization and Spray Technology,* vol. 3, pp. 309-337, 1987.

[10] C. Baumgarten, Mixture formation in internal combustion engines, Springer Science & Business Media, 2006.

[11] T. Lucchini, G. D'Errico, A. Onorati, G.Bonandrini, L. Venturoli and R. D. Gioia, “Development and application of a computational fluid dynamics methodology to predict fuel–air mixing and sources of soot formation in gasoline direct injection engines,” *International Journal of Engine Research,* vol. 15, pp. 581-596, 2014.

[12] S. Weise, S. Popp, D. Messig and C. Hasse, “A Computationally Efficient Implementation of Tabulated Combustion Chemistry based on Polynomials and Automatic Source Code Generation,” *Flow, Turbul. Combust.,* vol. 100, p. 119–146, 2018.

[13] S. Salenbauch, M. Sirignano, D. L. Marchisio, M. Pollack, A. D. Anna and C. Hasse, “Detailed particle nucleation modeling in a sooting ethylene flame using a Conditional Quadrature Method of Moments (CQMOM),” *Proc. Combust. Inst.,* vol. 36, p. 1–9, 2016.

[14] S. Salenbauch, C. Hasse, M. Vanni and D. L. Marchisio, “A numerically robust method of moments with number density function reconstruction and its application to soot formation, growth and oxidation,” *J. Aerosol Sci.,* vol. 128, p. 34–49, 2019.

[15] F. Ferraro, S. Gierth and C. Hasse, “Advanced soot models for engine applications,” in *Fourth Two-day Meet. Intern. Combust. Engine Simulations Using OpenFOAM® Technol.*, 2020.

[16] C. Geschwindner, P. Kranz, C. Welch, M. Schmidt, B. Böhm, S. A. Kaiser and J. De la Morena, “Analysis of the interaction of Spray G and in-cylinder flow in two optical engines for late gasoline direct injection,” *International Journal of Engine Research,* p. 1468087419881535, 10 2019.

[17] S. Salenbauch, A. Cuoci, A. Frassoldati, C. Saggese, T. Faravelli and C. Hasse, “Modeling soot formation in premixed flames using an Extended Conditional Quadrature Method of Moments,” *Combustion and Flame,* vol. 162, pp. 2529-2543, 2015.