

Analysis of Transient Cavitating Flows in Diesel Injectors Using Diesel and Biodiesel Fuels

2010-01-2245

Published
10/25/2010

Michele Battistoni and Carlo Nazareno Grimaldi
Universita degli Studi di Perugia

Copyright © 2010 SAE International

ABSTRACT

The aim of the paper is the comparison of the injection process with different fuels, i.e. a standard diesel fuel and a pure biodiesel. Multiphase cavitating flows inside diesel nozzles are analyzed by means of unsteady CFD simulations using a two-fluid approach with consideration of bubble dynamics, on moving grids from needle opening to closure. Two five-hole nozzles with cylindrical and conical holes are studied and their behaviors are discussed taking into account the different properties of the two fuels. Extent of cavitation regions is not much affected by the fuel type. Biodiesel leads to significantly higher mass flow only if the nozzle design induces significant cavitation which extends up to the outlet section and if the injector needle is at high lift. If the internal hole shaping is able to suppress cavitation, the stabilized mass flows are very similar with both fuels. On the contrary, as long as the lifts are small and the flow is turbulent, diesel fuel gives higher mass flows due to lower frictions along the needle seat and the nozzle walls. Detailed analyses of the injection processes are presented, including flow pattern development inside the nozzles.

INTRODUCTION

Despite a continuous debate on sustainability of biofuels for large mass production, their popularity keeps on growing and their production systems, from different biomasses, keeps on expanding all over the world. First generation biofuels and, more recently, second generation biofuels, are coming out as a realistic energy source for light and heavy duty transportation needs, especially for those applications where fuel economy and environmental impact are of primary concern, like diesel passenger cars or trucks. It is well recognized that for this kind of applications, biodiesel can offer a significant contribution in reducing pollutant emissions especially in terms of opacity. Besides, it is a bio-

derived product and thus one of the most efficient ways of reducing the carbon dioxide emissions. Manufacturers and researchers are working on engine optimizations since very long time and recently carbon dioxide reduction has become their main driver. As a consequence it is apparent that bio-derived fuels, if carefully tailored for modern powertrains, are an effective and reliable way for decreasing the total CO₂ emissions. Among renewable fuels, biodiesel made through transesterification of vegetable oils is one of the most common bio-fuels and it has been largely tested in standard injection system for High Speed Diesel Engines (HSDI) like commercial common rail systems [1, 2, 3, 4, 5] and injection tuning has been demonstrated to succeed in mitigating usual larger NOx emissions of biodiesel [6, 7] and in decreasing soot emissions considerably.

Nowadays, potentials have been well demonstrated, consequently engines and aftertreatment systems need systematic assessment and specific design to be ready for these new fuels, especially in terms of durability and repeatability. DI diesel engine performances are strongly affected by the nozzle flow characteristics, which control the atomization process of the exiting fuel and the subsequent spray development. In many operating conditions, cavitation occurs inside the injector holes and this is one of the most important parameters affecting fuel spray atomization [8]. The necessity for contemporaneous usage of diesel and biodiesel in the same engine makes the effort of having a deep knowledge of flow physics inside the injectors unavoidable, in order to ensure reliable calibrations of the injection parameters at all operating conditions and for all fuels [9].

Due to the difficulties in obtaining real-scale and real-time measurements from nozzle flow experiments, simplifications are often made to the injector or only integral quantities are collected. Moreover diesel injectors nowadays operate with

multiple injection events per engine cycle, with very strict requirements for dwell times, so that internal transient flows can be of high interest. Numerical simulation, by means of CFD codes, applied to the nozzles multiphase flow is becoming a fundamental point in the design process of injectors and combustion chambers. In this area an increasing number of numerical methods have been proposed for modeling the cavitation phenomena, typically encountered in these applications.

Literature data about biodiesel application to standard common rail injection systems are found in [2, 10, 11, 12]. Payri et al. [10] found that the higher density and kinematic viscosity of bio-derived fuel strongly affect how injection system operates. Their tests showed that the use of biodiesel fuels leads to a higher mass flow when the injector is fully open and long energizing times are used. As a consequence the spray evolution is affected, biodiesel penetrates more and the spray is narrower. For short injections they also noticed that biodiesel delays the needle lift due to the increased relevance of viscosity effects and in these conditions a longer time is required to inject the same amount of fuel, being also reduced the instantaneous flow rate. Priesching et al. [2] found that at low engine load (1500rpm @ 3.2bar of IMEP) the main injection mass flow is reduced when biodiesel is used, while at a higher engine load (2000 rpm @ 7 bar IMEP) with longer injection duration and higher metal temperatures biodiesel and diesel practically give the same mass flow. Grimaldi et al. [11] found that biofuel delays start of injection, favors spray penetration and decreases break-up time. Postrioti et al. [12] noticed that at high injection pressures biodiesel causes a decrease of the rising slope of the mass flow. They also observed strong effects on the injection duration, at the same energizing time, which were attributed to the higher viscosity of the biofuel.

From the modeling standpoint three dimensional un-steady fluid dynamics simulations of the injector internal fluid domain have to be carried out in order to capture as much physics as possible, though it is clear that mutual effects of the fuel characteristics on the needle movement are far from being practicable in 3D.

Basically, the solution of the governing Navier-Stokes equations is required. Different approaches can be followed to treat the two-, or multi-, phase flow. They can be classified at least into three groups, namely the homogeneous Eulerian approach, the multi-fluid Eulerian approach and the Lagrangian approach. The common feature of the homogeneous method is the assumption that all the phases share the same velocity. In this case the fluid is a continuous mixture of liquid and vapor. They can assume different forms depending on how the density and the pressure are formulated, i.e. isothermal flows, isentropic flows, etc. The mass exchange between phases is usually due to the growth or collapse of spherical bubbles. The second aforementioned

approach is the Eulerian multi-fluid method. This is the method applied in this work. It is characterized by different sets of RANS equations one for each phase, thus the main feature is that each phase has its own different velocity. Mass exchanges between phases are usually modeled on the basis of bubble dynamics results [13]. Momentum exchanges can be modeled taking into account drag, turbulent dispersion, virtual mass, etc.. The third approach is the Lagrangian method. Here, only the liquid is a continuum, while the vapor is represented by parcels of bubbles. Mass and momentum exchange between the continuous phase and the dispersed phase are accounted. Vapor bubbles trajectories are tracked integrating the Newton equation of motion for each parcel. Also with this approach it is possible to take into account bubble dynamics, bubble break-up, drag, turbulence dispersion, virtual mass, lift [14, 15, 16, 17, 18].

In principle the Eulerian multi-fluid or the Lagrangian approaches should be more accurate, since both can treat momentum of the two phases separately. However, all the three methods have been used successfully. As a matter of fact, an even more important subject is the accuracy of physical details with which exchange terms between phases are modeled [14].

In the present paper a numerical comparison of the injection process with diesel and biodiesel using different nozzle designs is presented. The work was carried out using a two-fluid model for the prediction of cavitation, as implemented in the commercial CFD software, AVL-Fire [19]. The first part of the work is devoted to a preliminary assessment of the CFD approach, by comparing numerical results with literature experimental data on simple geometry nozzles. The second part of the work focuses on the modeling of a diesel injector geometry with cylindrical and conical nozzles and different rounding radii during the transient opening and closing of the needle, using moving mesh with a prescribed law. This approach clearly can give useful information on the cavitating flow, apart from the possible effects due to the coupling of fluid properties and needle movement, as observed in some of the aforementioned experimental results; as a consequence it could provide results much more realistic for direct-acting piezo-driven injectors than for solenoid-driven injectors.

NUMERICAL MODEL

The current study makes use of the Eulerian multi-fluid method for the simulation of a two-phase flow in cavitating conditions. The method is an application of a generalized computational code where n different phases can coexist, based on the assumption of interpenetrating continua. In general, a multi-phase flow can comprise liquid, vapor, incondensable gases, droplets, bubbles, solid particles, etc... In this framework, each phase is considered as a continuous medium and for each one a set of conservation equations

applies. The mathematical formalism is obtained through ensemble averaging to remove the microscopic interfaces at a sub-scale level, resulting in n sets of macroscopic conservation laws which are analogous to the Reynolds-Averaged-Navier-Stokes equations, with the addition of a new scalar quantity, that is the volume fraction α_i of the phase and the relevant source/sink terms for interfacial exchange of conserved quantities. For $n = 1$, they actually reduce to the RANS equations for a single phase flow.

Averaged continuity and momentum equations [19, 20, 21, 22] read as follows

$$\frac{\partial \alpha_i \rho_i}{\partial t} + \nabla \cdot \alpha_i \rho_i \mathbf{v}_i = \sum_{j=1, j \neq i}^n \Gamma_{ij} \quad (1)$$

$$\begin{aligned} \frac{\partial \alpha_i \rho_i \mathbf{v}_i}{\partial t} + (\nabla \cdot \alpha_i \rho_i \mathbf{v}_i) \mathbf{v}_i &= -\alpha_i \nabla p + \dots \\ \nabla \cdot \alpha_i (\boldsymbol{\tau}_i + \mathbf{T}_i) + \alpha_i \rho_i \mathbf{f} + \sum_{j=1, j \neq i}^n \mathbf{M}_{ij} &+ \dots \\ \mathbf{v}_i \sum_{j=1, j \neq i}^n \Gamma_{ij} \end{aligned} \quad (2)$$

Turbulent stresses are computed by adding to the standard turbulent viscosity of the continuous phase a bubble induced viscosity term, as proposed by Sato [23].

Pressure is assumed identical for all phases, $p = p_i$.

The compatibility condition must also be satisfied, $\sum_{i=1}^n \alpha_i = 1$.

Also for energy conservation and turbulence closure models analogous derivations apply [19, 20, 21, 22], here omitted.

In this paper the flow is assumed to be isothermal, hence no energy equation is solved. For the turbulence closure problem the k - ζ - f model is adopted with a hybrid wall treatment for all y^+ values [24, 25, 26]. Turbulence level of the dispersed phase is assumed to equal the level of the continuous phase.

Numerically, the Eulerian multiphase model is solved using the commercial implementation available in AVL Fire 2009 [19, 26]. The solution method is based on a fully conservative finite volume approach.

In the present work, convection terms are discretized using a first-order upwind scheme for all quantities, apart from pressure where a central differencing scheme is employed and from momentum where a blended upwind-minmod relaxed (2nd order) scheme is adopted. Temporal discretization is carried out using a first-order (Euler) implicit

differencing scheme. The overall iterative solution procedure is based on the semi-implicit method for pressure-linked equations algorithm (SIMPLE). For the solution of a linear system of equations, a conjugate gradient type of solver is used.

CAVITATION MODEL

In order to model cavitation, a two-phase flow has been considered ($n = 2$) including the liquid fuel phase and its vapor. Calculations have been performed assuming incompressible phases.

The interfacial exchange terms, relevant for two-phase cavitation, are the mass exchange Γ_{12} and the momentum exchange $\mathbf{M}_{12}$ terms. These take into account the effects exerted at the interfaces between the phases. Since the microscopic details of the interface are lost due to ensemble averaging, exchange quantities need to be modeled to represent their macroscopic effects.

Interfacial turbulence exchange has been neglected.

Mass exchange

For cavitation phenomena the mass exchange is modeled according to the Rayleigh-Plesset equation which governs bubble growth dynamics [13, 19].

$$\frac{\Delta p(t)}{\rho_l} = R(t) \ddot{R}(t) + \frac{3}{2} (\dot{R}(t))^2 \quad (3)$$

$$\Delta p = p_s - p + C_E \frac{2}{3} \rho_l k_l \quad (4)$$

The driving force for mass exchange is the pressure difference between the liquid and the vapor pressures. An additional term is also included, proposed by Egler [27], which accounts for the turbulent pressure fluctuations. C_E is usually in the range between 1 and 1.4 (1.2 in this work). To solve Eq. (3) for the bubble radius R , two approaches can be followed: a direct implicit solution of Eq. (3) itself, which is called non-linear approach; or alternatively a linearized model, where inertia effects are neglected. The latter leads to

$$\frac{\Delta p(t)}{\rho_l} \cong \frac{3}{2} (\dot{R}(t))^2 \quad (5)$$

The mass exchange term is then calculated from the rate of change of bubble vapor mass due to isothermal evaporation or condensation. For a single bubble the following equation applies

$$\frac{\partial m_b}{\partial t} = \rho_2 4\pi R^2 \dot{R} \quad (6)$$

If the bubble diameter distribution is mono-dispersed (i.e. constant) the substitution of (3), or (5), into (6) leads straightforwardly to the following equations

$$\Gamma_{12} = \frac{N'''}{C_R} \cdot \frac{\partial m_b}{\partial t} = \frac{1}{C_R} \cdot 3.95 \dots \frac{\rho_2}{\sqrt{\rho_1}} \text{sign}(\Delta p) (N''')^{\frac{1}{3}} (\alpha_2)^{\frac{2}{3}} \sqrt{|\Delta p|} \quad (7) \text{ linear model}$$

$$\Gamma_{12} = \frac{N'''}{C_R} \cdot \frac{\partial m_b}{\partial t} = \frac{1}{C_R} \rho_2 4\pi R^2 \sqrt{\frac{2}{3} \left(\frac{\Delta p}{\rho_1} - R\ddot{R} \right)} \quad (8) \text{ implicit non linear model}$$

where $C_R = 1$ for evaporation. Due to lack of modeling, for condensation rate the same formulation is assumed, with the introduction of a condensation reduction coefficient, $C_R \geq 1$, which reflects the fact that subcooled vapor could exist in fast transients and non equilibrium processes. Hence it could be used as a tuning constant of the model.

The bubble number density is assumed to be a function of the dispersed volume fraction, i.e. $N''' = N'''(\alpha_2)$, according to

$$N''' = \begin{cases} N_0''' & \text{for } \alpha_2 \leq 0.5 \\ 2(N_0''' - 1)(1 - \alpha_2) + 1 & \text{for } \alpha_2 > 0.5 \end{cases} \quad (9)$$

The value of N_0''' , for low concentrations of vapor phase, depends on the characteristics of the liquid. Values usually reported for diesel range from 10^{12} m^{-3} to 10^{14} m^{-3} [13, 19, 28]. The bubble radius is directly linked to N''' and to α_2 according to

$$R = \frac{1}{2} \left(\frac{6\alpha_2}{\pi N'''} \right)^{1/3} \quad (10)$$

On the other hand, if the bubbles have not a uniform size, the system is said to be in a poly-dispersion state. This can be of interest or can affect macroscopic results in many applications, and also in diesel injectors where, as an example, erosion phenomena are recognized to be significantly affected by very small bubble dynamics, which is better captured by this modeling approach. For polydispersed flows a probability distribution function for bubble size can be defined. Following [19, 29] two transport

equations for N''' , the bubble number density, and A''' , the interfacial area density, can be derived and solved.

$$\frac{\partial N'''}{\partial t} + \nabla \cdot (N''' \mathbf{v}) = \sum R_j + R_{ph} \quad (11)$$

$$\frac{\partial A'''}{\partial t} + \nabla \cdot (A''' \mathbf{v}) = \sum \Phi_j + \Phi_{ph} \quad (12)$$

Through these two quantities, N''' and A''' , interfacial mass exchange Γ_{12} can be solved. As an example, it can be shown that the mean Sauter diameter of the bubbles is

$$D_{32,b} = \frac{6\alpha_2}{A'''} \quad (13)$$

Details of the polydispersed flow implementation can be found in [19, 29], where closure models are provided and discussed. The mechanisms taken into account are coalescence due to turbulent random collisions and turbulent impact induced break-up associated to R_j and Φ_j terms, in addition to a bubble generation effect due to vaporization of cavitating liquid associated to R_{ph} and Φ_{ph} terms.

Momentum exchange

Interfacial momentum exchange is modeled taking into account drag and turbulent dispersion effects; inertia and lift effects are neglected.

Drag on a bubble is obtained assuming spherical shape, using a drag coefficient C_D depending on Reynolds number.

$$\mathbf{M}_{D,21} = C_D \frac{1}{8} \rho_1 A''' |\mathbf{v}_2 - \mathbf{v}_1| (\mathbf{v}_2 - \mathbf{v}_1) = -\mathbf{M}_{D,12} \quad (14)$$

$$C_D = \begin{cases} 192(1 + 0.10 Re^{0.75}) Re^{-1} & Re \leq 1000 \\ 0.438 & Re > 1000 \end{cases} \quad (15)$$

$$\text{where } Re = \frac{|\mathbf{v}_2 - \mathbf{v}_1| D}{\nu_1}$$

The turbulent mixing process between phases is associated to the momentum interaction at the interface, which induces turbulence production on the liquid phase through Sato model [23]. Turbulent dispersion force relates bubble diffusion to the continuous phase turbulent kinetic energy and dispersed phase volume fraction gradient.

Table 1. Fluid properties @ 40 °C. (In *Italic*, modified properties for “virtual fuels”).

Fuel		Liquid density [kg/m ³]	Liquid viscosity [kg/(m·s)]	Vapor density [kg/m ³]	Vapor viscosity [kg/(m·s)]	Saturation pressure [Pa]	References
1	EU diesel	825	0.0021	0.05	1E-6	1000	[12, 30]
2	SME biodiesel	870	0.0039	0.001	1E-6	10	[12, 31-33]
3	D2 diesel	854	0.0029	0.1	1E-5	1000	[30]
3.A	Viscosity of #1	854	<i>0.0021</i>	0.1	1E-5	1000	
3.B	Viscosity of #2	854	<i>0.0039</i>	0.1	1E-5	1000	
3.C	P _s of #2	854	0.0029	0.1	1E-5	<i>10</i>	
3.D	Density of #2	<i>870</i>	0.0029	0.1	1E-5	1000	

$$\mathbf{M}_{T,21} = C_T \rho_1 k_1 \nabla \alpha_2 = -\mathbf{M}_{T,12} \quad (16)$$

where $C_T \cong 0.1$

The overall momentum exchange term is

$$\mathbf{M}_{21} = \mathbf{M}_{D,21} + \mathbf{M}_{T,21} \quad (17)$$

FLUID PROPERTIES

The main purpose of this study is the comparative analysis of diesel and biodiesel performance through nozzles of high-pressure common-rail injectors. Diesel fuel is a blend of thousands of hydrocarbons and its composition depends on many factors such as the geographical origin of the oil or the production process. Literature data are huge, especially for what concerns density and viscosity of the liquid mixture, as a function of temperature. Vapor phase data are less common, and usually these are given for pure substances. Mixtures are characterized by different vapor tensions and hence this becomes an issue for the purpose of cavitation modeling. For modeling purpose, saturation pressure at the given temperature has to be specified taking into account all the possible range of variability of hydrocarbons. Lighter molecules are more prone to evaporate than heavier fractions; hence average properties are probably not very useful in this context.

Biodiesel on the contrary is produced by transesterification of vegetable oil. Depending on its origin, composition can be quite different. Usually these fuels are characterized by higher molecular masses, higher viscosity and, most of all, by a restricted number of pure substances in the mixture. This feature can facilitate the job for selecting a representative saturation pressure.

Fuels evaluated in this work are listed in [Table 1](#). Fuel #1 is a typical European diesel for passenger cars. Fuel #2 is a methyl ester of soybean oil (SME). Fuel #3 is a D2 diesel

typical of the North American market. Fuels from 3.A to 3.D are “virtual fuels” that have been used to test the numerical model and to study separately the sensitivity of the calculation to each property.

Properties of interest for the multi-fluid model are densities and viscosities of both phases, in addition to vapor pressure at given temperature. All data have been evaluated at 40 °C.

Vapor pressure data are worthy of further analyses. [Figure 1](#) shows data for three typical hydrocarbons contained in diesel fuel compared to two pure Fatty Acid Methyl Esters (FAMES) contained in Soybean Methyl Ester. The two selected FAMES are practically the lightest and the heaviest compound found usually in SME [[31](#), [32](#), [33](#)]. Compounds found in diesel fuel show on the contrary a much wider range of variation and a much higher vapor pressure compared to biodiesel. Assuming that fuels behave as ideal solutions, Raoult's law could be used to determine the total vapor pressure of each mixture.

At the temperature of 40°C the light hydrocarbons contained in diesel fuel, such as decane, presents vapor pressure of approximately 800 Pa, while less volatile compounds have vapor pressure of 80 Pa approximately.

At the same temperature, if SME compounds are concerned, no experimental data are available. Nevertheless, Antoine equation can be drawn up to extremely low pressures (e.g. below 1 Pa), even if no experimental data are available to support this extrapolation.

In view of these data, in this work a conservative value of 10 Pa has been chosen for the biodiesel saturation pressure at 40 °C for cavitation modeling. For diesel fuel values from 500 to 1000 Pa seem realistic. Final selected values are listed in [Table 1](#).

Lastly, densities of vapors, as reported in [Table 1](#), have been chosen according to the compound molecular weight, taking account of the pressure and temperature conditions of the cavitating fluid.

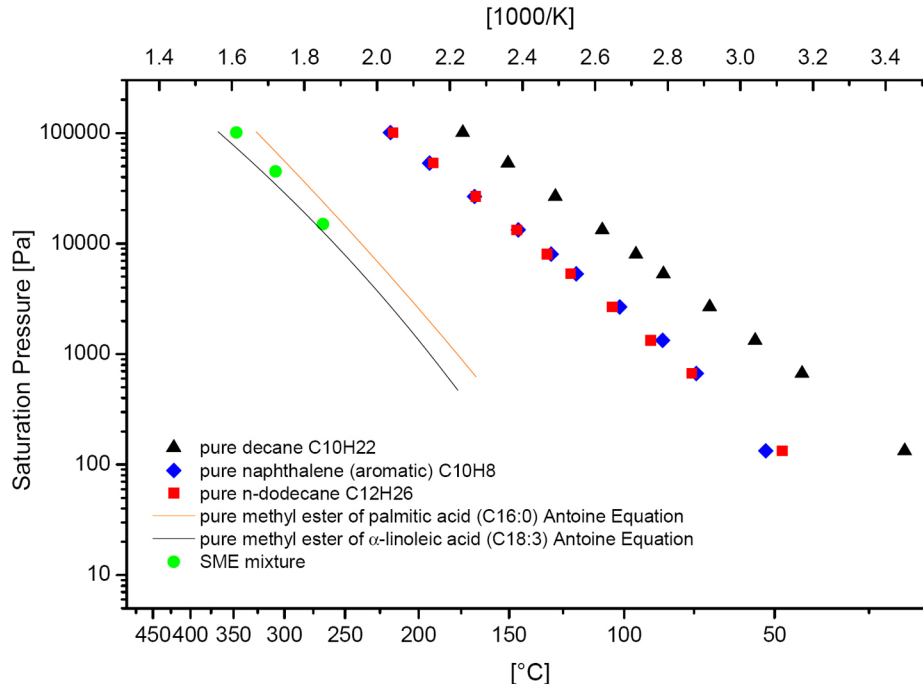


Figure 1. Vapor pressures of various pure hydrocarbons contained in diesel fuels and of pure FAMES contained in SME [30, 31, 32, 33].

TWO-FLUID MODEL ASSESSMENT IN SIMPLE CYLINDRICAL ORIFICES

A preliminary validation study has been carried out on the two-fluid model employing experimental data collected from literature [34, 35, 36, 37, 38]. Two experiments of cavitating flows through diesel nozzles have been modeled. One described in Soteriou et al. [34], here named H1, and one described in Chaves et al. [36], here named H2. Their geometrical characteristics are shown in Figure 2. Orifice H1 is characterized by a diameter D of 275 μm , a length of 1 mm, and cylindrical shape. Orifice H2 is characterized by $D = 200 \mu\text{m}$, $L = 1 \text{ mm}$, and cylindrical shape. In both cases, simulations have been carried out assuming axial symmetry of the flow. A very large outlet domain has also been included for achieving better convergence of the solution, accommodating the outlet fuel jet. All the simulations shown in this Section have been carried out using the linearized R-P equation and the monodispersed state approach, with an initial value for N_0''' set to $1.0\text{E}12 \text{ m}^{-3}$. The time step varied from $1.0\text{E}-8 \text{ s}$ to $5.0\text{E}-7 \text{ s}$, from inception to the regime solution.

In addition to the results provided in the works by Soteriou and Chaves [34, 36], theoretical and experimental correlations by Nurick, Lichtarowicz and V. K. Sarre [35, 37, 38] have been used to provide further benchmarks for the simulation results.

A summary of assessment cases presented in this paper is given in Table 2.

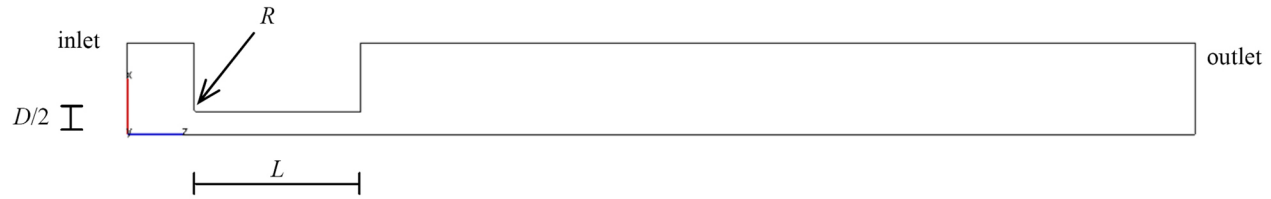
1. EFFECT OF ROUNDED INLET RADIUS - NOZZLE H1

The first model assessment test concerned the effect of inlet edge rounding. Details are presented in Table 2. Simulations were performed on hole H1 and the fuel #3, listed in Table 1. A large number of cases have been computed with CN ranging from 0.75 to 17. Back pressure has been fixed to 120 bar; in this way, for the selected CN values, inlet pressure ranges from 210 bar up to 2100 bar. The high value of the back pressure, being the pressure drops sufficiently high, guarantees that the flow regime is always turbulent, even at very low CN values.

Outlet chamber is filled with liquid fuel as well, thus the discharge type is of liquid in liquid.

Simulations here shown were performed by using the linearized form of the Rayleigh-Plesset equation, Eq. (5) and (7), for the cavitating mass exchange model and by considering the bubble diameter distribution as monodispersed.

Data reported in Figure 3 show the comparison between experimental measurements by Soteriou, for liquid in liquid discharges, and the results of the numerical simulations.



Hole	D [μm]	L [mm]	R/D	Fuel	Outlet region extension	Minimal mesh size	(1 layer – cake mesh)	
							Hole cells	Total cell number
H1	275	1	0	#3 (D2 diesel)	5 × L	5 μm	2700	8700
			0.03					
H2	200	1	0	#3 (D2 diesel)	5 × L	5 μm	2700	8700
						2 μm	8300	25000

Figure 2. Geometry and computational domain of the axisymmetric nozzles.

Table 2. Assessment cases for simple cylindrical orifices.

Test #	Hole	R/D	p_{in} [bar]	p_{out} [bar]	CN	Fuel	Cavitation model mass exchange
1. Inlet rounding radius	H1	0 0.03	210 -2100	120	0.75 - 17	#3	- linear R-P equation - mono-dispersed model; $N_0''' = 10^{12} \text{ m}^{-3}$
2. Mesh refinements	H2	0	120 -1260	60	1 - 20	#3	- linear R-P equation - mono-dispersed model; $N_0''' = 10^{12} \text{ m}^{-3}$
3. Fuel viscosity	H1	0	270, 1200	120	1.25 , 9	#3 #3.A #3.B	- linear R-P equation - mono-dispersed model; $N_0''' = 10^{12} \text{ m}^{-3}$
4. Fuel saturation pressure	H1	0	330, 1200	120	1.75 , 9	#3 #3.C	- linear R-P equation - mono-dispersed model; $N_0''' = 10^{12} \text{ m}^{-3}$
5. Liquid density	H1	0	210-2100	120	0.75 to 17	#3 #3.D	- linear R-P equation - mono-dispersed model; $N_0''' = 10^{12} \text{ m}^{-3}$

Discharge coefficient of the orifice is defined as

$$C_D = \frac{\dot{m}}{A_0 \sqrt{2\rho_l \Delta p}} \quad (18)$$

For low values of CN , below 1.5 - 2, discharge coefficient is almost constant. The flow is in pure liquid phase and the regime is fully turbulent, being Reynolds number far above 10,000 for all the points. Approaching the critical cavitation number, which in general is a function of the geometry, the flow starts to cavitate at the inlet edge and for a narrow range of CN values the flow reattaches before reaching the exit section. The discharge coefficient is still constant. Continuing to increase CN values, the vapor phase will reach the hole exit, then the effective area will be reduced and C_d will begin to decrease (supercavitation or full cavitation). Figure 4 shows three exemplary flow conditions: turbulent with reattachment before hole exit, supercavitation.

The magnitude of the turbulent discharge coefficient is well predicted by the model with a small rounded inlet. Moreover the decreasing trend of C_d in the supercavitating conditions is satisfactorily predicted as well. Actually, the model with sharp edge shows a slight underestimation of the C_d in all the points, but assuming at the inlet edge a radius of $0.03 \cdot D$ seems to improve the agreement between experimental and numerical data. This assumption seems permissible since it is recognized [10] that it is very difficult to manufacture an exactly defined inlet radius in real-size nozzles and in [34] no exact specification of the radius was provided. Furthermore, for turbulent non cavitating flows through cylindrical orifices, Lichtarowicz et al. [37] proposed the following correlation for sharp edges

$$C_d = 0.827 - 0.0085 \cdot L/D \quad (19)$$

which provides $C_d = 0.796$ in the current case, hole H1. If this is the case, model with $R = 0$ predicts reasonably well the turbulent regime.