

## SPRAY INTERACTIONS: MODELING OF COLLISION/COALESCENCE PHENOMENA

Gautier Luret<sup>\*o</sup>, Grégory Blokkeel<sup>o</sup>, Romain Lebas<sup>1</sup>, Thibaut Ménard<sup>\*</sup>, Alain Berlemont<sup>\*</sup>, François-Xavier Demoulin<sup>\*</sup>

<sup>\*</sup> CNRS UMR 6614 CoRIA, Site Universitaire du Madrillet, BP12, 76801 St Etienne du Rouvray - France

<sup>o</sup> PSA Peugeot-Citroën, Research Department - Route de Gisy, 78943 Vélizy – France

<sup>1</sup>IFP, Avenue Bois-Préau, 92852 Rueil-Malmaison - France

### ABSTRACT

The primary breakup of the high-velocity Diesel jet is a difficult process to apprehend. The presence of a liquid core in the vicinity of the injector exit contributes extensively to the primary atomization and requires particular attention. The modeled spray behaviour depends strongly on the description of this region. The Eulerian-Lagrangian Spray Atomization (ELSA) model takes into account explicitly the dense part of the spray thanks to a transport equation of the liquid/gas interface density. Several source terms related to physical phenomena acting on the interface topology are necessary. This work is devoted to the improvement of the liquid-gas interface transport modelling of the ELSA model by means of a Direct Numerical Simulation. A two-phases flow confined into a cubic chamber with periodic conditions has been simulated for a prescribed turbulence level. During this work, a particular attention has concerned the contribution of turbulence effects on the liquid/gas interface development. For fixed conditions (turbulence and liquid volume fraction), an equilibrium of the liquid-gas interface can be reached. Such equilibrium can be characterized by a Weber number.

### INTRODUCTION

Energy and environment are becoming public policy issues, such as greenhouse effect and global warming. In response, drastic limitations of pollutant emissions are imposed to the automotive industry. The European car manufacturers will be subject to more stringent emissions regulation (such as Euro 5 and post-Euro 5 standards), concerning nitrogen oxides (Nox), unburned hydrocarbons (HC) and particle emissions. To reach these tolerance thresholds, the automotive research is looking for multiple processes controlling combustion: fuel distribution (injection), internal aerodynamics (vaporization, mixing) and combustion itself (ignition, chemical reactions).

Since many decades, a great attention has been devoted to the injection phase. It conditions the fuel distribution inside the combustion chamber, and so, contributes indirectly to the pollutants formation. Its optimization may notably lead to a cleaner Internal Combustion Engine (ICE). For instance, following the injector nozzle geometry [1] (Sarre *et al.*) and the injection strategy used [2] [3] (Su *et al.*, Lee *et al.*), the overall spray behaviour and its characteristics may be radically different. Thus, the process of liquid jet atomization has to be well understood. Unfortunately, during this step, several complex mechanisms, such as turbulence, primary and secondary breakup, droplet collisions and coalescence, act mutually. All these mechanisms have to be taken into account to determine the spray dispersion and locally the droplet diameter and velocity distributions.

Furthermore, one particularity of high-pressure Diesel jets is without doubt the presence of a liquid core attached to the exit of the nozzle. This zone, more commonly called dense zone, is the most difficult zone to model in particular due to the lack of experimental data. But it was shown that a realistic description of the dense part improves significantly the global modelling of

the injection [4] [5] [6] (Blokkeel *et al.*, Beau *et al.*, Lebas *et al.*). To deal with the dense part, the previous authors have adapted to Diesel injection the Eulerian Lagrangian Spray Atomization (ELSA) model originally proposed by Vallet *et al.* [7]. It is based on a single-phase Eulerian description of the flow that is composed of a liquid and a gas mixture. The initial dispersion and atomization of the liquid jet are supposed to be dominated by the turbulence taking into account the highly variable density. A transport equation for the mean liquid/gas interface density is also considered to describe the complex liquid topology. Indeed, initially the notion of droplet diameter is not applicable, since no droplet is formed yet. But to determine the different interactions between the liquid and gas phases, the quantity of interface is a first order parameter that has to be determined.

In this paper, the ELSA formalism will be presented, and the notions of the critical liquid-gas interface density and more particularly the critical Weber number on which is based this RANS model, will be put the emphasis on. Then, a Direct Numerical Simulation will be presented in order to evaluate more precisely this number.

### PRESENTATION OF THE EULERIAN-LAGRANGIAN SPRAY ATOMIZATION MODEL

#### Basics equation

First of all, the ELSA model considers liquid-gas flows as single-phase flows with a highly variable density  $\bar{\rho}$  which can be determined thanks to the following equation:

$$\frac{1}{\bar{\rho}} = \frac{\tilde{Y}_l}{\rho_l} + \frac{1 - \tilde{Y}_l}{\rho_g} \quad (1)$$

$\tilde{Y}_l$  corresponds to the mean liquid mass fraction which is transported thanks to the equation 3.  $\rho_g$  and  $\rho_l$  are respectively the gas and the liquid densities. The first one follows the state equation of a perfect gas (by taking into account the liquid volume fraction) and the second one takes into account the fact that the liquid density is modified according to the liquid temperature.

Based on the assumption that the Weber and Reynolds numbers have to be high, the Eulerian-Lagrangian Spray Atomization model is naturally well adapted to Direct Diesel Injection conditions. The assumption corresponds to an initial atomization dominated by aerodynamic forces.

This "mixture" approach has to be combined with a turbulence model, the  $(k-\varepsilon)$  model is generally used. The Boussinesq hypothesis is chosen to model the Reynolds Stress Tensor.

Considering the two-phase flow as a unique mixture flow with a highly variable density implies the fact that the transport equation for the mean velocity does not contain any momentum exchange terms between the liquid and the gas phases. Additionally the present form of the model is developed under the assumption of high values of both Reynolds and Weber number. Accordingly laminar viscosity and surface tension force can be neglected in the mean velocity evolution :

$$\frac{\partial \bar{\rho} \tilde{u}_i}{\partial t} + \frac{\partial \bar{\rho} \tilde{u}_i \tilde{u}_j}{\partial x_j} = - \frac{\partial \bar{P}}{\partial x_i} - \frac{\partial \bar{\rho} u_i'' u_j''}{\partial x_i} \quad (2)$$

The first specific transport equation of the ELSA model concerns the mean liquid mass fraction  $\tilde{Y}_l$ . A regular transport equation is used with a source term representing the effect of vaporization:

$$\frac{\partial \bar{\rho} \tilde{Y}_l}{\partial t} + \frac{\partial \bar{\rho} \tilde{Y}_l \tilde{u}_i}{\partial x_i} = - \frac{\partial \bar{\rho} u_i'' y_l''}{\partial x_i} - \bar{\rho} \dot{m}_{v,ELSA} \tilde{\Omega} \quad (3)$$

$\tilde{\Omega}$  and  $\dot{m}_{v,ELSA}$  represent the liquid-gas interface density per unit of mass and the vaporization rate per unit of mass respectively. This vaporization rate is deduced to the Abramzon and Sirignano's approach [8] [9]. The first term on the Right Hand Side (RHS) of the equation 3 is unclosed. This point needs to be discussed in details and the closure of this turbulent flux is presented below. This equation is of greatest importance because the dispersion of the liquid is represented by this way. A similar equation is used in order to model the vaporized fuel in the combustion chamber (see Lebas *et al.* [10]).

In order to close the equation 3, it is necessary to introduce a method to determine the quantity of surface of contact between the two phases. Classical approaches consist in considering spherical liquid drops and using the diameter but a more general notion than the droplet diameter has been used: the liquid-gas interface density per unit of mass. It enables to evaluate the vaporization rate per unit of surface and the heat transferred to the liquid phase. Initially proposed by Vallet *et al.* [7], its form is postulated from an analogy with the transport equation of flame surface density:

$$\begin{aligned} \frac{\partial \bar{\rho} \tilde{\Omega}}{\partial t} + \frac{\partial \bar{\rho} \tilde{u}_j \tilde{\Omega}}{\partial x_j} = & - \frac{\partial \bar{\rho} u_i'' \omega''}{\partial x_i} \\ & + \Psi (\phi_{init.} + \phi_{turb.}) \\ & + (1 - \Psi) (\phi_{coll./coal.} + \phi_{2ndBU} + \phi_{vapo.}) \end{aligned} \quad (4)$$

On the RHS of the equation 4, the first term is unclosed and as it is said before for the equivalent one for the mean liquid mass fraction, the closure of these two turbulent fluxes will be discussed below.

An equivalent diameter can be defined thanks to this liquid-gas interface density and the mean liquid mass fraction :

$$D_{32} = \frac{6 \tilde{Y}_l}{\rho_l \tilde{\Omega}} \quad (5)$$

Each source term  $\phi_i$  of the equation 4 models a phenomenon encountered by the liquid blobs or droplets.

$$\phi_{init.} = \frac{12 \bar{\rho} \mu_t}{\rho_l \rho_g Sc_t L_t} \frac{\partial \tilde{Y}_l}{\partial x_i} \frac{\partial \tilde{Y}_l}{\partial x_i} \quad (6)$$

$\phi_{init.}$  can be considered as an initialization term, taking high values near the injector nozzle, where the mass fraction gradients take its highest values. It corresponds to the minimum production of liquid-gas interface density necessarily induced by the mixing between the liquid and gas phases, see Beau *et al.* [5].

$$\phi_{turb.} = \frac{\bar{\rho} \tilde{\Omega}}{\tau_t} \left( 1 - \frac{\tilde{\Omega}}{\tilde{\Omega}_{crit.,1}} \right) \quad (7)$$

$\phi_{turb.}$  corresponds to the production/destruction of liquid gas interface density due to the turbulent flow stretching and the effects of collision and coalescence in the dense part of the spray. It is supposed to be driven by a turbulent time scale  $\tau_t$ . This production/destruction term is defined in order to reach, at the equilibrium between production and destruction of surface, a critical liquid-gas interface density  $\tilde{\Omega}_{crit.,1}$ . It corresponds to the quantity of surface of a stable blob under the considered flow conditions. Several formulations can be proposed, here a critical Weber number is considered  $We_{crit.,1} = 12$ :

$$\tilde{\Omega}_{crit.,1} = \frac{12 \bar{\rho} k \tilde{Y}_l}{\rho_l \sigma_l We_{crit.,1}} \quad (8)$$

$\sigma_l$  corresponds to the surface tension of the liquid phase.

$$\phi_{coll./coal.} = \frac{\bar{\rho} \tilde{\Omega}}{\tau_{coll}} \left( 1 - \frac{\tilde{\Omega}}{\tilde{\Omega}_{crit.,2}} \right) \quad (9)$$

$\phi_{coll./coal.}$  models the production/destruction of liquid-gas interface density due to the effects of collision and coalescence in the dilute spray region. It is supposed to be driven by a collision time scale  $\tau_{coll}$  initially proposed by Iyer

*et al.* [11] [12] [13]. This production/destruction term is defined in order to reach, at the equilibrium between production and destruction of surface, a critical liquid-gas interface density  $\tilde{\Omega}_{crit,2}$  which corresponds to a critical diameter  $D_{crit,2}$  (see equation 5). Several formulations can be done, here the model initially proposed by Beau *et al.* [5] is chosen:

$$D_{crit,2} = D_{32} \frac{1 + \frac{We_{crit,2}}{6}}{1 + \frac{We_{coll}}{6}} \quad (10)$$

with  $We_{crit,2} = 12$  and the collisional Weber number  $We_{coll}$  is defined thanks to the turbulent kinetic energy  $\tilde{k}$  as a velocity reference to characterize the collision/coalescence regime:

$$We_{coll} = \frac{\rho_l \tilde{k} D_{32}}{\sigma_l} \quad (11)$$

$$\phi_{2ndBU} = \text{Max} \left[ \frac{\bar{\rho} \tilde{\Omega}}{\tau_{2ndBU}} \left( 1 - \frac{\tilde{\Omega}}{\tilde{\Omega}_{crit,3}} \right); 0 \right] \quad (12)$$

$\phi_{2ndBU}$  deals with the production of liquid-gas interface density due to the effects of secondary breakup in the dilute spray region. This source term comes from the model proposed by Pilch and Erdman [14]. It enables to estimate the breakup time scale  $\tau_{2ndBU}$  according to the Weber number of the gas phase  $We_g$ , thanks to empirical correlations. Moreover, it determines the critical Weber number  $We_{crit,3}$  via the following relation:

$$We_{crit,3} = 12 \left( 1 + 1.077 Oh^{1.6} \right) \quad (13)$$

$Oh$  is the Ohnesorge number. The critical liquid-gas interface density of the secondary breakup regime can then be obtained:

$$\tilde{\Omega}_{crit,3} = \frac{6 \rho_l u_{rel}^2 \tilde{Y}_l}{\rho_l \sigma_l We_{crit,3}} \quad (14)$$

with  $u_{rel}$  the relative velocity between the liquid and gas phases.

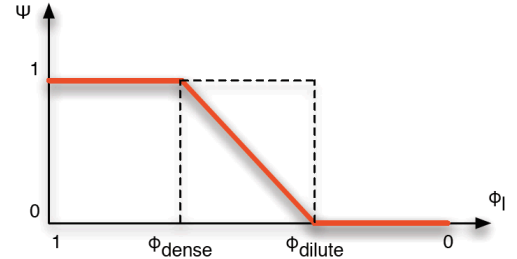
The ‘‘Max’’ function is due to the fact that in the secondary breakup regime, an increase of the droplet diameter is impossible. So, only positive values of this source term are taken into account.

$$\phi_{coll./coal.} = -\frac{2}{3} \frac{\bar{\rho} \tilde{\Omega}^2}{\tilde{Y}_l} \dot{m}_{v,ELSA} \quad (15)$$

$\phi_{vapo.}$  comes from a classical adaptation of the ‘‘ $D^2$ ’’ law of vaporization models for droplets and deals with the effects of destruction of liquid-gas interface density due to vaporization.

The transport equation for  $\tilde{\Omega}$  takes into account, via source terms, several physical phenomena encountered by the liquid phase. Some of them are specifically observed in the dense zone of spray. On the other hand, some of them are dedicated

to dilute spray regions. The idea was to introduce a function which could be used as an indicator of the dense or dilute region in the spray. A balance function  $\Psi$  has been chosen in order to return a value of 1 in the dense region and a value of 0 in the dilute region. The transition between these two cases is made via a linear regression, as it can be seen in figure 1.



**Figure 1: Indicator function  $\Psi$  proposed to balance the effects of the source terms of the equation 6 between dense and dilute regions of the spray.**

$\phi_{dense}$  and  $\phi_{dilute}$  correspond to specific values of the liquid volume fraction  $\phi_l$  obtained thanks to the following relation:

$$\phi_l = \frac{\bar{\rho} \tilde{Y}_l}{\rho_l} \quad (16)$$

They can be considered as numerical parameters, but during all this study they have been fixed to:  $\phi_{dense} = 0.5$  and  $\phi_{dilute} = 0.1$ .

### The turbulent flux closure

The equations 3 and 4 require a closure of the turbulent flux (first term on the RHS). It is important to notice that this term is exactly related to the drift velocity. Classically, a first order closure based on a gradient law is chosen (the Schmidt number equals to 1 for both the mean liquid mass fraction and the liquid-gas interface density):

$$\begin{aligned} \frac{\partial \bar{\rho} u_i'' x''}{\partial x_i} &= - \frac{\partial \bar{\rho} \tilde{X} (\bar{u}_{j,x} - \tilde{u}_j)}{\partial x_j} \\ &\approx - \frac{\partial}{\partial x_j} \left( \frac{\mu_t}{Sc_t} \frac{\partial \tilde{X}}{\partial x_j} \right) \end{aligned} \quad (17)$$

The classical gradient law is probably not able to represent all the phenomena involved in the dilute spray region. Two possibilities are available:

- a second order closure with a transport equation for the turbulent flux leading to a quasi-multiphase approach [15], Beau *et al.*;
- a hybrid method that couples eulerian and lagrangian descriptions of the spray, Lebas *et al.* [16].

The ELSA model itself takes into account this hybrid method that enables to benefit of both euler and lagrange formulations in order to close these turbulent fluxes. For more details concerning the hybrid method [16], see Lebas *et al.*

## Equilibrium or critical state: Weber number

As previously seen, for each physical phenomenon for which the aerodynamic and the surface forces are in competition, such as turbulence, collision, coalescence and secondary breakup, a source term in the equation 4 can be formulated in function of a critical liquid-gas interface density:

$$\phi_i = \frac{\bar{\rho}\tilde{\Omega}}{\tau_i} \left( 1 - \frac{\tilde{\Omega}}{\tilde{\Omega}_{crit.,i}} \right) \quad (18)$$

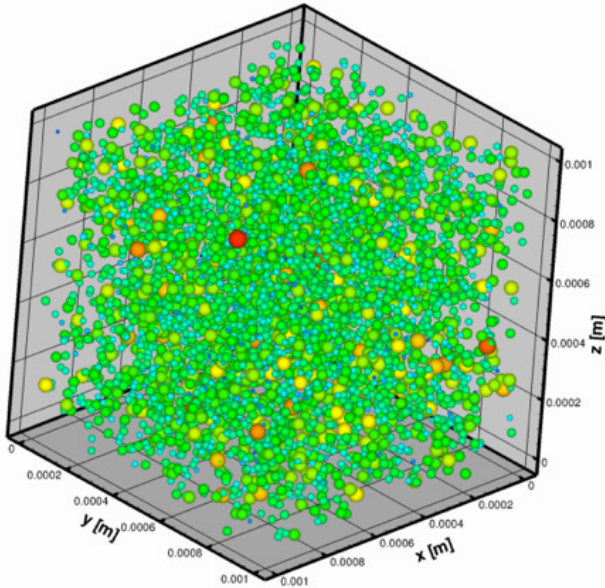
These critical liquid-gas interface densities depend on a critical Weber number, which can be derived, assuming at the equilibrium the liquid kinetic energy balances the liquid surface energy.

$$We_{crit} = \frac{1/2 m_l V_l^2}{\sigma_l S_l} = 1 \quad (19)$$

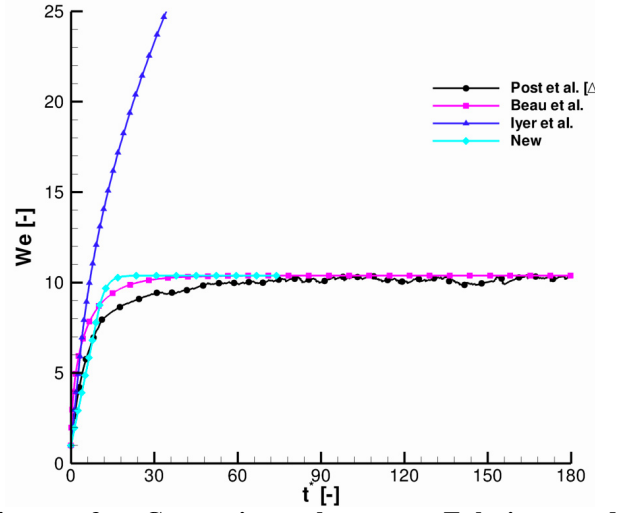
From the Eulerian characteristics (liquid volume fraction, liquid-gas interface density), it writes:

$$We_{crit} = \frac{\rho_l V_l^2 D_{32}}{\sigma_l} = 12 \quad (20)$$

The characterization of the Weber number, concerning the collision and coalescence processes, was investigated by Luret *et al.* [17]. Indeed, an equilibrium state between coalescence and collision-induced breakup regimes has been identified and may be described by a critical Weber number. Several Lagrangian collision submodels have been used in a simple test case (see figure 2) and compared with Eulerian collision source terms (for example, equation 9 and see figure 3).



**Figure 2: Test case: Cubic cell with periodic conditions in which Lagrangian particles were injected with prescribed Eulerian characteristics. (see Luret *et al.* [17])**



**Figure 3: Comparison between Eulerian and Lagrangian collision models. Existence of an equilibrium state.**

A common drawback of all collision models is the assumption that collisions occur between spherical droplets at rest. But after a collision issued droplets are very perturbed and they are animated of a strong interna motion. In many cases, the time needed to dissipate this internal motion can be longer that the collisional time. Hence internal agitation of the droplet has to be considered for the next collision. The idea of our work is to go on the investigation about the equilibrium state and its characterization through Weber number for denser medium and by incorporating the effect of internal agitation. For dense region of the spray, the Lagrangian hypothesis (isolated and spherical droplets) is not respected anymore. Thus, using Direct Numerical Simulation (DNS) has been a natural choice. The next section will concern the DNS code used and developed by Ménard *et al.* [18] and will be followed by the first results, in a diluted medium.

## DIRECT NUMERICAL SIMULATION: PRESENTATION OF THE CODE

Direct Numerical Simulation can be a powerful tool to study, for example, the primary break up of a liquid jet (see Ménard *et al.* [18]). Thus, from DNS simulations, statistical information can be collected in the dense zone of the spray where nearly no experimental data are available. Furthermore, these simulations are predictives and quantitatives. They permit to validate the ELSA model for the atomization process of Diesel-like jet (see Lebas *et al.* [10]). It is interesting to use DNS tool to develop and affine models, for example. This is the first objective of work presented below by focusing on the liquid-gas interface density, but with a finer description of turbulence effects on the development of the interface thanks to the DNS tool.

Thus, the numerical method should describe the interface motion precisely, handle jump conditions at the interface without artificial smoothing, and respect mass conservation. Accordingly a 3D code was developed by Ménard *et al.*, where interface tracking is performed by a Level Set method, the Ghost Fluid Method is used to capture accurately sharp discontinuities, and the Level Set and VOF methods are coupled to ensure mass conservation. A projection method is

used to solve incompressible Navier-Stokes equations that are coupled to a transport equation for level set and VOF functions.

### Interface tracking method

Level Set methods are based on the transport of a continuous function  $\phi$  which describes the interface between two mediums [19] [20] (Sussman *et al.*, Sethian *et al.*). That function is defined by the algebraic distance between any point of the domain and the interface. The interface is thus described by the 0 level of the level set function. Solving a convection equation determines the evolution of the interface in a given velocity field  $\mathbf{V}$  [20].

$$\frac{\partial \phi}{\partial t} + \mathbf{V} \cdot \nabla \phi = 0 \quad (21)$$

Particular attention must be paid to this transport equation. Some problems may arise when the level set method is developed: a high velocity gradient can produce wide spreading and stretching of the level sets, such that  $\phi$  will no longer remain a distance function. A re-distancing algorithm [19] (Sussman *et al.*) is then applied to keep  $\phi$  as the algebraic distance to the interface.

To avoid singularities in the distance function field, a 5<sup>th</sup> order WENO scheme can be used for convective terms [21] (Jiang and Shu). Temporal derivatives are computed with at least a third order Runge Kutta scheme.

One advantage of the level set method is its ability to represent topological changes both in 2D or 3D geometry quite naturally. Moreover, geometrical information on the interface, such as normal vector  $\mathbf{n}$  or curvature  $\kappa$ , are easily obtained through:

$$\mathbf{n} = \frac{\nabla \phi}{|\nabla \phi|} \quad \kappa(\phi) = \nabla \cdot \mathbf{n} \quad (22)$$

It is well known that numerical computation of the equation (21) and redistance algorithm can generate mass loss in under-resolved regions. This is the main drawback of level set methods, but to improve mass conservation two main extensions of the method can be developed, namely the particle level set [22] (Enright *et al.*) and a coupling between VOF and Level Set [23] (Sussman and Puckett).

### Navier Stokes equations

The Level Set method is coupled with a projection method for the direct numerical simulation of incompressible Navier-Stokes equations expressed as follows:

$$\frac{\partial \mathbf{V}}{\partial t} + (\mathbf{V} \cdot \nabla) \mathbf{V} + \frac{\nabla p}{\rho(\phi)} = \frac{\nabla(2\mu(\phi)\mathbf{D})}{\rho(\phi)} \quad \mathbf{D} = \frac{1}{2}(\nabla \mathbf{V} + \nabla \mathbf{V}^T)$$

$$\nabla \cdot \mathbf{V} = 0$$

where  $p$  is the pressure,  $\rho$  and  $\mu$  are the fluid density and viscosity respectively.

Spatial derivatives are estimated with a 2<sup>nd</sup> order central scheme, but convective terms are also approximated by 5<sup>th</sup> order WENO scheme discretization in order to ensure a robust

behavior of the solution. Temporal derivatives are approximated with an Adams Bashforth algorithm.

Poisson equation discretization, with a second order central scheme, leads to a linear system; the system matrix is symmetric and positive definite. Different methods can be derived to solve that system. According to different authors [24] (Tatebe), the MultiGrid method for preconditioning Conjugate Gradient methods (MGCG) combines Incomplete Choleski Conjugate Gradient (ICCG) robustness with the multigrid fast convergence rate. The MGCG method greatly decreased computational time compared to the ICCG algorithm.

### Discontinuities

The interface is defined by two different phases and discontinuities must be taken into account for density, viscosity and pressure. Specific treatment is thus needed to describe the jump conditions numerically.

The pressure jump related to surface tension  $\sigma$  and to the interface curvature reads [25] (Tanguy) :

$$[p] = \sigma \kappa(\phi) + 2[\mu] (\nabla u \cdot \mathbf{n}, \nabla v \cdot \mathbf{n}, \nabla w \cdot \mathbf{n}) \cdot \mathbf{n} \quad (3)$$

where  $[.]$  represents a jump in the considered variable.

Two different approaches can be used to represent the above conditions, namely the Continuum Surface Force (delta formulation) or the Ghost Fluid Method.

To overcome smoothing of the CSF method, the Ghost Fluid Method (GFM) has been developed by Fedkiw *et al.* [26]. The formalism respects jump discontinuities across the interface, and avoids an interface thickness. Discretization of discontinuous variables is more accurate, and spurious currents in the velocity field are thus much lower than with CSF methods. This procedure is used to discretize all discontinuous variables, namely density, viscosity, pressure and viscous tensors [25] [27] (Tanguy, Kang *et al.*).

In the GFM methods, ghost cells are defined on each side of the interface [27] [28] (Kang *et al.*, Liu *et al.*) and appropriate schemes are applied for jump conditions. As defined above, the interface is characterized through the distance function, and jump conditions are extrapolated on some nodes on each side of the interface. Following the jump conditions, the discontinued functions are extended continuously and then derivatives are estimated.

More details can be found in [28] (Liu *et al.*) on implementing the Ghost Fluid Method to solve the Poisson equation with discontinuous coefficients and obtain solution with jump condition. The method has been used to compute two-phase flows, and it compares favorably with the Continuum Surface Method in both 2D and 3D cases. It is clearly shown that spurious currents are  $10^2$  times lower with GFM than with CSF for a static drop [25] (Tanguy).

### Level Set-VOF coupling

A lot of liquid parcels (droplets, ligaments, liquid sheets...) are generated in the primary break up of a jet and the coupling between the VOF and the Level Set methods was necessary. The numerical method used here is quite similar to the CLSVOF of Sussman and Puckett, whose details are described in [23] (Sussman and Puckett) and [18] (Ménard *et al.*). The main differences with the CLSVOF are that the initial redistance algorithm was kept in our approach, and the

reconstruction technique was modified to define the interface in a cell from the Level Set position.

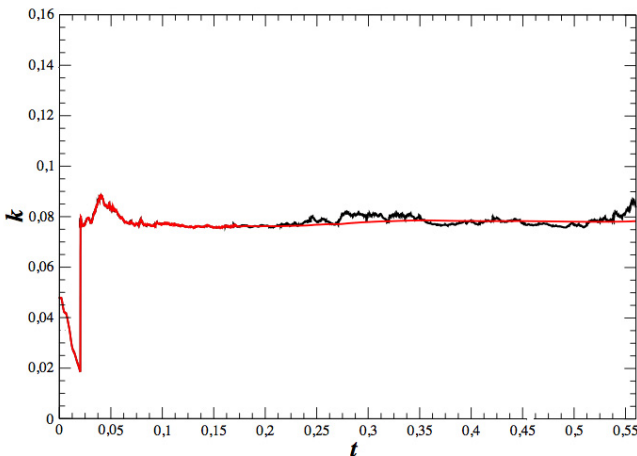
## TEST CASE AND RESULTS

Adapted to the Direct Numerical Simulation algorithm, the simulation of this study uses the general idea of those above-cited with Lagrangian method [17] (Luret *et al.*). A two-phases mixture, defined by a prescribed liquid volume fraction, is confined into a cubic chamber with periodic conditions. A linear forcing inspired from Rosales and Meneveau [29], initially used for the gas phase is applied here for the liquid-gas mixture. Thanks to this forcing, the turbulent kinetic energy reaches a targetted level and the two-phases flow is isotropic and homogeneous in average. The linear forcing consists in adding one term related to the velocity fluctuations multiplied by a constant, to the Navier-Stokes equations. This constant is continuously determined during the run in such way to keep constant the turbulent kinetic energy. The figure 4 represents the temporal evolution of the turbulent kinetic energy. Initially, the linear forcing is unactive to initiate the turbulent field and to avoid numerical instabilities. During the phase starting from instable condition, the flow becomes turbulent and relaxes gently. This avoids strong inaccuracy between the velocity field and the liquid volume fraction field. Such inaccuracies were at first the cause of the creation of liquid droplets smaller than mesh size. This phase without forcing is the reason for which the turbulent kinetic energy decreases at the beginning of the simulation before reaching its prescribed level.

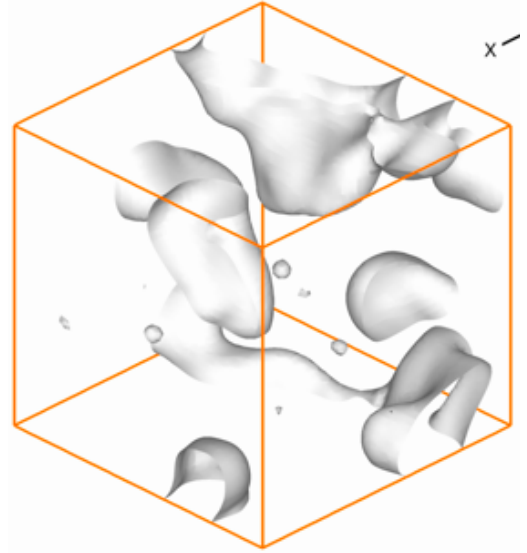
The characteristics of the simulation are summarized in the table 1.

**Table 1: Test case characteristics.**

Domain sizes ( $m^3$ )	0.01 <sup>3</sup>
Grid	64 <sup>3</sup>
Target turbulent kinetic energy ( $m^2.s^{-2}$ )	0.07
Liquid volume fraction	0.05
Gas density ( $kg.m^{-3}$ )	50
Liquid density ( $kg.m^{-3}$ )	1000
Liquid surface tension ( $N.m^{-1}$ )	0.08



**Figure 4: Temporal evolution of the turbulent kinetic energy.**



**Figure 5: Iso-surface 0 of the Level Set function at a given time.**

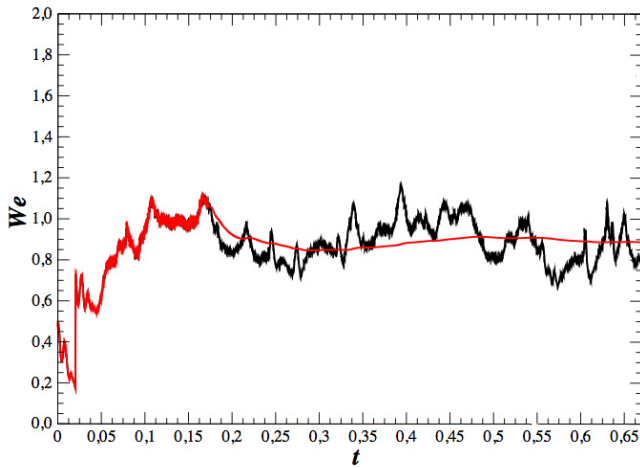
The figure 5 represents a snapshot of the liquid-gas interface at given time. We can distinguish different liquid structures, more or less tortuous, which interact with themselves and with the gaseous phase. With the conditions of this run, one can see that the droplets between each collision are so perturbed by their internal energy that it is even difficult to speak of droplets.

Accordingly in contrary to resorting to the classical Lagrangian approach (see [17], Luret *et al.*), here, we don't reason in terms of droplet mean diameter. But we have directly captured information about the liquid-gas interface, through the Level Set function. It is possible to express the previous Weber number from the liquid-gas interface density per unit of volume  $\Sigma$ .

The Weber number is defined as the ratio between kinetic and surface energies (equation 23).

$$We = \frac{\bar{\rho}k}{\sigma\Sigma} \quad (23)$$

The figure 6 represents the temporal evolution of this Weber number. Thus, after that the turbulent kinetic energy is reached a quasi-stable level, the Weber number seems to oscillate around a value, considered as an equilibrium state, close to the value of 1. This implies that the total available energy is distributed equally between kinetic energy and surface energy, for these flow conditions. This result is quite simple but this cannot be a direct rule. Indeed the amount of energy that can be stored by the surface must decrease with the liquid volume fraction. This is the reason why it is necessary to go on this investigation for various conditions (liquid volume fraction and target turbulent kinetic energy for example).



**Figure 6: Temporal evolution of Weber number. The crude signal corresponds to the black line. For the red line, the Weber number is averaged from the time for which an equilibrium state is considered.**

## CONCLUSION

The main aim of this work has been to find a configuration for the Direct Numerical Simulation code in order to study, in the soon future, the dense two-phase flow, and more particularly, to observe the behaviour of the liquid-gas interface due to effects of turbulence, for different turbulence levels and medium concentration.

Using the Direct Numerical Simulation lets foresee good future prospects in order to, on the one hand, develop and to improve the Eulerian-Lagrangian Spray Atomization model, and the other hand, to better understand the turbulent two-phases flow (liquid and gas), the turbulence effects on the topology of the liquid-gas interface and its development, the turbulence distribution between the liquid and the gas phases, and so on. Numerous simulations with various conditions (volume liquid fraction and target turbulent kinetic energy) are in progress. With this kind of numerical tool, to explore denser medium, where the liquid volume fraction is close to 1, is certainly one of perspectives most interesting.

## ACKNOWLEDGMENT

This work is financially supported by PSA Peugeot Citroën. All the authors would like to thank them for their continuous support and their confidence.

## NOMENCLATURE

Symbol	Quantity	SI Unit
<b>Latin</b>		
$D$	Diameter	m
$k$	Turbulent kinetic energy	$\text{m}^2.\text{s}^{-2}$
$Oh$	Ohnesorge number	-
$P$	Pression	Pa
$Sc$	Schmidt number	-
$t$	Time	s
$u$	Velocity	$\text{m}.\text{s}^{-1}$
$We$	Weber number	-
$Y$	Mass fraction	-

## Greek

$\varepsilon$	Dissipate rate	$\text{m}^2.\text{s}^{-3}$
$\phi$	Source term / Level Set function	-
$\kappa$	Curvature	
$\mu$	Dynamic viscosity	$\text{kg}.\text{m}^{-1}.\text{s}^{-1}$
$\rho$	Density	$\text{kg}.\text{m}^{-3}$
$\sigma$	Surface tension	$\text{kg}.\text{s}^{-2}$
$\tau$	Characteristic time scale	s
$\Sigma$	liquid-gas interface density per unit of volume	$\text{m}^{-1}$
$\Omega$	liquid-gas interface density per unit of mass	$\text{m}^2.\text{kg}^{-1}$

## Subscripts

$coll$	Collision
$crit$	Critical
$g$	Gas
$l$	Liquid
$rel$	Relative
$t$	Turbulent

## REFERENCES

- [1] C.v.K. Sarre, S.-C. Kong and R.D. Reitz, Modeling the effects of injector nozzle geometry on diesel sprays, *SAE Technical Paper*, 1999-01-0912, 1999.
- [2] T.F. Su, M.A. Patterson, R.D. Reitz and P.V. Farrell, Experimental and numerical studies of high-pressure multiple injection sprays, *SAE Technical Paper*, 960861, 1996.
- [3] C.S. Lee, K.H. Lee, R.D. Reitz and S.W. Park, Effect of split injection on the macroscopic development and atomization characteristics of a diesel spray injected through a common-rail system, *Atomization and Sprays*, vol. 16, pp. 543-562, 2006.
- [4] G. Blokkeel, B. Barbeau and R. Borghi, A 3D eulerian model to improve the primary breakup of atomizing jet, *SAE Technical Paper*, 2003-01-0005, 2003.
- [5] P.-A. Beau, R. Lebas, T. Ménard, A. Berlemont and F.-X. Demoulin, Numerical jet atomization – Part II: Modeling information and comparisons with DNS results, *ASME, FEDSM 2006-98166*, 2006.
- [6] R. Lebas, P.-A. Beau, G. Blokkeel and F.-X. Demoulin, ELSA model for atomization: To benefit of the eulerian and lagrangian descriptions of the liquid phase, *ASME, FEDSM 2006-98167*, 2006.
- [7] A. Vallet, A. Burluka and R. Borghi, Development of an eulerian model for the atomization of the liquid jet, *Atomization and Sprays*, vol. 11, pp. 619-642, 2001.
- [8] B. Abramzon and W.A. Sirignano, Approximate theory of a single droplet vaporization in a convective field: Effects of variable properties, Stefan flow and transient liquid heating, *Proceeding 2<sup>nd</sup> ASME-JSME Thermal Engineering Joint Conference Honolulu Hawaii*, pp. 11-18, 1987.
- [9] B. Abramzon and W.A. Sirignano, Droplet vaporization model for spray combustion calculations, *Journal of Heat ans Mass Transfer*, vol. 32 n°9, pp. 1605-1618, 1989.
- [10] R. Lebas, G. Luret, G. Blokkeel, T. Ménard and F.-X. Demoulin, Modeling vaporizing diesel sprays using the

- Eulerian Lagrangian Spray Atomization model, *Proceeding 6<sup>th</sup> International Conference on Multiphase Flow*, Leipzig, Germany, 2007.
- [11] V.A. Iyer and J. Abraham, Modelling of diesel sprays using an eulerian liquid eulerian gas two-fluid model, *ASME*, 2001.
- [12] V.A. Iyer, J. Abraham and V. Magi, Exploring injected droplet size effects on steady liquid penetration in a diesel spray with two-fluid model, *International Journal of Heat and Mass Transfer*, vol.45, pp. 519-531, 2002.
- [13] V.A. Iyer and J. Abraham, An evaluation of a two-fluid eulerian liquid eulerian gas model for diesel sprays, *Journal of Fluids Engineering*, vol. 125, pp. 660-669, 2003.
- [14] M. Pilch and C.A. Erdman, Use the breakup time data and velocity history data to predict the maximum size of stable fragments for acceleration-induced breakup of a liquid drop, *International Journal of Multiphase Flow*, vol. 13, pp. 741-757, 1987.
- [15] P.-A. Beau, M. Funk, R. Lebas and F.-X. Demoulin, Applying quasi-multiphase model to simulate atomization processes in diesel engines: Modeling of the slip velocity, *SAE Technical Paper*, 2005-01-0220, 2005.
- [16] R. Lebas, G. Blokkeel, P.-A. Beau and F.-X. Demoulin., ELSA model for atomization: To benefit of the eulerian and lagrangian descriptions of the liquid phase, *ICLASS Papers*, Kyoto, 2006.
- [17] G. Luret, R. Lebas, G. Blokkeel, A. Berlemont and F.-X. Demoulin, Study of the collision/coalescence effects with the ELSA model for diesel injection, *Proceeding 6<sup>th</sup> International Conference on Multiphase Flow*, Leipzig, Germany, 2007.
- [18] T. Ménard, S. Tanguy and A. Berlemont, Coupling level set/VOF/ghost fluid methods: Validation and application to 3D simulation of the primary break-up of a liquid jet, *International Journal of Multiphase Flow*, vol. 33, pp. 510-524.
- [19] M. Sussman, E. Fatemi, P. Smereka and S. Osher, An improved level set method for incompressible two-phase flows, *Computers and Fluids*, vol. 27, pp. 663-680, 1998.
- [20] J.A. Sethian, Level Set methods and fast marching methods: Evolving interfaces in computational geometry, fluid mechanics, computer vision and material science, *Cambridge university Press*, Cambridge, 1999.
- [21] G.S. Jiang and C.W. Shu, Efficient implementation of weighted essentially non-oscillatory schemes, *Journal of Computational Physics*, vol. 126, pp. 202-228, 1996.
- [22] D. Enright, R. Fedwick, J. Ferziger and I. Mitchell, A hybrid particle level set method for improved interface capturing, *Journal of Computational Physics*, vol. 183, pp. 83-116, 2002.
- [23] M. Sussman and E.G. Puckett, A coupled level set and volume-of-fluid method for computing 3D and axisymmetric incompressible two-phase flows, *Journal of Computational Physics*, vol. 162, pp. 301-337, 2000.
- [24] O. Tabebe, The multigrid preconditioned conjugate gradient method, PhD Thesis, University of Tokyo, 1996.
- [25] S. Tanguy, Développement d'une méthode de suivi d'interface. Applications aux écoulements diphasiques, PhD Thesis, University of Rouen, France, 2004.
- [26] R. Fedkiw, T. Aslam, B. Merriman and S. Osher, A non-oscillatory eulerian approach to interfaces in multimaterial flows (the ghost fluid method), *Journal of Computational Physics*, vol. 152, pp. 457-492, 1999.
- [27] M. Kang, R. Fedkiw and X.D. Liu, A boundary condition capturing method for multiphase incompressible flow, *Journal of Scientific Computing*, vol. 15, pp. 323-360, 2000.
- [28] X.-D. Liu, R. Fedkiw and M. Kang, A boundary condition capturing method for Poisson's equation on irregular domains, *Journal of Computational Physics*, vol. 160, pp. 151-178, 2000.
- [29] C. Rosales and C. Meneveau, Linear forcing in numerical simulations of isotropic turbulence: Physical space implementations and convergence properties, *Physics of Fluids*, vol. 17, pp. 95-106, 2005.