A CFD code for diesel direct injection simulation

Marcer R.

richard.marcer@principia.fr
PRINCIPIA R.D., Zone Athélia 1 – 13705 La Ciotat – France

In the recent years, the studies devoted to Diesel direct injection have considered the use of common rail injectors with injection pressures as high as 160 Mpa. One of the essential aspects of this type of injector is the occurrence of cavitation in each injection hole due to the fall of the static pressure at the hole entrance. The location of the cavitation which depends on the relative position of the holes and the needle, strongly affects the instantaneous values of the injection velocity, the Diesel fuel hydraulic section and the injection angle.

In the context of a GSM project (which involves RENAULT, PSA and IFP), PRINCIPIA has developed a numerical code (EOLE) for the simulation of Diesel sprays. It is based on a multiphase Navier-Stokes model using an improved VOF interface tracking method able to calculate the cavitation evolution within Diesel injectors, taking into account the mass transfer process at the liquid-vapour interfaces.
The model allows to describe numerically the onset and development of cavitation within Diesel injectors, as well as the spray break-up at the nozzle exit induced by cavitation.
The reliability of the code, based on numerous validations, allows us to use it for industrial applications.

1. Introduction

Progress in Diesel spray modelling highly depends on a better knowledge of the instantaneous injection velocity and the hydraulic section at the exit of each injection hole.
Spray calculations are currently based on a set of submodels for which the coefficients have been revisited due to the lack of representativity of the academic configurations for which these models have initially been developed. The tuning of these coefficients is generally biased due to the poor knowledge of the boundary conditions, notably for the injection velocity, in other terms the values of these coefficients incorporate the uncertainty of the boundary conditions in order to present acceptable comparisons between calculations and measurements.

Additionally a better identification of the mechanisms which cause break-up at high injection pressure and the implementation of submodels incorporating these mechanisms, is needed in order to reproduce correctly the tendencies observed in the engine. It follows from all of this that the knowledge of the boundary conditions and the exact link between the injector two-phase flow and the primary break-up phenomena is really necessary to progress seriously in the numerical description of Diesel sprays. For that aim, a multiphase Navier-Stokes model (code EOLE) based on an improved VOF type interface tracking method has been developed to simulate injector flows [1]. It allows to describe numerically the onset and development of cavitation within Diesel injectors, taking into account the mass transfer process at the interface of the cavity, as well as the spray break-up at the nozzle exit induced by cavitation [2].

2. The Navier-Stokes model EOLE

The unsteady 3D Navier-Stokes equations are written in curvilinear formulation for each phase, liquid and vapour.

The liquid flow is considered as compressible because of the high velocity and great pressure gradients existing in the injector cavitating flow. The density variation being related essentially to dynamic effect (and not thermal), the equation of state can be written as [2]:

$$Ln(\frac{\rho}{\rho_0}) = \int_{p_0}^{p} \frac{1}{\beta(p)} dP \tag{1}$$

where $1/\beta(p)$ is the isothermal compressibility coefficient.

Space discretization is based on a finite volume method expressed on multi-block curvilinear deforming grids, using a centered scheme with artificial viscosity [3]. Time discretization is ensured using a fully implicit second order scheme. The solution of the non-linear system for the unknown values at step $n+1$ is based on the pseudo-compressibility technique [4]. It consists in introducing a pseudo-unsteady term associated with a time-like variable τ called pseudo-time. The system is integrated step-by-step in pseudo-time until convergence towards a solution independent of τ which is then the numerical solution at time level $n+1$.
This method is particularly convenient to deal with multi-phase flows having phases with high density ratios.

3. The SL-VOF free-surface tracking method

The interface and its movement are obtained by an original method, called SL-VOF [1], using the two combined well known concepts of VOF [5] and PLIC (Peacewise Linear Interface Calculation) [6]. The interface is calculated in each cell thanks to a discrete function *C* whose value (*C* between 0 and 1) in each cell is the fraction of the cell occupied by the denser fluid (VOF concept). Depending on the flow regions, the fraction *1-C* of the cell volume is occupied by fuel vapour (cavitation) or gas in the combustion chamber.

The original SOLAVOF method [5], based on the resolution of a conservation equation of the VOF function, assumes the interface to be parallel to the grid faces, so the accuracy of this method is low. On the contrary, the SL-VOF method allows the interface to be represented by plans (or segment in 2D) of any orientation (PLIC concept) and due to its lagrangian nature, there is no need to solve a conservation equation of the VOF function. So, the main advantages of this method compared to the original VOF method are on the one hand the higher precision of the interface description and on the other hand the capacity to use larger time steps and then a significant gain in computational time.

The cavitation model is integrated in the VOF model in order to take into account the thermodynamical effects occurring during the vaporization and the condensation of the vapor. The model allows to compute a mass transfer term which will be introduced in the VOF scheme. Based upon the hypothesis that the thermal diffusion effects are neglegible inside the two-phase domain, one shows that the vapour or liquid source C_t can be written as :

$$C_t = \frac{dC}{dt} = -\frac{1}{A}(\frac{\partial p}{\partial t} + B\frac{dP}{dt} + \rho\frac{dE_c}{dt}) \tag{2}$$

where A and B are functions of the liquid VOF function, the pressure, the liquid and vapour densities and the velocity of sound in the two-phase domain, E_c the kinetic energy, P the pressure and ρ the liquid density.

4. Numerical results

The studied injector is a 5 sharp edge holes injector (figure 1). The characteristics of the holes are mentioned in table 1.

<table>
<tr><th></th><th>Angle / injector vertical axis</th><th>Diameter</th><th>Length</th></tr>
<tr><td>Holes 1 and 5</td><td>84°</td><td rowspan="3">190 µm</td><td rowspan="3">1000 µm</td></tr>
<tr><td>Holes 2 and 4</td><td>64°</td></tr>
<tr><td>Hole 3</td><td>52°</td></tr>
</table>

Table 1 : injector characteristics

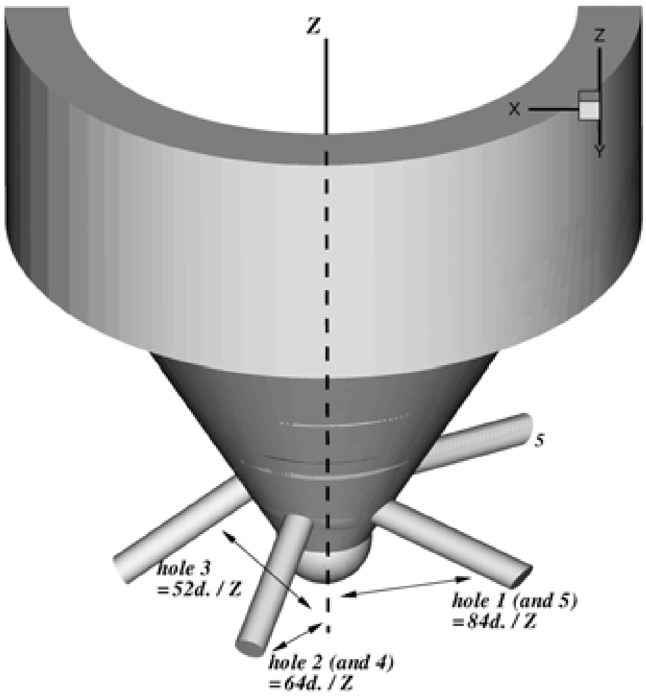


Fig. 1 : 5 holes injector

Due to the symmetry conditions, the calculations are done on half of the geometry. The mesh is of multi-domain type and the needle lift is taken into account with a technique of grid deformation.

The behaviour of the cavitating flow has been studied for two operating conditions defined as the following :

	Maximal needle lift	Opening duration	Pinj	Pch
Part load (PL) (1200 tr/min)	95 (µm)	660 (µs)	65 (Mpa)	5 (Mpa)
Full load (FL) (1200 tr/min)	200 (µm)	1340 (µs)	65 (Mpa)	5 (Mpa)

Table 2 : studied operating conditions

The cavitation number defined by (*Pinj-Pch*)/*Pch* is 12.

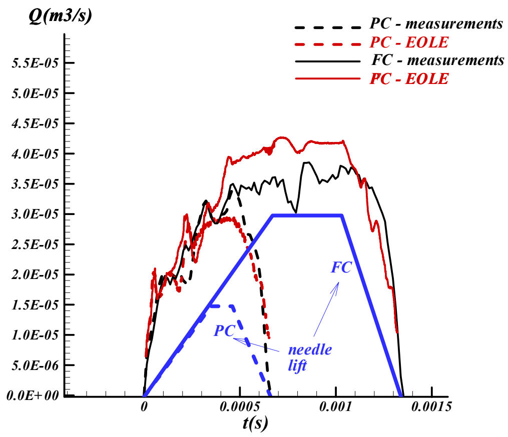


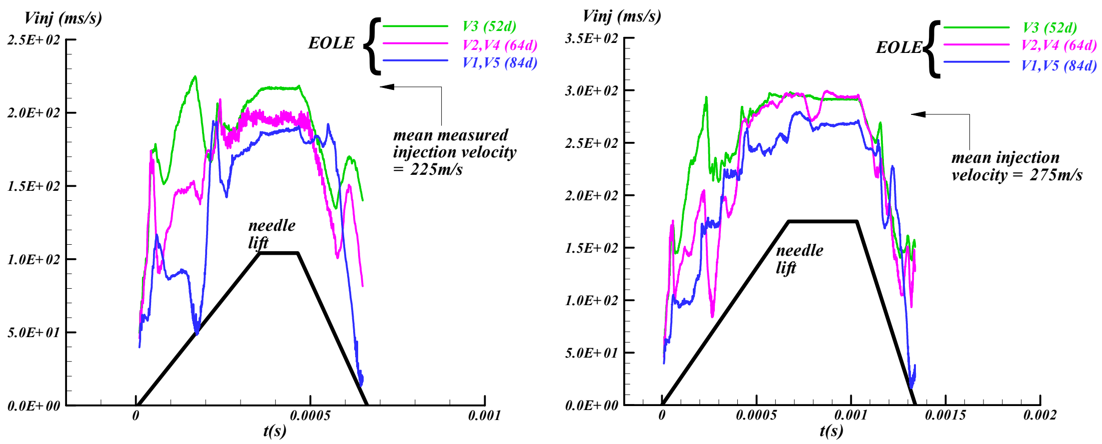
Fig. 2 : comparison of the global injector flow rate

Figure 2 shows a comparison between our simulations and measurements, carried out by IFP, of the injector flow rate. One can observe very similar tendencies for the flow rate during the needle lift evolution, particularly the pulsation of the flow rate due to the instabilities of the cavitation development.

The small differences appearing at full lift are certainly due to some experimental uncertainties of the maximal lift which is used for the calculations. Actually, a small variation of about 10µm of the lift induces a variation of more than 10% on the flow rate.

Injected mass (mg)		
	Part load	**Full load**
Hole 3 (52°)	2.75	7.68
Holes 2 and 4 (64°)	2.37	6.94
Holes 1 and 5 (84°)	2.04	6.39
Total (EOLE)	**11.57**	**34.34**
Measurements	**12.2**	**32.6**
Difference	-5.2%	+5.3%

Table 3 : comparisons of the injected mass



Part load Full load

Fig. 3 : injection velocity at the exit of each hole

The injection velocity (*Vinj*) is obtained by integration of the liquid phase velocity at the exit section of the hole.
The results show logically that the flow rate and the injection velocity are as much higher as the angle of the hole axis is small and then the losses of flow rate created by the sharp edge at the inlet of the hole are reduced.
As an indication, we give a "theoretical" value of the injection velocity deduced from the measurements, obtained by dividing the total flow rate by the number of holes and the geometrical section of the hole. So, the level of the "measured" injection velocity is quite similar to the calculated one in each hole.
For the full load, the higher lift tends to give closer performances on the holes presenting quite near angles (only 12° difference between hole 3 and holes 2 and 4).

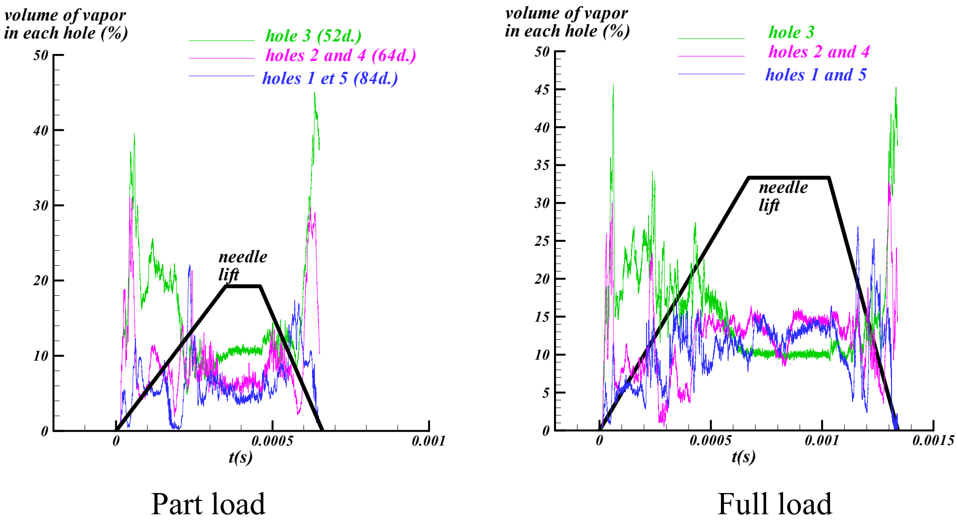
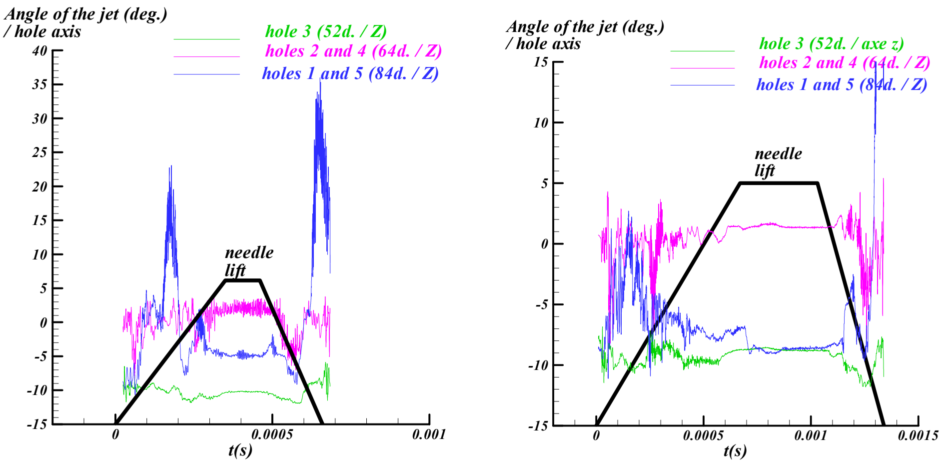


Fig. 4 : evolution of the vapour rate in the holes

Figure 4 shows the total amount of vapour generated within each hole. The fluctuations of the vapour volume in the holes are directly related to the unsteadiness of the cavitation process. The instabilities are particularly pronounced during the first part of the opening phase (which corresponds to the transient phase of cavitation development) and at the end of the closing phase which corresponds to a great acceleration of the liquid at the needle seat, losses of the pressure level and so cavitation appearance.
During the full lift, the cavitation becomes more stable and the mass transfer processes (vaporisation and condensation) have a smaller intensity. We can notice the effect of the maximum needle lift : the hole 3 is the one that cavitates the more for the part load and the less for the full load.



Part load Full load

Fig. 5 : deviation of the jet at the exit / axis of the hole

One defines the deviation of the jet on the exit section by the mean value of all the angles formed by the velocity vector on each cell with the hole axis (in the meridian plane).
Figure 5 shows that the deviation is very fluctuating with sharp peaks directed alternatively up and down (the negative value indicates a deviation under the hole axis).

Figure 6 shows a sequence of the cavitating flow evolution during one needle lift cycle. The cavitation develops as films from the holes entrance and breaks-up into small vapour structures which move towards the hole exit.
Cavitation processes are very instable and different from one hole to another due to the various orientations of these holes.
Cavitation appears also in the needle seat where the flow is very constrained.

Figure 7 shows an example of the velocity field on a 2D vertical plane near the entry of a hole (here hole 2). An unsteady and unsymmetrical swirl effect is evidenced. Theses instabilities of the flow are directly associated with the upstream chaotic development of the cavitation in the holes.

Theses results show a correct global coherence of the numerical results compared with visualisations of the cavitating flow in similar injectors, performed by the CRMT and the I.F.F. [1][7], and Arcoumanis et al. [8].

5. Conclusions

Some examples of calculations performed with the multiphase code EOLE based on an improved VOF method taking into account mass transfer process, evidence the great capacities of the model to simulate cavitation features in Diesel injectors.

The cavitation development and then the flow rate and the injection velocity depend on the hole orientation. The best hydraulic performances are obtained for the hole presenting the smallest deviation from the injector axis (hole 3).
Quantitative validations on experimental results were obtained for the mass flow rate and for the total injected mass.

Numerous other validations on measurements have been already carried out on parameters such as discharge coefficient, injection velocity, visualisations of the cavitation shape, influence of the inlet shape of the channel (straight entrance, rounded entrance), emission frequency of cavitation of cavities break-up of the film [1].
The reliability of the code allows us to use it for industrial applications.

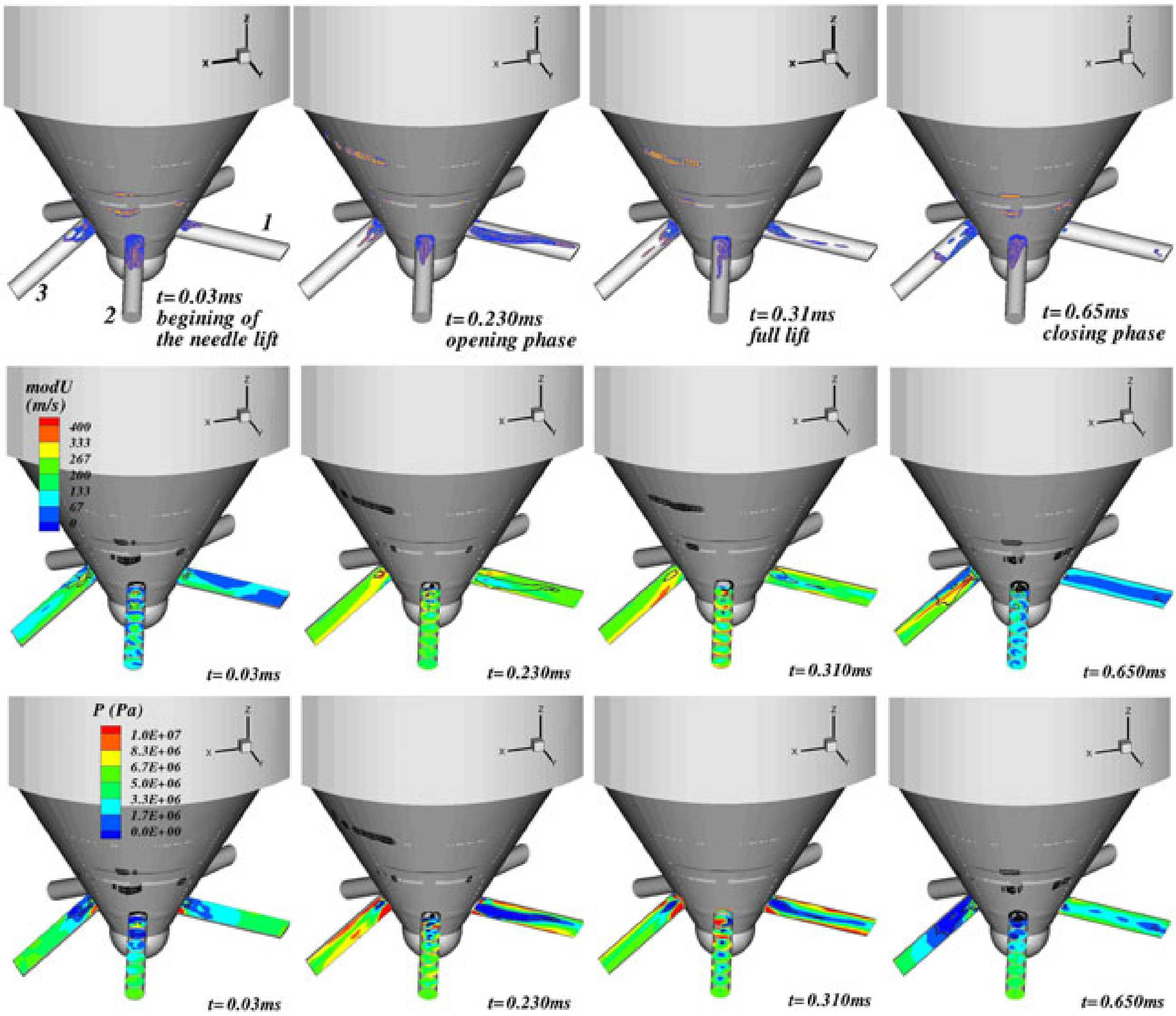


Fig. 6 : cavitation, velocity modulus (with cavitation in black) and pressure

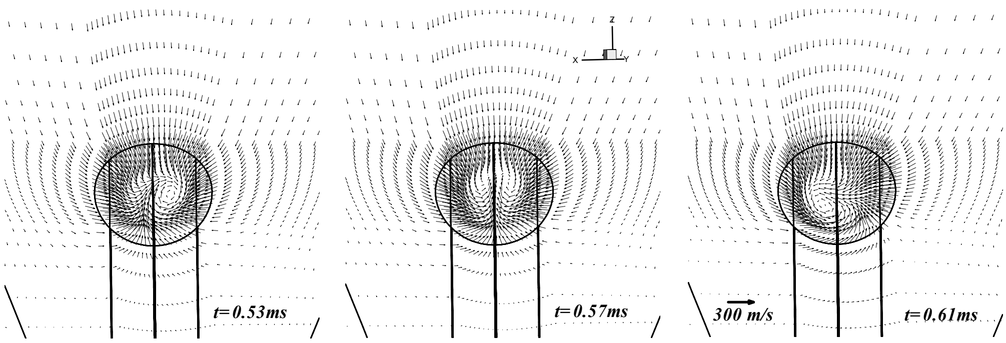


Fig. 7 : velocity field near the entry of the hole 2

Aknowledgements

This work was supported by the Groupement Scientifique Moteurs (Renault, PSA, IFP). The authors thank B. Argueyrolles and D. Postic (Renault), B. Barbeau (PSA), C. Habchi (IFP), P. Le Cottier (CRMT) and H. Chaves (IFF) for many fruitful discussions.

References

[1] Marcer R., Le Cottier P., Chaves H., Argueyrolles B., Habchi C., Barbeau B. "A Validated Numerical Simulation of Diesel Injector Flow Using a VOF Method", *SAE Paper* 2000-01-2932, 2000.
[2] Marcer R., LeGouez J.M., Simulation of unsteady cavitating flows in Diesel Injector with an improved VOF method, ILASS, Annual Conference on Liquid Atomization & Spray Systems, Zurich, September 2001.
[3] Jameson A., Schmidt A., Turkel E. "Numerical Solutions for the Euler Equations by finite volumes methods using Runge-Kutta Time-Marching Schemes", *AIAA 14 Th. Fluid and Plasma Dyn., Conf.* Palo Alto California, June 1995.
[4] Viviand H. "Analysis of pseudo-compressibility systems for compressible and incompressible flows", Computational Fluid Dynamics, Review 1995, Wiley publishers, Editors Halez Oshima, 1995.
[5] Hirt C.W., Nichols B.D "Volume Of Fluid (VOF) Method for the Dynamics of Free Boundaries", *J. of Comp. Physics*, Volumes 39, pages 201-255, 1981.
[6] Li J. "Piecewise Linear Interface Calculation", *C. R. Acad. Sci*. Paris, t. 320, Serie II b,pp 391-396., 1995.
[7] CRMT, private communication.
[8] Arcoumanis C., Badami M., Flora H. and Gavaises M. "Cavitation in real-size multi-hole Diesel injector nozzles", *SAE Paper* 2000-01-1249, 2000.