On the role of Cavitation in Marine Large Diesel injector: Numerical investigation of nozzle orifices eccentricity

Conference Paper · September 2014

DOI: 10.13140/2.1.2017.7280

Chaouki Habchi
IFP Energies nouvelles
101 PUBLICATIONS **1,367** CITATIONS

Nicolas Gillet
IFP Energies nouvelles
10 PUBLICATIONS **134** CITATIONS

Julien Bohbot
IFP Energies nouvelles
37 PUBLICATIONS **366** CITATIONS

A. Schmid
Winterthur Gas & Diesel
31 PUBLICATIONS **154** CITATIONS

On the role of Cavitation in Marine Large Diesel injector: Numerical investigation of nozzle orifices eccentricity

Chawki Habchi*[1], Nicolas Gillet[1], Antony Velghe[1], Julien Bohbot[1]
Andreas Schmid[2], Beat von Rotz[2], Kai Herrmann[3]
[1]IFP Energies nouvelles, 1 et 4 avenue de Bois-Préau, 92852 Rueil-Malmaison, France
[2]Wärtsilä Switzerland Ltd., CH-8401 Winterthur, Switzerland
[3]Paul Scherrer Institute, CH-5232 Villigen PSI, Switzerland
*Corresponding author: Chawki.HABCHI@IFPEN.fr

Abstract

The injector geometry of large marine two-stroke diesel engines differs substantially from the configurations used in most other diesel engine applications, because the injector orifices are distributed in a highly non-symmetric manner. In order to investigate the impact of key features of such asymmetrical orifice arrangements on the liquid jet and mixture preparation, a simplified generic single-hole nozzle has been designed. In this work, the in-nozzle flow has been investigated numerically in the aim of identifying the role of cavitation in large marine Diesel injectors. Two configurations of such nozzles with different eccentricity of the hole have been computed. High Fidelity Simulations (HFS) of the entire process including cavitation and liquid jet atomization have been carried out using a newly developed two-fluid version of the software IFP-C3D [1, 2] which includes the advanced *GERM* (Gibbs Equilibrium Relaxation Model) cavitation model [3] and the Two-Surface Density (*TwoSD*) atomization model [4].
First, the results have shown that cavitation appears classically close to the orifice inlet edge, but it has a smaller size for the eccentric nozzle. For both configurations, the cavitation induces a flow deviation inside the holes which affects significantly the liquid jet atomization. The liquid flow has been deviated from the orifice axis by the cavitation pocket, leading to an off-axis spray in the combustion chamber. This asymmetry is at the origin of the spray deviation observed in the "Spray Combustion Chamber" (SCC) [5] experiments for an eccentric nozzle.
Finally, the numerical results have shown a strong correlation between the in-nozzle cavitating flow and the spray direction and atomization that makes very difficult the simulations using a weak coupling methodology which consist of a preliminary in-nozzle flow simulation and then a second simulation for the spray injection, the mixture preparation and the combustion.

Introduction

Over the past decade large efforts have been made to improve the efficiency of large marine engines while reducing emissions. Significant parts of this work have been achieved during the HERCULES projects (see e.g. http://www.hercules-b.com, http://www.hercules-c.com). During the first part of the HERCULES projects the Spray Combustion Chamber (SCC) has been developed [5] [6]. Since its completion the setup has been successfully used to generate high-quality reference data (e.g. [7]). Latest investigations on spray morphology [8] have shown that an eccentric arrangement of the orifice – with respect to the central bore of the injector – has significant impact on the spray: On the one hand, high eccentricity levels lead to strong deflection of the spray plume, mainly against the direction of eccentricity. On the other hand, the spray cross-section is becoming more and more distorted with increasing eccentricity, thus making the commonly applied assumption of axis-symmetry of the spray invalid. In addition, initial Large Eddy Simulations (LES) of similar configurations carried out by Hensel et al. [9] seem to indicate the establishment of rather crescent-type shapes of the spray liquid core. The experiments also showed that the spray tip penetration is substantially altered in case of eccentric arrangements of the orifice: the spray tip penetration is slowed down considerably. As a consequence of the results of these experimental investigations, detailed simulations of the in-nozzle flow were performed to provide a better understanding of the underlying phenomena triggering these behaviors. In this work, the results of high fidelity simulations (HFS) are presented. In the following section, the different computational configurations of the nozzles will be defined. Next, the IFP-C3D software and its main sub-models will be described briefly since they are available elsewhere [3] [4]. Then, the results discussion section presents the main findings of the study, before the conclusions.

Computational configurations

In order to investigate the impact of key features of asymmetrical orifice arrangements on the liquid jet and mixture preparation, a simplified generic single-hole nozzle has been used (Figure 1). This specimen has been designed in order to mimic the injection process from a typical five holes nozzle of large marine Diesel engines. It consisted of an elongated tip with a single injection hole. The details of the nozzle configuration can be found in Figure 1. The direction of the fuel flow, the location of the spray orifice and the resulting main injection direction are indicated on the left. 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).

The layout depicted in Figure 2 presents the dimensions of the different components of the injector configuration. In particular, the discharge pipe has been simplified based on its estimated volume. In addition, only a small cylindrical part of the SCC has been considered for the simulation of the liquid jet. Figure 3 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 3c). This refined mesh is used to be able to compute the primary atomization of the liquid jet as explained in Devassy et al. [4].

Modeling methods

In this work, a comprehensive highly compressible two-fluid multi-species model is used. 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. Instead of the Reynolds Averaged Navier-Stokes (RANS) formulation adopted in [3] and [4], a turbulent viscosity-based model is used in this work in order to avoid the excessive liquid turbulent viscosity generated by standard k-ε turbulence model. A standard Smagorinsky model is adopted for sub-grid scale turbulence modeling for the numerical simulation carried out in this work. For the present cavitation modeling, an instantaneous relaxation procedure is used for the velocity, pressure and temperature; while a slower procedure is adopted for the Gibbs free energy relaxation model (*GERM*) at the interfaces. These models have already been applied by the authors in [3] for the simulation of cavitation inside an eccentric Diesel nozzle and the cavitation got developed on the right side of the orifice, very similar to experimental visualizations.

Present simulations also adopt the *TwoSD* atomization model developed recently by Devassy et al. [4]. This model uses a two-surface density approach within the framework of an Eulerian two-phase system. 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 modeling stages for both the droplets and liquid core. Thereby, 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. Indeed, the complexity of expressing the complete system of primary atomization and secondary breakup processes is thus reduced. 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 probability density function (PDF) and can later be vaporized using appropriate evaporation models.

Numerical methods

The numerical simulations have been carried out using the newly developed two-fluid version of the software IFP-C3D [1] which includes the *GERM* cavitation model [3] and the *TwoSD* atomization model [4] described in the previous section. The accuracy of the numerical schemes of IFP-C3D are first order in time, second order in space for diffusion terms and quasi-second order in space for convection terms [2]. Since the time steps used in HFS simulations are relatively small, first order accuracy in time seems to be reasonable. However, a higher accuracy in space is usually requested for Large Eddy Simulations (LES), the present work may be considered as a first attempt to simulate two-phase flows with highly compressible liquid in complex configurations including cavitation and atomization of liquid jets. Indeed, these physical phenomena are known to be very stiff, rendering more unstable the simulations using spatially centered high order schemes.

The initial and boundary conditions are given in Figure 4. 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 (Figure 4a). Due to the two-fluid modelling, each phase contains a weak volume fraction α_ε of the other phase. On one hand, the volume fraction of gas in the liquid $\alpha_{\varepsilon g}$ is assumed equal to 0.1. On the other hand, a much smaller value of liquid volume fraction $\alpha_{\varepsilon l}$ in the gas is necessary and a value $\alpha_{\varepsilon l}= 10^{-6}$ is adopted from previous works [3] [4]. The injection pressure is 70 MPa. 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 (Figure 4b). The temperature of the gas in the combustion chamber is set to 400 K; while the injected fuel temperature is assumed equal to ambient temperature, 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. Finally, the computations are assumed to start with a zero velocity and turbulent kinetic energy.

Results and Discussion

Figure 5 and 6 depict the injection process in terms of liquid volume fraction in the A-A and B-B cross-sections (see Figure 1) for the non-eccentric (e=0) and the eccentric (e=0.8) nozzle configurations, respectively. The cavitation appears classically close to the orifice inlet edge, but it has a smaller size for the eccentric hole. Comparing the two sprays in axial direction, one can see that for both cases, the liquid flow circumvents the cavitation. This leads to a flow field at the exit of the orifice, which in case without eccentricity results in a deviation of the liquid jet from the orifice axis against the direction of the flow in the nozzle (i.e. the deflection is in the same side as the cavitation pocket, see cross-section B-B in Figure 5). The lateral spray on the other hand shows a different behavior. In case of eccentricity the spray plume is deflected against the direction of the eccentricity (i.e. the deflection is again in the same side as the cavitation pocket, see cross-section A-A in Figure 6). For the non-eccentric case, the simulation are to a certain extend in good correlation with the experiments obtained recently and presented in [10].

In the aims to better identify the origin of the spray deviation for the case with eccentricity, Figure 8a shows the liquid core (iso-surface of volume fraction equal 0.5) evolution and the velocity field in the cross-section A-A. One can see the velocity deflection and acceleration at the sharp inlet of the eccentric hole. This deviation persists until the hole exit and directs the liquid core slightly off the axis. In addition, one can note in Figure 8b the swirling flow field established inside the hole and particularly the velocity magnitude oscillations due to pressure waves and cavitation collapse, that may be seen in Figure 6. This study corroborates the well-known huge effect of the pressure oscillations and the cavitation collapse on the liquid core deformation (see Figure 8a) and atomization (see Figure 9). Indeed, lower primary atomization rate is obtained during the liquid deceleration owing to the cavitation collapse (see Figure 9 with Figure 8). It is also important to notice on Figure 9b that smaller droplets are produced in the side of the cavitation pocket, especially at the end of the transient injection (t > 200 µs). The order of magnitude of the droplets Sauter mean diameter (SMD) seems fairly reasonable. But, validation of the *TwoSD* atomization model is needed in future work.

Finally, the temperature field is shown in the A-A and B-B cross-sections in Figure 10. This Figure gives a more clear view of the spray deviation than the liquid volume fraction shown in Figure 6. In addition, it is worth to notice in Figure 10b that before entering into the channel leading to the orifice, the liquid experiences a significant pressure and temperature increase which are up to 90 MPa and 325 K, respectively. These values are to be compared to the injection pressure, 70 MPa, and inlet liquid temperature, 295 K. Nevertheless, the liquid has recovered a value close to the inlet temperature upon entering the hole. Moreover, a very small decrease of temperature (-2 K) is obtained at the cavitation pocket center. This low cooling value seems to be correct in the present gaseous cavitation regime due to the very small vapor pressure at the present cold condition (liquid n-dodecane inlet temperature and wall temperature are equal to 295 K). Due to phase change in the vaporous cavitation regime, stronger cooling effects are expected at higher liquid temperature conditions (compare [3]).

Conclusions

In this study, the role of cavitation on the liquid spray characteristics of large marine injectors has been investigated using a high fidelity simulation method. In this aim, the IFP-C3D code including the *GERM* cavitation model and the *TwoSD* liquid jet atomization model, recently developed, have been used.

The numerical results seem to indicate that cavitation is at the origin of the spray deviation off-axis observed experimentally for the 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 which is against the eccentricity direction, by a highly asymmetric and swirling velocity profile at the hole exit. This strong correlation between the in-nozzle cavitating flow and the liquid jet morphology (injection direction and atomization processes) makes very difficult the simulations using a weak coupling methodology which consist of a preliminary in-nozzle flow simulation and then a second simulation for the spray and the combustion.

Future work will focus on quantitative validation of the different physical models. In addition, the eventual necessity of more accurate numerical schemes will be investigated in the context of the LES simulations of two-phase highly compressible flows.

Abbreviation

GERM	Gibbs Energy Relaxation Model
HFS	High Fidelity Simulation
IFP-C3D	Commercial 3D two-phase CFD and combustion software developed by IFPEN and distributed by LMS/Siemens co.
LES	Large Eddy Simulation
SCC	Spray Combustion Chamber
SMD	Sauter Mean Diameter
TwoSD	Two Surface density atomization model

Acknowledgements

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

References

[1] J. Bohbot, N. Gillet, and A. Benkenida, “IFP-C3D: an Unstructured Parallel Solver for Reactive Compressible Gas Flow with Spray,” *Oil & Gas Science and Technology*, vol. 64, no. 3, pp. 309–335, 2009.

[2] A. Velghe, N. Gillet, and J. Bohbot, “A high efficiency parallel unstructured solver dedicated to internal combustion engine simulation,” *Computers & Fluids*, vol. 45, no. 1, pp. 116 – 121, 2011. Also in the Proceedings of the 22nd International Conference on Parallel Computational Fluid Dynamics (ParCFD 2010).

[3] C. Habchi, “A Gibbs free Energy Relaxation Model for Cavitation Simulation in Diesel injectors,” in *Proceedings ILASS - Europe 2013, 25th European Conference on Liquid Atomization and Spray Systems, Chania, Greece, 1-4 September*, 2013.

[4] B. M. Devassy, C. Habchi, and E. Daniel, “A New Atomization Model for High Speed Liquid Jets using a Turbulent, Compressible, Two-Phase Flow Model and a Surface Density Approach,” in *Proceedings ILASS - Europe 2013, 25th European Conference on Liquid Atomization and Spray Systems, Chania, Greece, 1-4 September*, 2013.

[5] K. Herrmann, B. V. Rotz, R. Schulz, G. Weisser, B. Schneider, and K. Boulouchos, “A "Spray Combustion Chamber" facility for investigations in relation to large 2-stroke Marine Diesel engine combustion system optimization,” in *Proceedings of the International Symposium on Marine Engineering (ISME) October 17-21, Kobe, Japan Summary or Paper-ISME111*, 2011.

[6] K. Herrmann, R. Schulz, and G. Weisser, “Development of a reference experiment for large diesel engine combustion system optimization,” in *CIMAC*, (Vienna, Austria), 2007.

[7] R. Schulz, S. Hensel, B. von Rotz, A. Schmid, K. Herrmann, and G. Weisser, “Development of Spray and Combustion Simulation Tools and Application to Large Two-Stroke Diesel Engine Combustion Systems,” in *Proceedings of the Conseil International des Machines a Combustion (CIMAC) Congress, Shanghai*, no. NO. 259, 2013.

[8] A. Schmid, B. von Rotz, R. Schulz, K. Herrmann, G. Weisser, and R. Bombach, “Influence of nozzle hole eccentricity on spray morphology,” in *Proceedings ILASS - Europe 2013, 25th European Conference on Liquid Atomization and Spray Systems, Chania, Greece, 1-4 September*, 2013.

[9] S. Hensel, K. Herrmann, R. Schulz, and G. Weisser, “Numerical analysis and statistical description of the primary breakup in fuel nozzles of large two stroke engines for the application in CFD engine simulations,” in *Proceedings of the COMODIA conference*, 2012.

[10] A. Schmid, C. Habchi, J. Bohbot, B. von Rotz, R. Schulz, K. Herrmann, R. Bombach, and G. Weisser , “Influence of in-nozzle flow on spray morphology ,” in *Proceedings ILASS - Europe 2014, 26th European Conference on Liquid Atomization and Spray Systems, Bremen, Germany, 8-10 September*, 2014.

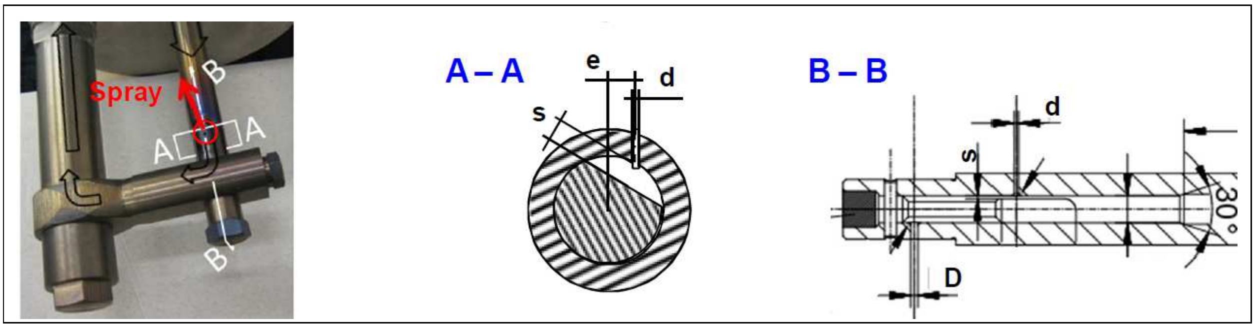


Figure 1. Generic injector configuration. Photograph inside the spray combustion chamber (left). Cross-section A-A through the injection orifice (d=0.75 mm and e=1.5 mm for the eccentric configuration) (middle). Cross-section B-B through the injector bore with s=0.75 mm and D=1.3 mm (right).

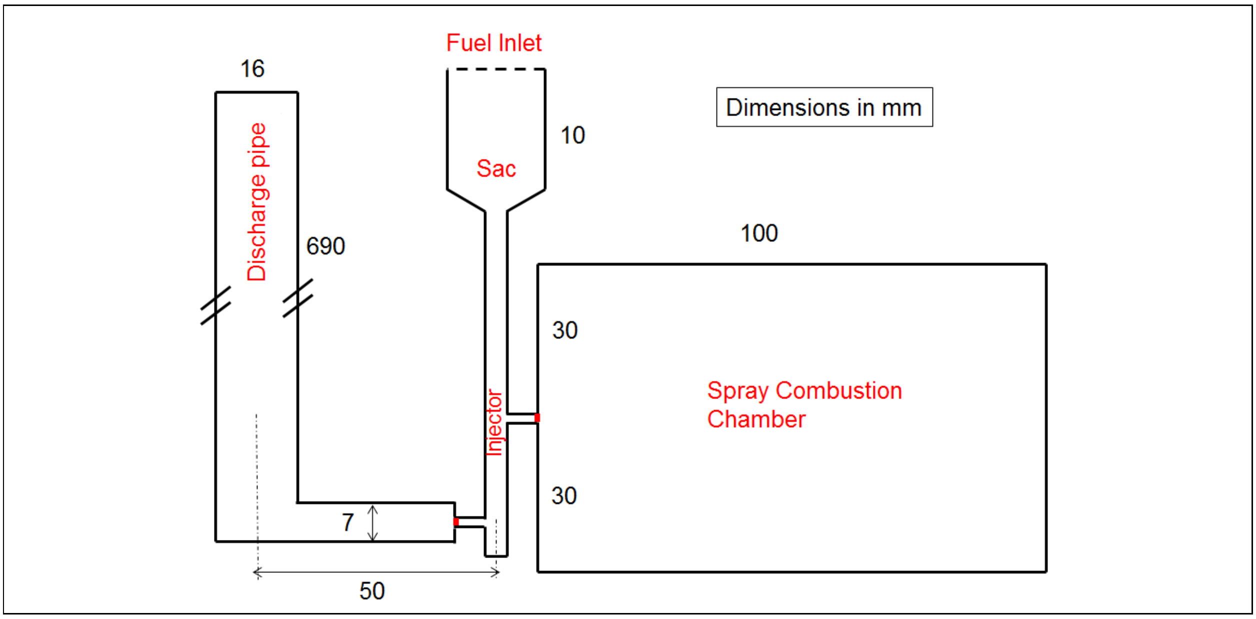


Figure 2. Dimensions of the different components of the 3D injector configurations.

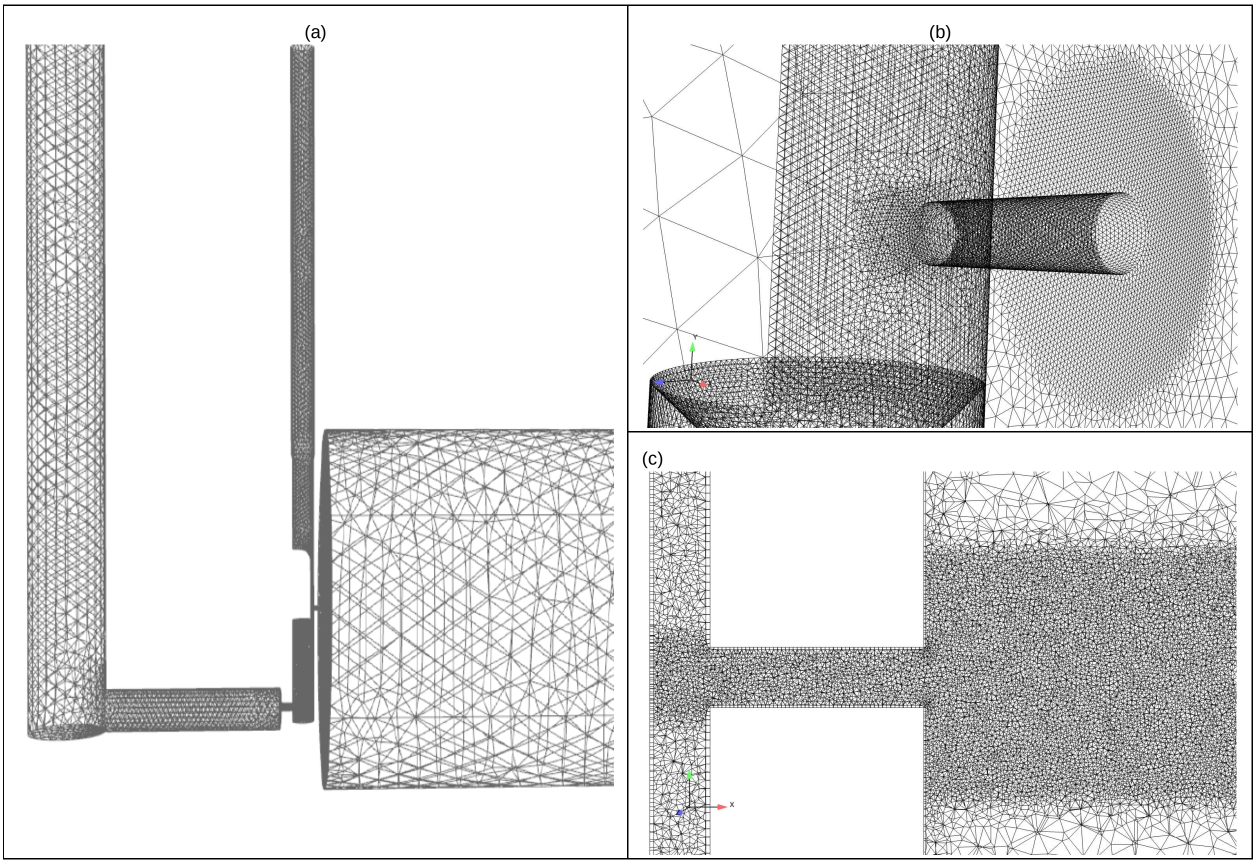
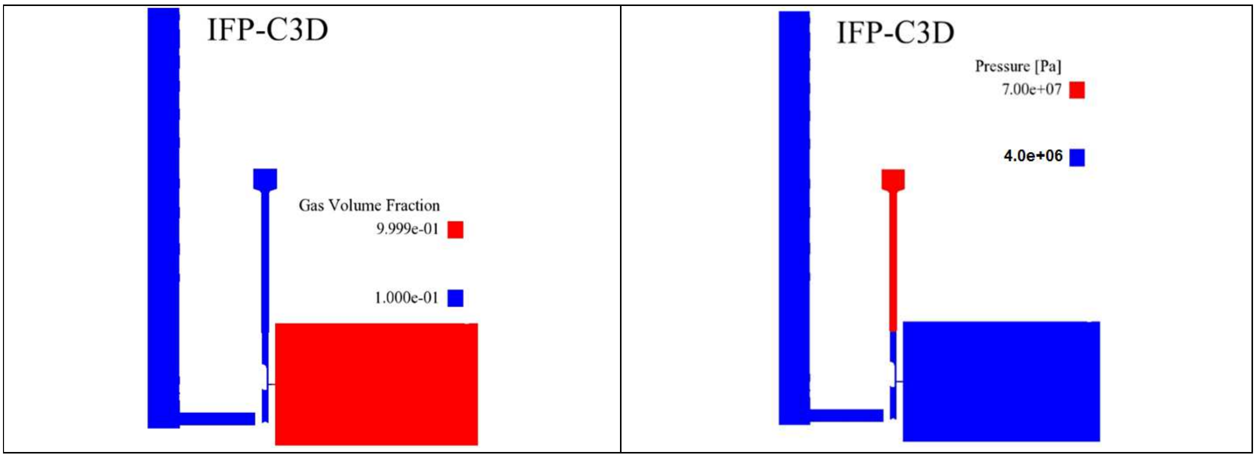


Figure 3. The 3D mesh. (a) Global view of the injector including the discharge system. (b) Zoom on the injection hole.

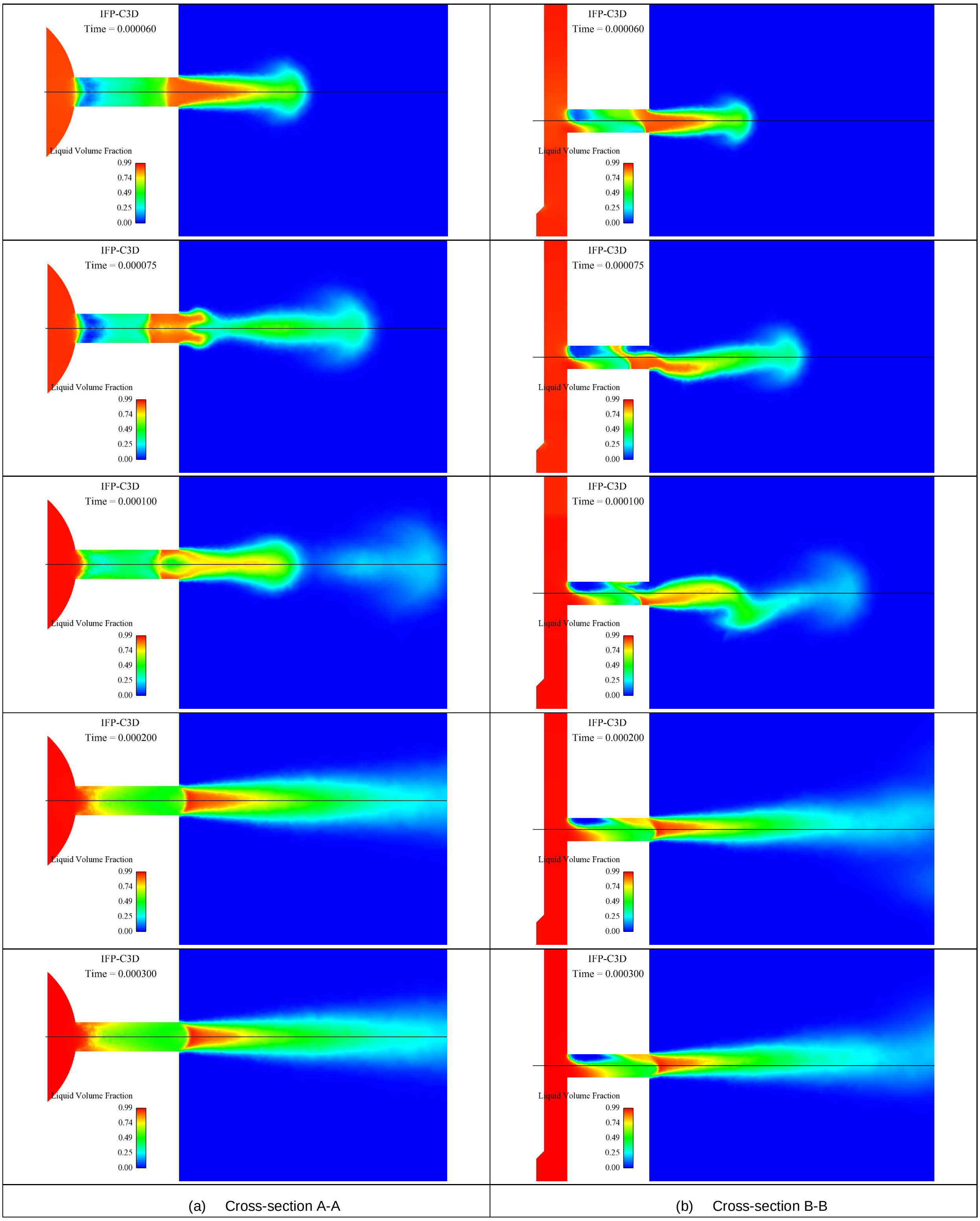
(c) Meshing in the cross-section B-B highlighting the refinement extent inside the hole and in the SCC.



(a) Initial volume fraction

(b) Initial pressure

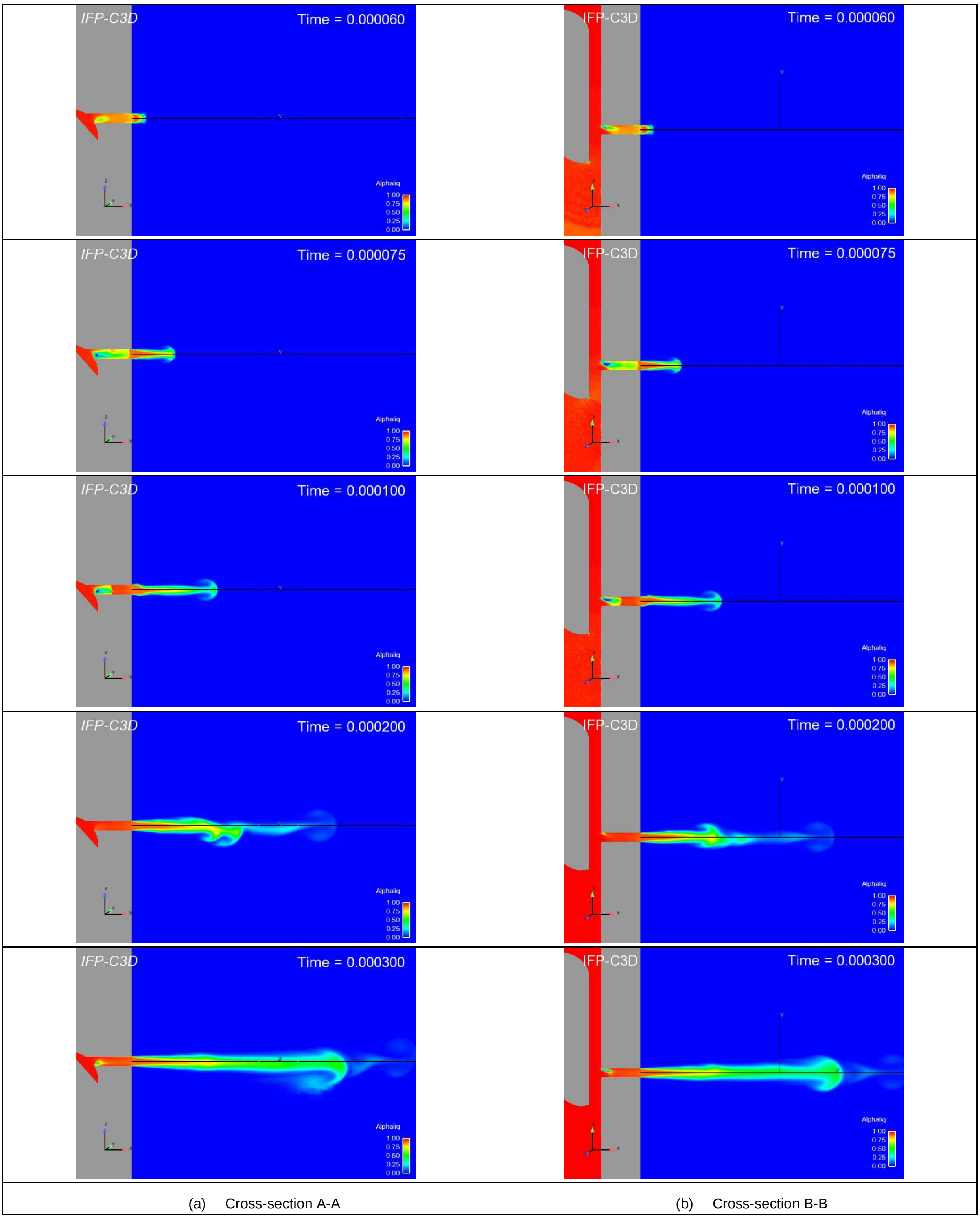
Figure 4. (color online) Initial conditions for the gas volume fraction and pressure (Injector cross-section B-B in Figure 1).



(a) Cross-section A-A

(b) Cross-section B-B

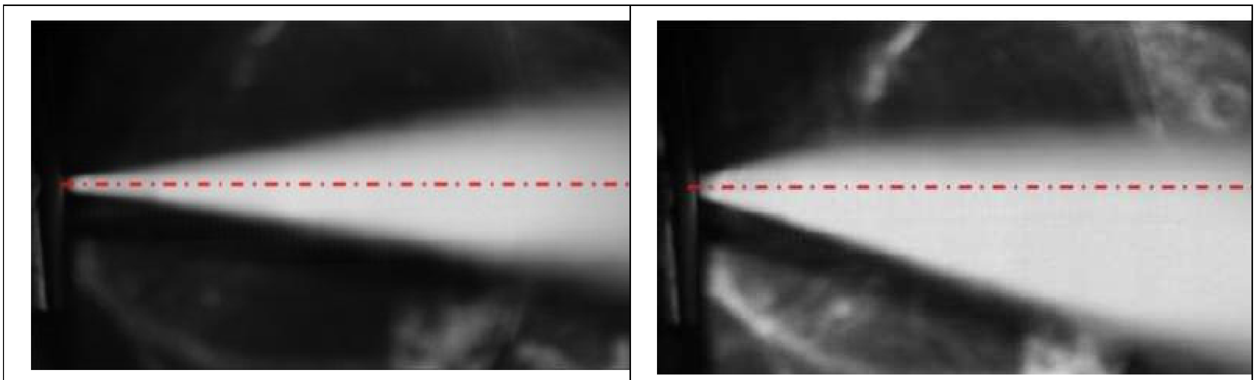
Figure 5. (color online) Liquid volume fraction for the case without eccentricity. Cavitation and liquid jet in cross-sections A-A and B-B.



(a) Cross-section A-A

(b) Cross-section B-B

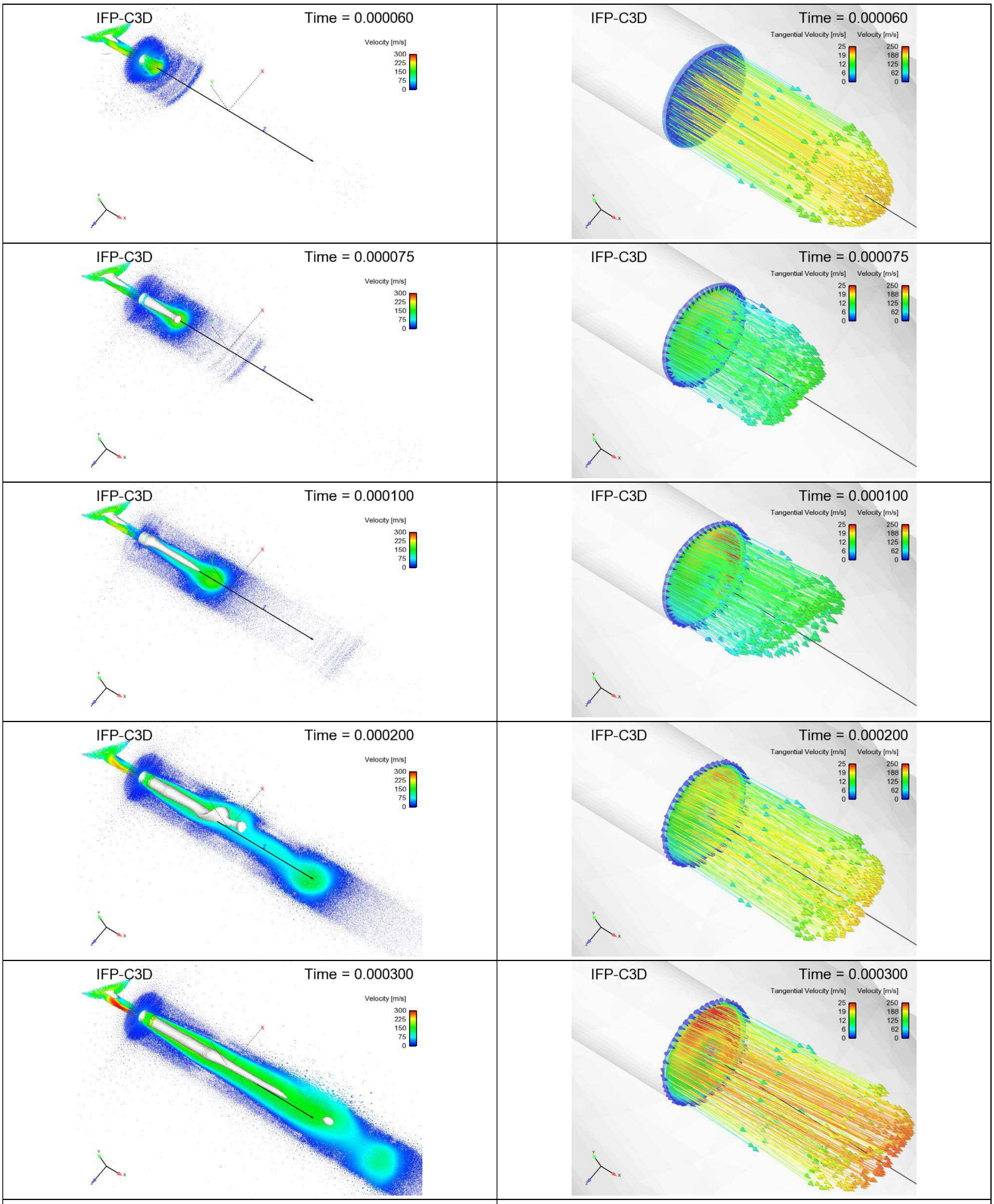
Figure 6. (color online) Liquid volume fraction for the case with eccentricity. Cavitation and liquid jet in cross-sections A-A and B-B.



(a) Nozzle without eccentricity

(b) Nozzle with eccentricity

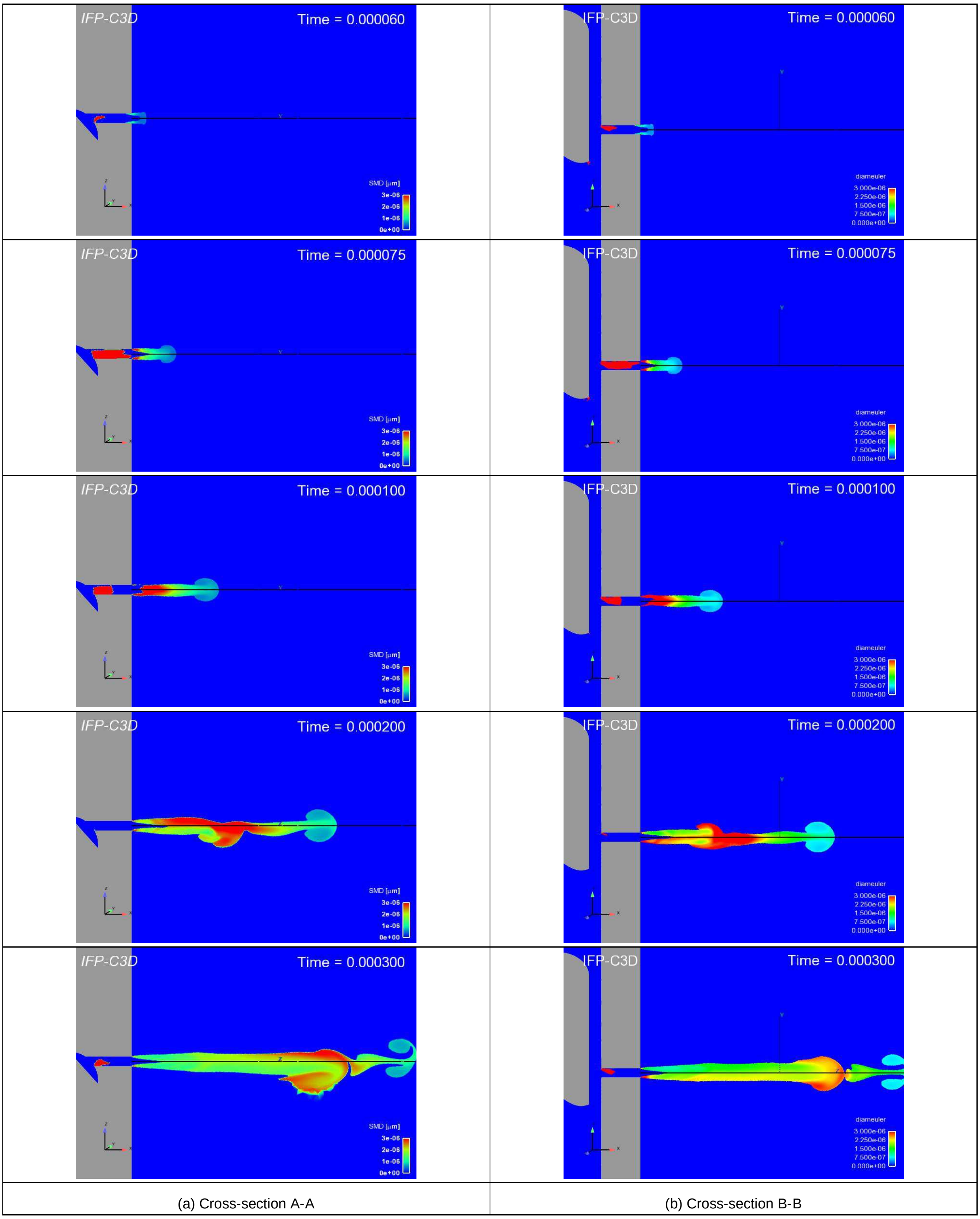
Figure 7. Spray visualization by means of Mie-scattering (lateral view from the front side: A-A). The images have ones been averaged over time and the resulting images have been averaged over ten individual experiments.



(a) Liquid core (iso-surface of volume fraction equal 0.5) and velocity field in the cross-section A-A (viewed from the top).

(b) Hole exit 3D velocity profile (arrows) and swirling tangential velocity (scalar). The swirl rotates in clockwise.

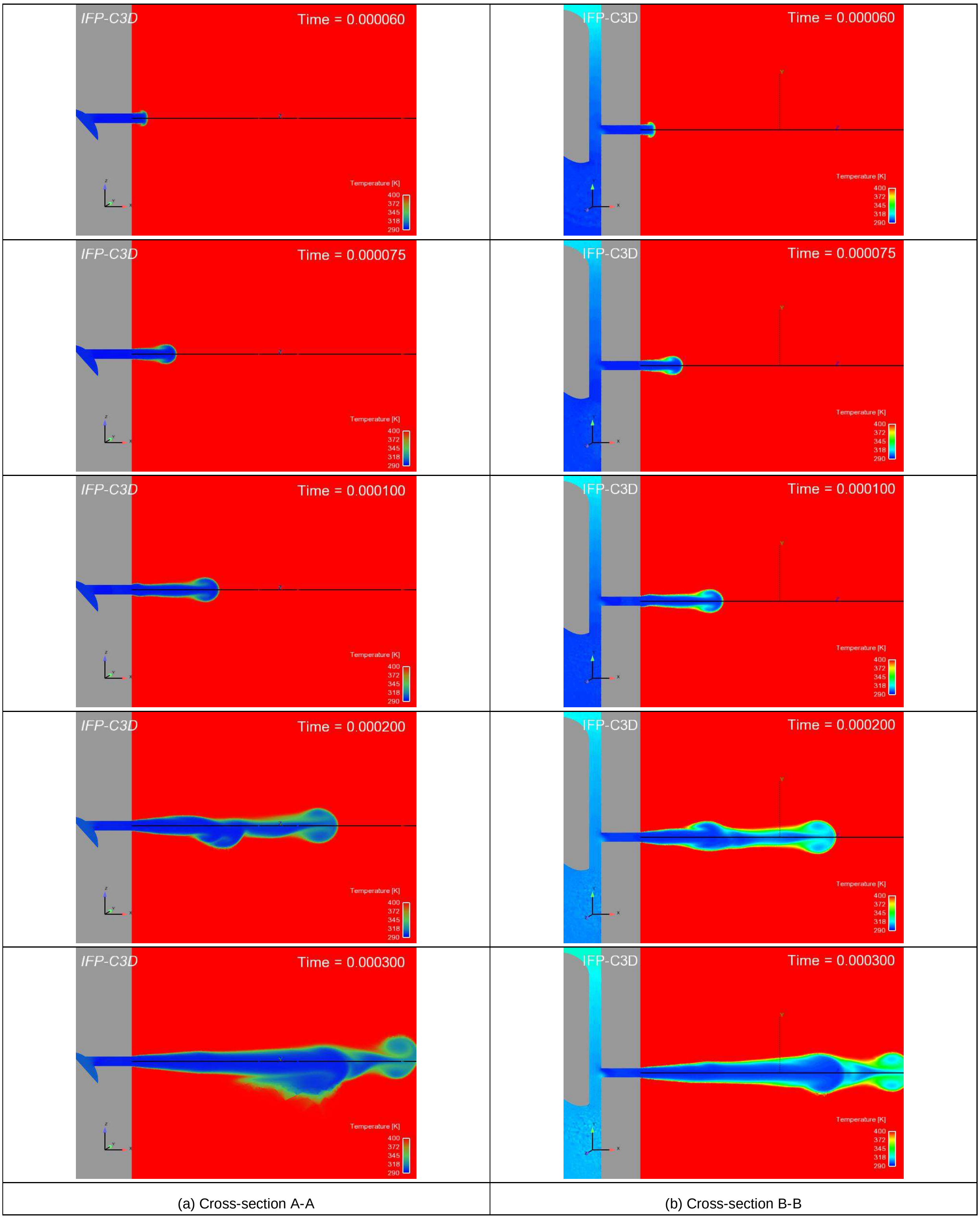
Figure 8. (color online) Highlighting of the origin of the deviation of the liquid jet for the case with eccentricity.



(a) Cross-section A-A

(b) Cross-section B-B

Figure 9. (color online) Droplets size in cross-sections A-A and B-B for the case with eccentricity.



(a) Cross-section A-A

(b) Cross-section B-B

Figure 10. (color online) Temperature fields in cross-sections A-A and B-B for the case with eccentricity.