Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 82 (2015) 51 – 58

Energy Procedia

ATI 2015 - 70th Conference of the ATI Engineering Association

# Numerical simulation of Diesel injector internal flow field

Ehsanallah Tahmasebi*, Tommaso Lucchini, Gianluca D'Errico, Angelo Onorati

*Internal Combustion Engine Group, Energy Department, Politecnico di Milano, Via Lambruschini 4, 20156 Milan, Italy*

**Abstract**

In Diesel engines, fuel-air mixing process and spray evolution drastically affect combustion efficiency and pollutant formation. Within this context, a detailed study of the effects of injector geometry and internal flow is of great importance to understand the effects of turbulence and cavitation on the liquid jet atomization process. To this end, both numerical and experimental tools are widely employed.

Objective of this work is to simulate the complex flow behavior inside the injector nozzle taking the most relevant physical phenomena into account. CFD simulations were carried out using a compressible solver with phase change modeling available in the OpenFOAM framework. In particular, cavitation was modeled by using an homogeneous equilibrium model based on a barotropic equation of state while the RANS k-ω SST model was used for turbulence. Experiments performed at Kobe University (Japan) on simplified nozzle geometries were used to validate the proposed approach in terms of velocity and vapor distributions. A rather good agreement between computed and experimental data was achieved in terms cavitation length, mass flow, momentum flux making possible to apply the proposed methodology also to real injector configurations in the near future.



Peer-review under responsibility of the Scientific Committee of ATI 2015

*Keywords:* Multiphase flow; Fuel injector; CFD analysis; Cavitation

## 1. Introduction

In modern Diesel engines, spray atomization is strongly influenced by the internal nozzle flow [1]. While in most of the engineering processes cavitation is undesirable according to its side effects, in fuel injection systems and particularly in modern diesel injectors such process inside injector hole can enhance jet turbulence improving the atomization process. Hence, detailed studies are needed to understand the complex interplay between nozzle design, fuel properties, turbulent flow and onset of cavitation.

However, observing the flow in diesel injectors is very difficult as they are characterized by high pressure injections through very small nozzles with diameters around 100 microns or less. One of the first comprehensive experimental studies on cavitation structures geometries was performed by Winklhofer et

* Corresponding author. Tel.: +39-3297527248;
*E-mail address:* ehsanallah.tahmasebi@polimi.it.


Peer-review under responsibility of the Scientific Committee of ATI 2015
doi:10.1016/j.egypro.2015.11.882

al. [2];other experiments have been performed in scaled-up nozzles [3]. However, it must be mentioned that cavitation behavior does not scale up and therefore actual-size experiments are needed to fully understand such physical phenomenon. Furthermore, since carrying out two-phase flow measurements is very complex, numerical simulations can be very useful to understand the main flow properties inside the the injector and at its exit section.

Purpose of this study is to apply a computational fluid dynamics approach to understand the cavitation and two phase flow behavior inside the injector nozzles. This approach is described in detail in this paper.

For compromising the results with experimental results, a specified geometry for visualizing the flow behavior inside the injector nozzles [4] is selected.

Simulations for several cases with different geometry, boundary conditions, and grid quality are done and results are compared with experimental results.

Despite the fluctuating behavior of the flow inside the cavitated injector [4], the results are comprisable with experimental results and their trend is acceptable. This validity shows that current approach can be used for simulation of the cavitation inside the fuel injectors as a robust, accurate and relatively fast approach.

**Nomenclature**

| | |
|---|---|
| $a$ | Speed of sound |
| $p$ | Pressure |
| $t$ | Time |
| $u$ | Velocity |
| Greek symbols | |
| $\gamma$ | Vapour fraction |
| $\mu$ | Fluid viscosity |
| $\rho$ | Fluid density |
| $\rho_{l\,sat}$ | Liquid density at saturation |
| $\rho_l^0$ | Liquid density at given temperature condition |
| $\rho_{v\,sat}$ | Vapour density at saturation |
| $\Psi$ | Fluid compressibility |
| Subscripts | |
| $l$ | Liquid |
| $sat$ | Saturation |
| $v$ | Vapour |

## 2. CFD modeling of injector internal flows with cavitation

Cavitation modeling could be performed considering two different approaches: two fluid flow models or continuum flow models. The first ones treat the liquid and vapor phases separately, whereas the continuum flow models consider the liquid and vapour as a homogeneous mixture. In these models, an equation of state which relates pressure and density (here barotropic equation) allows the calculation for cavitation growth[5]. Also, the role of turbulence in nozzle flow is essential and for this reason it has to be taken into account. In the current study, a Reynolds Averaged Navier Stokes multiphase compressible solver available in the OpenFOAM package was applied to study the cavitation development and flow behaviour inside the injector nozzle. Liquid and vapor are assumed to be perfectly mixed in each cell, and compressibility of the liquid and vapor phase is considered. The barotropic equation of state which relates pressureand density is used in this model:

$$\frac{D\rho}{Dt} = \Psi \frac{Dp}{Dt} \tag{1}$$

With $\Psi$ being the compressibility of the mixture, which is the inverse of the speed of sound squared:

$$\Psi = \frac{1}{a^2} \tag{2}$$

The equation of state should be consistent with the liquid and vapour equations of state when only one phase is present and also at intermediate states when there is a mixture of them. Both phases can be defined with a linear equation of state:

$$\rho_v = \Psi_v . p \tag{3}$$

$$\rho_l = \rho_l^0 + \Psi_l . p \tag{4}$$

To compute the amount of vapour in the mixture, the parameter $\gamma$ is used which is computed as:

$$\gamma = \frac{\rho - \rho_{lsat}}{\rho_{vsat} - \rho_{lsat}} \tag{5}$$

where:

$$\rho_{vsat} = \Psi_v . p_{sat} \tag{6}$$

It can be seen that in a flow without cavitation $\gamma = 0$, whereas $\gamma = 1$ for fully cavitated flow. In Eq. 6, $\Psi_v$ is the compressibility of the vapour. The density of the mixture is calculated taking into account the amount of vapour in the fluid ($\gamma$) together with a correction term based on the pressure (mixture's equilibrium equation of state):

$$\rho = \gamma . \rho_v + (1-\gamma) . \rho_l + \Psi (p - p_{sat}) = (1-\gamma) . \rho_l^0 + [(\gamma . \Psi_v + (1-\gamma) . \Psi_l) - \Psi] . p_{sat} + \Psi . p \tag{7}$$

with

$$\rho_l^0 = \rho_{l,sat} - \Psi_1 . p_{sat} \tag{8}$$

As far as the mixture's compressibility is concerned, it is modeled by a simple linear model:

$$\Psi = \gamma.\Psi_v + (1-\gamma).\Psi_l \tag{9}$$

with $\Psi_l$ equal to the compressibility of the liquid. Despite of this, there are models which describe the compressibility of the mixture in a more physical way, but a linear model is chosen in this work because of stability and convergence advantages. As done for compressibility, it is possible to obtain the viscosity of the mixture with a linear model:

$$\mu = \gamma.\mu_v + (1-\gamma).\mu_l \tag{10}$$

The methodology used by the solver starts solving the continuity equation for $\rho$:

$$\frac{\partial\rho}{\partial t} + \nabla(\rho.u) = 0 \tag{11}$$

The value of obtained $\rho$ is used to determine preliminary values for $\gamma$ and $\Psi$ by using equations (5) and (9), and also for solving momentum equation (Eq. 12) which is used to get the matrices used to calculate the velocity $u$:

$$\frac{\partial(\rho.u)}{\partial t} + \nabla(\rho.u.u) = -\nabla p + \nabla(\mu_f.\nabla u) \tag{12}$$

Convection terms in both mass and momentum conservation equations are discretized by using the Gauss theorem with an upwind scheme. This choice was done in order to ensure a stable simulation in presence of large pressure and density gradients, despite first-order schemes are known to introduce numerical diffusion increases when mesh resolution is reduced. Concerning diffusion terms, the non-orthogonal part of the gradient was included due to the relatively low mesh non-orthogonality for the configurations tested in this work [6].

An iterative PISO algorithm was used to solve for p and correct the velocity to achieve continuity. The equation solved within the PISO loop is the continuity equation transformed into a pressure equation by use of the equation of state (Eq. (7)):

$$\frac{\partial(\Psi.p)}{\partial t} - (\rho_l^0 + (\psi_l - \psi_v).p_{sat}).\frac{\partial\gamma}{\partial t} - p_{sat}.\frac{\partial\psi}{\partial t} + \nabla(\rho.u) = 0 \tag{13}$$

Once continuity has been reached, the properties $\rho$, $\gamma$ and $\psi$ will be update by means of Eqs. (7), (5), and (9) respectively which are taken into account to solve again momentum equation, and so, repeating the algorithm until convergence.

The time step is limited by both the Courant number and the acoustic Courant number, defined as:

$$Co = \max\left(\frac{|u|}{\Delta x}\right)\Delta t \tag{14}$$

$$Co_{acoustic} = \max\left(\frac{1}{\sqrt{\psi}\Delta x}\right)\Delta t \tag{15}$$

The chosen maximum Courant number is generally a compromise between results accuracy and computational cost. As shown in Eq. (14), the time step decreases at the Courant number decrease, so if this parameter is very small, computational cost can increase considerably. However, if the Courant number is sufficiently high, the accuracy of numerical results provided by the code can get worse. The same consideration is valid also for the acoustic Courant number [1], which is related to the propagation of pressure waves in compressible flows.

*2.1 Turbulence modeling in internal flows*

Due to the presence of solid boundaries, the flow behaviour and turbulence structure are considerable different from free turbulent flows. In particular, the Reynolds number is always very large. This implies that the inertia forces are extremely larger than the viscous forces at these scales. Menter [7] noticed that the results of the $k-\varepsilon$ model are much less sensitive to the arbitrary assumed values in the free stream, but its near- wall performance is unsatisfactory for boundary layers with adverse pressure gradients. This led to a hybrid model using (i) a transformation of the $k-\varepsilon$ model into a $k-\omega$ model in the near-wall region and (ii) the standard $k-\varepsilon$ model in the fully turbulent region far from the wall. Due to the performance of the $SST\ k-\omega$ model and its benefits for modeling internal flows with high Reynolds number and adverse pressure regimes, this model is selected for the current study. [8]

## 3. Experimental apparatus

In this work, experimental data from Pratama et al. [4]were chosen to validate the proposed numerical methodology. In [4], high speed images and Particle Image Velocimetry (PIV) technique were used to visualize cavitation and flow in simplified, two-dimensional nozzle configurations, aimed at reproducing multi-hole, mini sac nozzles as it is illustrated in Figure 1. Different layouts of the sac were tested by changing the main dimensions and the needle lift. In Figure 2 a schematic picture of experimental setup used in [4] is reported. Finally, Figure 3 reports the geometry which was used for the current study.

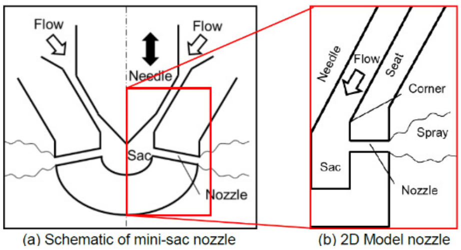


Fig.1. Mini-sac nozzle and current model nozzle [4]

The simulations have been performed for about 100 cases with different mesh qualities, boundary conditions, geometry, inlet pressure, and turbulence model.

Study on variety of inlet pressure allowed to analysis of the cavitation inception and development inside the injector and mass flow choking induced by cavitation inside the nozzle hole.

In following, a brief explanation of the equations will present. After introducing the geometry and simulations conditions, results will present. Compromising the result with experimental studies will present later.

## 4. Geometry and case setup

The geometry of the main case is shown in Figure 3. Also two other cases as shown in this figure are considered for study about effect of the upstream corner at the end of the seat and also distance between the corner and the nozzle. Table 1 summarizes the main geometry dimensions which are used in this study.Actually nozzle N2 is baseline and two other nozzles are selected in order to study on effect of hole distance from the sac inlet corner.

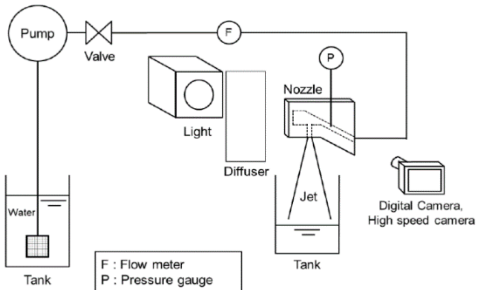


Fig. 2. Schematic of experimental apparatus [4]

Table 1. Dimensions of the nozzles

| Nozzle | L (mm) | W (mm) | S (mm) | $L_{SN}$ (mm) | S/W | $L_{SN}$/W |
|---|---|---|---|---|---|---|
| N1 | 16 | 4 | 4.0 | 4.0 | 1.0 | 1.0 |
| N2 | 16 | 4 | 4.0 | 8.0 | 1.0 | 2.0 |
| N3 | 16 | 4 | 4.0 | 12.0 | 1.0 | 3.0 |

A block-structure hexahedral mesh was employed in the simulation and it is shown in Figure 4. The mesh has 290,000 cells and this size was selected after a sensitivity analysis where the mesh resolution was varied from 19,000 up to 1,200,000 cells.

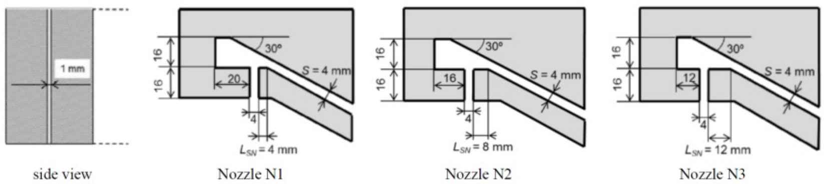


Fig. 3. Geometry of nozzles

### *4.1 Boundary conditions and fluid properties*

Suitable boundary conditions should be imposed to ensure the convergence and the accuracy of the simulations. With reference to Figure 4, total pressure boundary condition was set at inlet boundary with values ranging from 2 to 7 bar according to the simulate operating condition. Fixed pressure equal to 1 bar was fixed at the outlet. No-slip condition was imposed at the wall boundaries where wall functions were applied. Consistently with experiments, properties of water either in liquid and vapor phase were used.

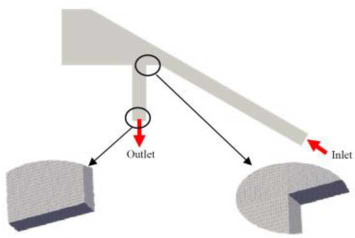

Fig. 4. Case setup and mesh

## 5. Results

Influence of inlet pressure on cavitation length was studied and to carry out a consistent comparison between computed and experimental data, it was necessary to compute the outlet mean velocity. In figure 5(a), fuel vapor distribution inside the nozzle hole for different values of the inlet pressure presented. These results are published in [4]. Figure 5(b) shows the same results which are achieved in current study by simulation. Because of the lack of knowledge about contrast of the experimental visualization in references, two different threshold values for $\gamma$ were used to visualize the cavitating flow inside the nozzle hole. Also for achieving the desired mean velocity at outlet section, inlet pressure is changed in different simulations.

Figure 5, shows that in nozzle throat, because of the reduction of the flow pressure, cavitation occurred which make a cavitation regime in nozzle hole. The length and width of this cavitation regime increases with increasing the velocity of the fluid inside the injector. The effect of these phenomena must be considered in design and analysis of fuel injectors.

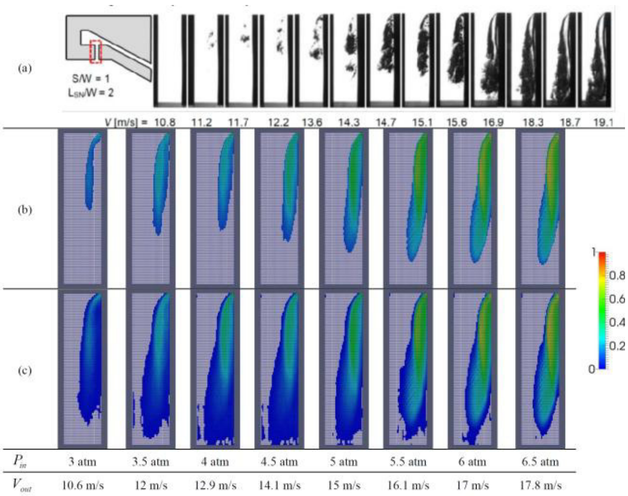


Fig 5. Cavitation in nozzle N2 nozzle hole, (a) Experimental results (b) Threshold 0.1 (c)Threshold 0.01

Finally, Figure 6 reports a comparison between computed and experimental data of cavitation length ratio ($L_c/L_{nozzle}$) as function of the mean outlet velocity which is related to inlet pressure. In simulations, cavitation length $L_c$ was computed as the farthest distance from the nozzle inlet where $\gamma = 0.1$ was found. As it seen in this figure, in low mean velocities, pressure inside the nozzle hole is totally more than saturation pressure and because of that, cavitation does not occurred inside the nozzle, but with increasing the flow velocity, effect of the flow rotation inside the solid corner and injector throat become dominant and reduce the fluid pressure up to saturation pressure which starts the cavitation inside the injector hole. Increasing of the cavitation length in higher velocitiescan reach the total length of the nozzle.

It must mention that, behavior of the flow in cavitated geometries, according to the behavior of the attached, semi attached and detached bubbles is fluctuating. This fluctuation is shown by error bars in experimental results in Figure 6. Considering this fluctuating behavior, Figure 6 shows that, simulation results are inside the experimental bandwidth.

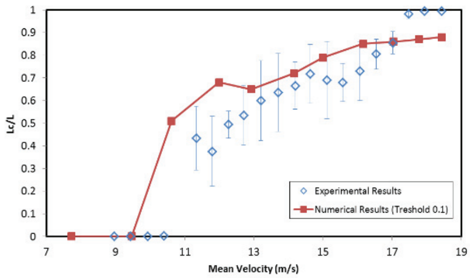


Fig. 6Cavitation length ratio versus mean velocity for nozzle N2

## 6. Conclusion

In this study, a methodology for modeling cavitation phenomena with considering the turbulence effect has been applied to a specific geometry and results are compared with experimental results. Results show that increasing the flow velocity inside the nozzle will increase the cavitation regime inside the hole. Study on velocity field near to the hole inlet corner shows that the results of the simulation are conforming the experimental results and theoretical estimations.

The simplicity and accuracy of this methodology make a valuable structure for simulation of the cavitation inside the diesel injectors. Also results for outlet section can be used for simulation of the fuel spray inside the engine.

## 7. References

[1] Salvador, F. J., J-V. Romero, M-D. Roselló, and J. Martínez-López. "Validation of a code for modeling cavitation phenomena in Diesel injector nozzles." Mathematical and Computer Modelling 52, no. 7 (2010): 1123-1132.

[2] Winklhofer, E., E. Kull, E. Kelz, and A. Morozov. "Comprehensive hydraulic and flow field documentation in model throttle experiments under cavitation conditions." In ILASS-Europe 2001, 17 International Conference on Liquid Atomization and Spray Systems. 2001.

[3] Chaves, H., and F. Obermeier. "Correlation between light absorption signals of cavitating nozzle flow within and outside of the hole of a transparent diesel injection nozzle." Proc. ILASS-EUROPE. Manchester, UK (1998): 6-8.

[4] Pratama R. H., Sou A., Wada Y., and Yohohata H., “Cavitation in mini-sac nozzle and injected liquid jet” In THIESEL 2014 Conference on thermo and fluid dynamic processes in direct injection engines. Valencia, Spain (2014).

[5] Salvador, F. J., Jorge Martínez-López, M. Caballer, and C. De Alfonso. "Study of the influence of the needle lift on the internal flow and cavitation phenomenon in diesel injector nozzles by CFD using RANS methods." Energy conversion and management 66 (2013): 246-256.

[6] Jasak, Hrvoje. "Error analysis and estimation for the finite volume method with applications to fluid flows." (1996).

[7] Menter, F. R. "Influence of freestream values on k-omega turbulence model predictions." AIAA journal 30, no. 6 (1992): 1657-1659.

[8] Versteeg, Henk Kaarle, and Weeratunge Malalasekera. An introduction to computational fluid dynamics: the finite volume method. Pearson Education, 2007.

**Biography**

Ehsanallah Tahmasebi, is a PhD scholar in Politecnico di Milano. He graduated from K.N.Toosi University of Technology, Tehran, Iran at 2009, in field of Aerospace Propulsion. Currently he is a member of Internal Combustion Engines group, Politecnico di Milano.