A comprehensive Two-Fluid Model for Cavitation and Primary Atomization Modelling of liquid jets - *Application to a large marine Diesel injector*

Chawki Habchi[1,2], Julien Bohbot[1,2], Andreas Schmid[3] and Kai Herrmann[4]

[1] IFP Energies nouvelles,1 et 4 avenue de Bois-Préau, 92852 Rueil-Malmaison, France
[2] Institut Carnot IFPEN Transports Energie
[3] Winterthur Gas & Diesel Ltd., CH-8401 Winterthur, Switzerland
[4] Paul Scherrer Institute, CH-5232 Villigen PSI, Switzerland

E-mail: Chawki.HABCHI@ifpen

Abstract. In this paper, a comprehensive two-fluid model is suggested in order to compute the in-nozzle cavitating flow and the primary atomization of liquid jets, simultaneously. This model has been applied to the computation of a typical large marine Diesel injector. The numerical results have shown a strong correlation between the in-nozzle cavitating flow and the ensuing spray orientation and atomization. Indeed, the results have confirmed the existence of an off-axis liquid core. This asymmetry is likely to be at the origin of the spray deviation observed experimentally. In addition, the primary atomization begins very close to the orifice exit as in the experiments, and the smallest droplets are generated due to cavitation pocket shape oscillations located at the same side, inside the orifice.

1. Introduction

Efficient transportation engines are required worldwide in order to decrease greenhouse emissions due to mobility not only by automobiles and airplanes, but also by ships. High-fidelity experimental data together with reliable two-phase flow simulation tools can help us better understand the physics governing the pollutants formation and consequently design more efficient engines in the future. Pollutant features and amounts from engines combustion are known to be influenced by in-nozzle flow and spray characteristics. But, currently, Computation Fluid Dynamic's (CFD) software cannot properly predict this key behaviour because of two main reasons: (1) internal injector scales can differ from internal combustion scales by three orders of magnitude; (2) a lack of understanding of the different physical phenomena (cavitation, primary atomization ...) and therefore their modelling. Some recent studies have focused on coupling the nozzle flow and ensuing spray in a "static" fashion [1] [2] [3]. In a static coupling approach the nozzle flow and spray simulations are performed separately. The nozzle flow simulations provide the boundary conditions at the nozzle exit to initialize a classical Eulerian-Lagrangian spray calculation. This kind of adhoc coupling at the nozzle exit boundary is not predictive in nature. Also, not all flow features can be captured in a static coupling approach since the spray results do not feedback to affect the nozzle flow development. In addition, transient injection periods (such as the opening and closing periods) can hardly be modelled using such static approach.

[1] Corresponding author (Dr C. Habchi, IFPEN).

Besides, high spatial accuracy in the spray can be ensured if the simulation of the dense region is performed using an Eulerian-Eulerian (EE) approach rather than the classical Eulerian-Lagrangian (EL) framework. Indeed, in the near nozzle region, the proportion of liquid phase is very high and there is experimental evidence that there are very few discrete droplets in this region. The liquid phase may also comprise of voids or bubbles due to absorption of the ambient gas and cavitation from inside the nozzle. In this case, the liquid will be the continuous phase and void fractions would be the discrete phase. For these reasons, a comprehensive two-fluid model has been recently developed at IFPEN. In this work, this model is applied to the numerical simulation of a typical large marine injector, for which experimental results are available [4] [5] [6]. The results of these high fidelity simulations are presented. In the following section, the main sub-models will be described briefly since they are available elsewhere [7] [8]. Next, the computational configuration of the nozzle will be defined. Then, the results discussion section presents the main findings of the study, before the conclusions.

2. Description of the Two-fluid model

In this work, a comprehensive highly compressible two-fluid multi-species model is used. This model has been recently developed in order to compute in-nozzle cavitating flow and the primary atomization of actual liquid jets, simultaneously. It involves an equation for the transport of the liquid volume fraction in addition to two different sets of partial differential equations for the gas and the liquid phase. The multicomponent gas species phase is governed by an ideal gas equation of state (EOS) while the stiffened gas EOS is specified to the single-component liquid phase as a first attempt to compute highly compressible flows using a thermodynamic consistent model. Instead of the Reynolds Averaged Navier-Stokes (RANS) formulation adopted in [7] and [8], a standard Smagorinsky model is adopted for sub-grid scale turbulence modelling. Besides, the Gibbs free energy relaxation model (*GERM*) cavitation model described in detail in [7] is used. Unlike previous models, the *GERM* model may simulate two different cavitation regimes. The first so-called “gaseous cavitation” may appear in regions in which the static pressure is close to but above the liquid saturation pressure. In this case, not only the disequilibrium of pressure but also the disequilibrium of temperature between the liquid and the gas are responsible for the growth or collapse of the bubbles without the help of any phase change. The second cavitation regime may happen when the static pressure goes below the liquid saturation pressure. In this case, the liquid becomes superheated and leads to a “vaporous cavitation” regime. In addition, the *GERM* model is able to reproduce the appearance and collapse of cavitation without using adjustable parameters. Present simulations also adopt the *TwoSD* atomization model developed recently by Devassy et al. [8]. This model uses a two-surface density approach within the framework of the same Eulerian two-phase approach of the *GERM* cavitation model. The method followed distinguishes the primary and secondary atomization processes using separate equations of surface density: One for the liquid core and the other for the dispersed phase (the spray droplets). As the phenomenon of primary atomization is different from droplet breakup, a two surface density model may facilitate the modelling stages for both the droplets and liquid core. Thereby, by separating the liquid phase into two sub-phases (droplets and liquid core); it is possible to deal with the atomization and breakup phenomena more precisely. For instance, with this approach the droplets generated due to the primary atomization and secondary breakup can be filtered and studied separately for determining the droplets size PDFs and can later be vaporized using appropriate evaporation models.

3. Computational configuration

In this study, a simplified generic single-hole nozzle is used (Figure 1) [6]. This specimen has been designed in order to mimic the injection process from a typical five holes nozzle of large marine Diesel engines [4] [5]. It consisted of an elongated tip with a single injection hole. The direction of the fuel flow, the location of the spray orifice and the resulting main injection direction are indicated on the photograph of injection configuration inside the spray combustion chamber (SCC) on left of Figure 1. Two cross-sections (in the middle and on the right side of Figure 1) show the details of the nozzle

configuration in lateral (A-A) and axial (B-B) direction of the nozzle bore. Important to mention here is the role of the drilled hole with diameter **D**, which bypasses the flow of four additional orifices – as they would exist in a serial injector – and has hence four times the area of the spray orifice with diameter **d**. This second hole leads the bypassed fuel amount into the discharge fuel system as indicated in Figure 1 (left). More details may be found in [9].

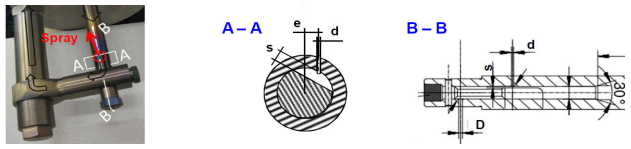


Figure 1. Generic injector configuration. Photograph inside the SCC (left). Cross-section A-A through the injection orifice (d=0.75 mm and e=1.5 mm for the eccentric configuration) (middle). Cross-section B-B through the injector bore with s=0.75 mm and D=1.3 mm (right).

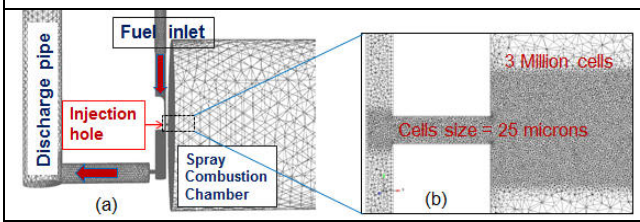
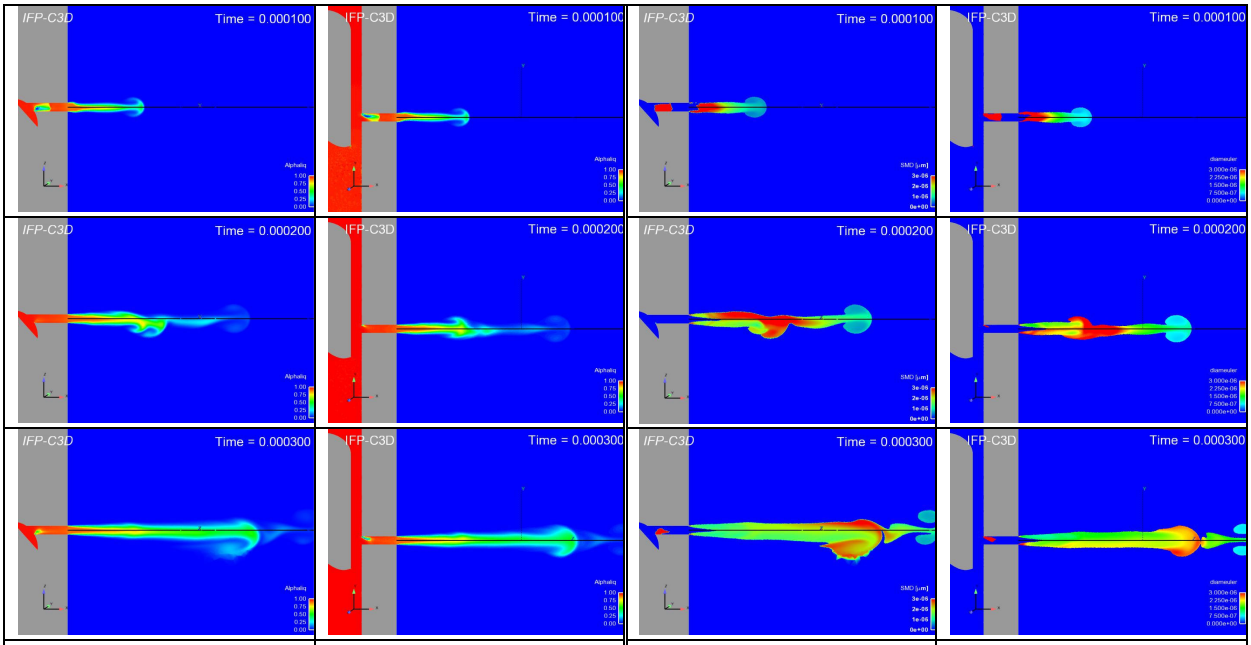


Figure 2. The 3D mesh. (a) Global view of the injector including the discharge system. (b) Meshing in the cross-section B-B highlighting the refinement extent inside the hole and in the SCC.

In addition, only a small cylindrical part of the SCC has been considered for the simulation of the liquid jet. Figure 2 shows the corresponding 3D unstructured mesh used in this study. It consists of ten million cells (tetra, prism, pyramid …) with a smallest characteristic size in the range of 25 to 50 microns inside the hole and in the first 20 mm downstream in the SCC (see Figure 2b). This refined grid is used to be able to compute the cavitation inception and the primary atomization of the liquid jet as explained in [7] [8]. Initially, the whole injector is full of liquid (n-dodecane) while the combustion chamber is full of gas (Nitrogen). As such, the liquid-gas interface is assumed to be located initially at the middle of the orifice. Since the transient needle lift of the injector is not simulated, the injection is initiated using "shock tube" like conditions upstream of the channel of width **s** (see Figure 1). The injection pressure value (70 MPa) is initialized in the inlet part of the injector while the rest of the configuration pressure is initialized equal to 4 MPa, which is the same pressure than the gas phase in the combustion chamber. The temperature of the gas in the combustion chamber is set to 400 K; while the injected fuel temperature is assumed equal to ambient, 295 K. This temperature is also assumed to be that of the walls. In addition, wall laws derived for the liquid-gas mixture, are applied for the computation of the velocity and heat transfer at the walls as the grid (although relatively refined around 25 µm, see Figure 2) does not generally capture properly the near wall effects. Finally, the computations are assumed to start with a zero velocity.

4. Results and Discussion

The numerical simulations have been carried out using the newly developed two-fluid model implemented in the in-house IFPEN code [10]. The cavitation appears classically close to the sharp inlet edge, whether in the A-A or B-B section (Figure 3). The spray plume is deflected in the same side as the cavitation pockets, as shown in the cross-section A-A (t > 100 µs). The liquid flow bypasses the cavitation pocket, and then leaves the orifice with a small deviation to the bottom. This result is in good correlation with the experiments obtained recently by Schmid et al. [6]. Besides, pressure oscillations due to the opening of the needle (not shown here) have led to a very transient behaviour of the liquid flow and cavitation. The maximum velocity is up to 400 ms^{-1} inside the orifice. It is also important to notice that smaller droplets are produced in the side of the cavitation pocket, especially at the end of the transient injection, as shown in the cross-section B-B in Figure 4 for (t > 100 µs). This is the main results of this study. But, validation of the *TwoSD* atomization model is needed in future work.



(a) Cross-section A-A | (b) Cross-section B-B (Movie available online)

Figure 3. (color online) Liquid volume fraction in the cross-sections A-A and B-B (see Figure 1).

(c) Cross-section A-A | (d) Cross-section B-B (Movie available online)

Figure 4. (color online) Droplets size distribution in the cross-sections A-A and B-B (see Figure 1).

5. Conclusions

In this article, a strong correlation of cavitation and primary atomization of a liquid jet emerging from a large marine injector was demonstrated using a comprehensive two-fluid model that includes the *GERM* cavitation model and the *TwoSD* atomization model newly developed at IFPEN. The numerical results seem to indicate that cavitation is at the origin of the spray deviation off-axis observed experimentally for an eccentric nozzle configuration. In addition, this liquid spray deviation has been triggered by the cavitation collapse in the injection starting period. Later, the deviation is maintained at the cavitation side, by a highly asymmetric and transient velocity profile inside the orifice. This strong correlation between the in-nozzle cavitating flow and the liquid jet morphology (injection orientation and atomization processes) makes very difficult the simulations using a "static" coupling methodology.

Acknowledgments

The present work has been partly conducted during the HERCULES-C project within EC's 7th Framework Program, Contract SCP1-GA-2011-284354.

6. References

[1] Arcoumanis, C., Gavaises, M., and French, B. *SAE Technical Paper 970799* (1997).
[2] Bianchi, G. et al. *ASME J. of Engineering for Gas Turbines and Power* **123**, 419–427 (2001).
[3] Som, S. and Aggarwal, S. *Combustion and Flame* **157**(6), 1179 – 1193 (2010).
[4] Herrmann, K. et al. In *Proceedings of ISME, Paper-ISME111, Kobe, Japan, Oct. 17-21*, (2011).
[5] Hensel, S. et al. In *Proceedings of the COMODIA conference*, (2012).
[6] Schmid, A. et al. In *Proceedings of ILASS-Europe, 8-10 Sep. 2014, Bremen, GERMany*, (2014).
[7] Habchi C., C. *Atomization and Sprays*, **25 (4)**, 317–334 (2015).
[8] Devassy, B. M., Habchi, C., and Daniel, E. *Atomization and Sprays*, **25(1)**, 47–80 (2015).
[9] Herrmann, K. Technical report, PSI on behalf of WCH (2015).
[10] Velghe, A., Gillet, N., and Bohbot, J. *Computers & Fluids* **45**(1), 116 – 121 (2011).