American Journal of Engineering Research (AJER)
e-ISSN : 2320-0847 p-ISSN : 2320-0936
Volume-4, Issue-1, pp-168-174
www.ajer.org
Research Paper Open Access

The differentiation between the turbulence and two-phase models to characterize a Diesel spray at high injection pressure

L. Souinida[1], M. Mouqallid[1] and L. Affad[2]
1(ENSAM-UMI Meknès, MOROCCO)
2 (FST Mohammedia, MOROCCO)

ABSTRACT: *The aim of this work is a study of the dynamics of the diesel spray in a combustion chamber of a Diesel engine of direct injection during the injection spray. To do it, the computer code Fluent (simulator of turbulent multiphase, multi-dimensional and unsteadies flows) is used to model the behavior of the spray in two dimensions. The spray evolution is simulated by using the Reynolds Averaged Navier-Stokes equations with a many models of closure of these equations, such as: Spalart Allmaras, k-ε, k-ω, k-k_L-ω, SAS, RSM. The two-phase flow is modeled by using the Volume of Fluid model and its coupling with Level-Set model, where the two phases behave as a pseudo-fluid with an indicator function determining the volume fraction of each phase. In this study, we demonstrate the evolution of the volume fraction field of the liquid and the averaged velocity field of the mixture, characterizing the behavior of the dynamic spray. Also, we drew the temporal evolution of the penetration length of our calculation (for all turbulence models cited above) with the experiment curve and that of computer code AVBP. Finally, we deduced the appropriate model of turbulence and two-phase flow for better characterizing the dynamic of the diesel spray.*

Keywords: *Diesel spray, CFD, RANS, Turbulence, VOF_Level-Set, Fluent.*

I. INTRODUCTION

Car manufacturers are facing increasingly severe regulations on pollutant emissions and fuel consumption. To respect these regulations, new combustion concepts in Internal Combustion Engine (ICE) have been developed. The Direct Injection (DI) diesel engine represents one of these concepts. Although it has shown a great potential, it still requires a comprehensive work to allow a better understanding. In fact, the efficiency of the combustion of the fuel and therefore the performance of a diesel engine depends on the quality of the fuel–air mixture. In conventional combustion systems, where the fuel is injected directly into air, it is necessary to provide adequate conditions for a good macrostructure of the mixture in the combustion chamber. This involves that the injected fuel must be uniformly distributed in the combustion chamber in the very short time interval available for the mixture formation. The fuel–air mixture process is strongly influenced by the spray behavior, which depends on several parameters. These parameters can be classified into two groups: parameters related to the diesel injection system (injection pressure, injection nozzle...) and others associated with the environment (ambient air density, ambient temperature...) where the spray is injected **[1-2]**.

During the recent years, Computational Fluid Dynamics has become one of the most important tools for both understanding and the considerable sensitivity of the model to small changes brought upon by numerical resolution, particularly in the area where onset of the transition occurs. The goal was the improvement of the diesel spray in the Internal Combustion Engine (ICE). The fuel injection process and the subsequent fuel–air mixing formation play a major role in the combustion and the pollutant emissions in the ICE. Even now, some of the processes involved in these phenomena, such as primary atomization and dense zone close to the nozzle, are not fully understood **[6–8]**.

The two-phase and turbulent nature of the diesel spray makes the study by DNS (Direct Numerical Simulation) or LES (Large Eddy Simulation) very heavy in computation time. Then, using the RANS (Reynolds Averaged Navier-Stokes) approach is privileged by Diesel motorists. The system of averaged equations the sense of Reynolds containing a nonlinear term characterizing the turbulence: the stress tensor of Reynolds. This term set a closure problem. To remedy this problem, one must resort to the turbulence modeling. Hence, several models

have been developed in literature. For example, the model of Spalart Allmaras, k-ε, k-ω, SAS, RSM and k-k_L-ω for several industrial applications of single-phase or multiphase flows at high Reynolds number. The aim of this paper is to see the applicability of these models to better characterize the behavior of the diesel spray.
In this study, we have chosen the Euler description and used the biphasic model VOF and its coupling with the Level-Set model, which is suitable for free-surface flow with a fixed mesh **[6-7]**. We will test both to decide which one better describes the two-phase nature of the diesel spray: is it the VOF model or the coupling "VOF_Level-Set".
The remaining of the presentation proceeds as follows. In Section II, we write the basic equations of interest. We present, in Section III, our numerical simulations. Results and discussion are pointed out in Section IV. We draw some concluding remarks in the last section.

II. BASIC EQUATIONS

The incompressible transient form of and the Reynolds Averaged Navier-Stokes equations with the assumptions of constant physical properties are described below in the following equations:

$$\frac{\partial U_i}{\partial x_i} = 0\,; \tag{1}$$

$$\frac{DU_i}{Dt} = \frac{\partial}{\partial x_j}\left[\vartheta \frac{\partial U_i}{\partial x_j}\right] - \frac{1}{\rho}\frac{\partial P}{\partial x_i} - \frac{\partial \overline{u_i u_j}}{\partial x_j} + F_s; \tag{2}$$

$$\frac{\partial \rho E}{\partial t} + \frac{\partial \rho E U_j}{\partial x_j} = \frac{\partial U_i \tau_{ij}}{\partial x_j} - \frac{\partial q_j}{\partial x_j}. \tag{3}$$

These equations express the mass conservation law, the quantity of movement and the total energy, respectively. There, U_i and u_i are the average and fluctuating velocity components in direction x_i, $\overline{u_i\, u_j}$ is the Reynolds stress tensor, P is the pressure, ϑ is the kinematic viscosity, ρ is the fluid density, F_s is the term of surface tension, E is the total energy, τ_{ij} is the strain rate tensor, and q_j is the heat flux. The term of Reynolds stress tensor causes a problem of closure. This is why we need turbulence models.
The inclusion of the turbulent fluctuation on the mean flow equations via the eddy viscosity is traduced by the following relation (Boussinesq hypothesis).

$$\overline{u_i u_j} = \frac{2}{3} k_{TOT}\delta_{ij} - \vartheta_{TOT}\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) \tag{4}$$

This term in equation (2) is modeled by many turbulence models in this study, such as: spalart allmaras **[8]**, $k - \varepsilon$ **[9-11]**, $k - \omega$ **[12-13]**, SAS **[14]**, RSM **[15-17]** and $k - k_L - \omega$ **[18]** models.
The relationship between the ambient pressure and air ambient density is:

$$P = \frac{\rho R T}{M} \tag{5}$$

Here, R is the constant of ideal gas, T the absolute temperature of the ambient air, and M the molar mass of the ambient air.

Finally, to describe the multi-phase flow character, used was made the VOF model and its coupling with the Level-Set model. To monitor the interface in its movement, we need the indicator function of each phase: The volume fraction α_k is advected by the local averaged velocity field, according to the following equation of the classical advection:

$$\frac{\partial \alpha_k}{\partial t} + U.\nabla \alpha_k = 0 \tag{6}$$

If we consider only the problem of two phases, it is sufficient to calculate once the scope of this α_k function to identify each phase of the two phases in the whole space: For example, we calculate α_1 (volume fraction of fluid 1) field, then we determine α_2 (volume fraction of fluid 2) by: $\alpha_1 + \alpha_2 = 1$.

III. NUMERICAL SIMULATION

The tools used in this work are Gambit and Fluent. The former is a geometry and mesh generator and the latter is a numerical solver in the finite volume.

The injection conditions correspond to those of some experiment made by **Verhoeven et al. [19]**. The temporal profile of the velocity is imposed at the input (figure 1) **[20]**, deducted from the mass flow measurement made in Ref. **[19]**. To validate the model used in this study, we assumed that the direction of the exit velocity is normal and the turbulent intensity at the spray exit is 1%. The density of the liquid is $\rho_l = 780\ kg.m^{-3}$, its absolute temperature is $T = 360\ K$ and the diameter of injection is $d = 0.2\ mm$.

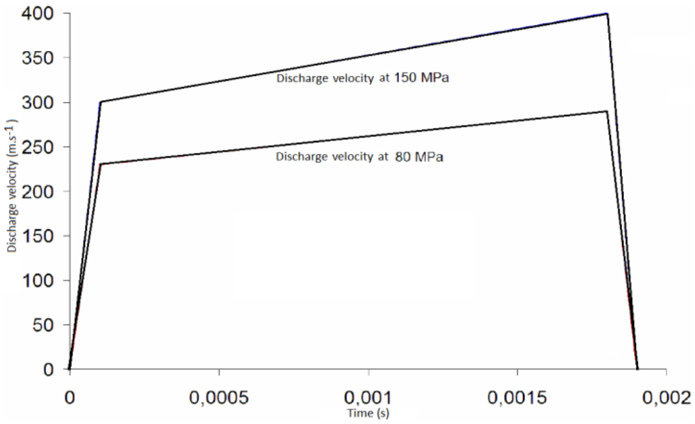


Figure 1: Injection velocity profile deducted by **Beau [20]**, from the injection rate profile measured **[19]**.

To study the effect of turbulence models on the behavior of the spray, we fixed everything about the other calculation conditions (two-phase model, numeric schemes, injection and ambient conditions, mesh...) and by computing for each model. We deduced the model that will better describe the diesel spray by comparing the calculation results with those of the literature.

After choosing the appropriate turbulence model, we study the behavior of the spray for both models of the two-phase flow: the Volume of Fluid and its coupling with Level-Set.

IV. RESULTS AND DISCUSSION

In this paragraph, we present the results obtained by using the above described models for simulating the development of the non-evaporating Diesel spray at high injection pressure. The macroscopic and microscopic parameters used for the characterization of the spray are the length penetration, the volume fraction field of the liquid and the field of the averaged velocity of the mixture.

First let's start by comparing between turbulence models for an injection pressure $P_i = 800\ bar$. Then, for the same pressure, we will analyze the effect of the two-phase model. Finally, after the choice of the appropriate models, we present the results for high injection pressure ($P_i = 1500\ bar$).

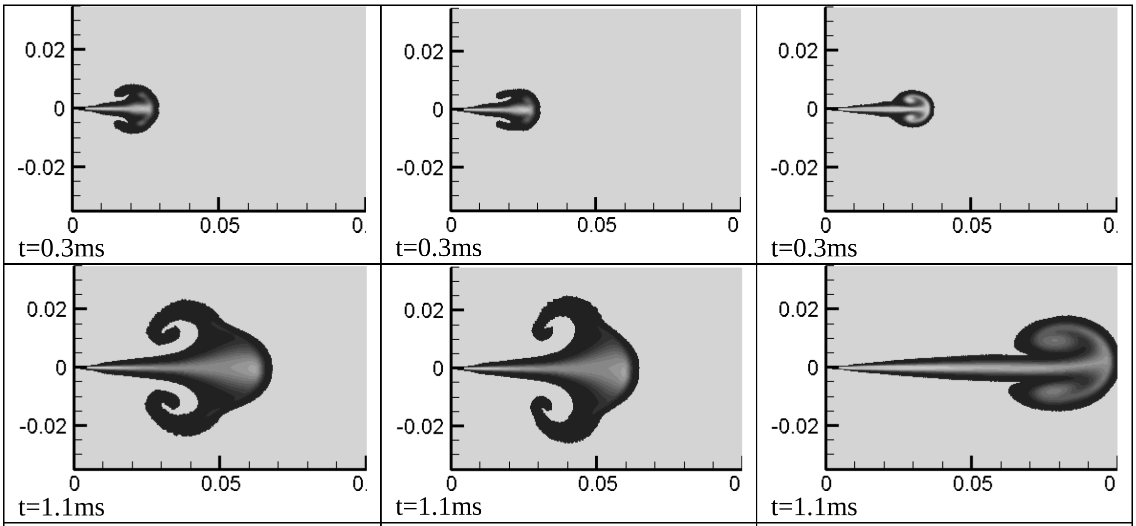
1. The comparison between the turbulence and two-phase models

We will do the comparison using the parameters that characterize the jet. These parameters are represented as follows:

The volume fraction field of the liquid

In figure 4, we visualized the volume fraction field of the liquid in two moments after injection (t=0.3ms; t=1.1ms) for three combinations of turbulence and two-phase models (a: VOF (Level Set) and k-k_L-ω, b: VOF and k-k_L-ω, c: VOF and k-ε).

The images of simulation show a symmetrical penetration of the jet into the gaseous environment. But, for the same moments after injection, the k-ε model gives greater penetration than the k-k_L-ω. We can note that the spray does not expands when using the k-ε model. For cons, we see very well the expansion of the jet using k-k_L-ω. This means that the k-k_L-ω takes into account the aerodynamic effects blocking longitudinally the jet to penetrate and expand radials.



(a) VOF_Level Set ; k-k_L-ω | (b) VOF ; k-k_L-ω | (c) VOF ; k-ε

Figure 2 : Volume fraction field of the liquid : ($P_i = 800\ bar$; $\rho_a = 25 kg.m^{-3}$)

❖ The penetration length of the spray

The spray penetration is a very important factor for a direct injection diesel engine. The penetration is particularly important for the formation of the mixture "fuel-air", the shock with the wall and the shape of the combustion chamber. We chose this parameter to validate the result of numeric simulation of many models of turbulence and the two models describing the two-phase flow (cited above).

In figure 3, we drew the curve of temporal evolution of the penetration length of the spray after injection using these conditions ($P_i = 800\ bar$; $\rho_a = 25 kg.m^{-3}$) for several turbulence models (k-ε ; SAS ; Spalart allmaras ; RSM ; k-k_L-ω) with the curve of the experiment **[19]** and of the curve of the filtered calculation [21].

The curves of figure 3 concerning the k-ε, the Spalart Allmaras and the SAS models show a linear penetration of the spray upon time after injection. Also, they highlight the concordance with that of the experiment between $t = 0ms$ and $= 0.2ms$: the phase of time after injection when the aerodynamics forces have low effect for the evolution of the spray. But, after $t = 0.2ms$, the curves move away of the curve of the experiment : the penetration remains fast, which means that the aerodynamics effects are negligible after that time. So, these turbulence models do not take into account the aerodynamic effects responsible of the blockage of spray after a time called "time to break up".

For the simulation curve using the k-k_L-ω model, it accords with the experiment and the filtered calculation by Martinez, highlighting two stages of penetration. Before $t = 0.2ms$, there is a rapid penetration and after this time, the penetration becomes slow. So, this model describes correctly the evolution of the penetration of spray taking into account the aerodynamics effect of the ambient milieu.

About the RSM model, the corresponding curve is very far than other simulation and experience curves.

After the analysis of different turbulence models, used to close the system of the averaged equations in the sense of Reynolds (to study the dynamic behavior of the diesel spray at the level of the temporal evolution of the penetration length), we can deduce that the k-k_L-ω is the model that can be valid in this case study.

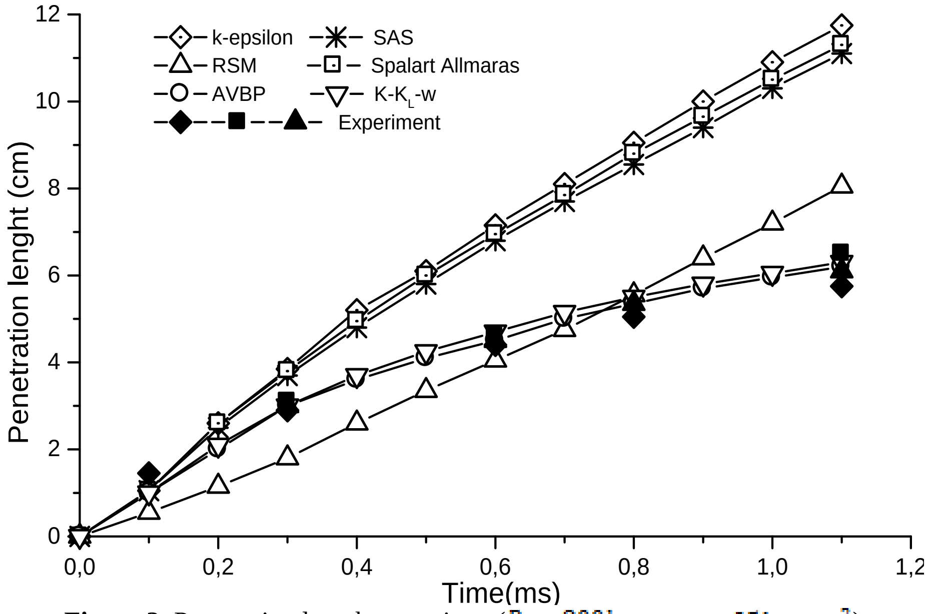


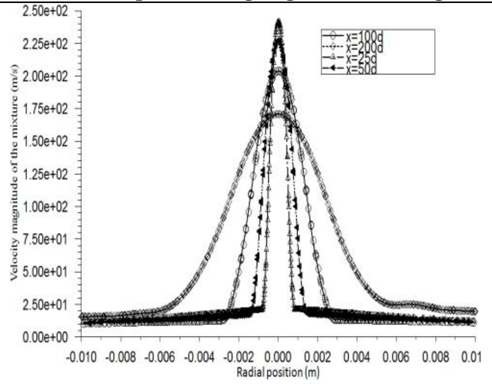
Figure 3: Penetration length upon time: ($P_i = 800bar$, $\rho_a = 25kg.m^{-3}$)

❖ **The averaged velocity field of the mixture**

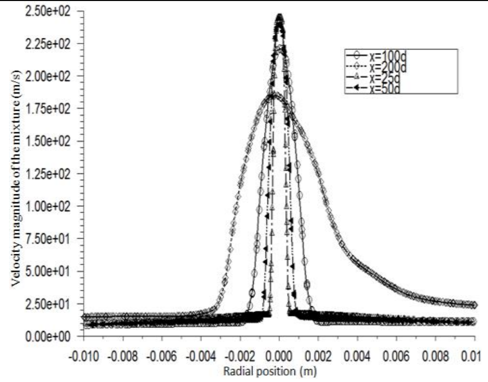
In the figure 3, we plotted the curves of the radial evolution of the averaged velocity of the mixture at several axial positions from the exit spray ($x = 25d$; $x = 50d$; $x = 100d$; $x = 200d$) for many turbulence models ((a) : k-ω, (b) : RSM, (c) : k-ε , (d) : Spalart allmaras, (e) and (f) : k-k$_L$-ω), used with VOF model which modeling the two-phase flow. Except the case (figure 3(f)), we used the coupling "VOF_Level-Set".
We can note, for all the curves that the radial evolution of the averaged velocity of the mixture is symmetrical relative to the jet axis. Except for the curve that corresponds to the RSM model (b), where: when we advance downstream, we lose the symmetry (the curve for $x = 200d$).

One can also note on the curves corresponding to the models (k-ω, RSM, k-ε, Spalart Allmaras) that the velocity remains maximal and equal to the axial jet exit velocity ($t = 0.6ms$ after injection) which overestimates the length of the potential core (the length where the velocity remains constant on the jet axis). For cons, the curves corresponding to the model k-k$_L$-ω, the velocity starts to diminish close by the outlet of the jet and which becomes very low away (at $x = 200d$).

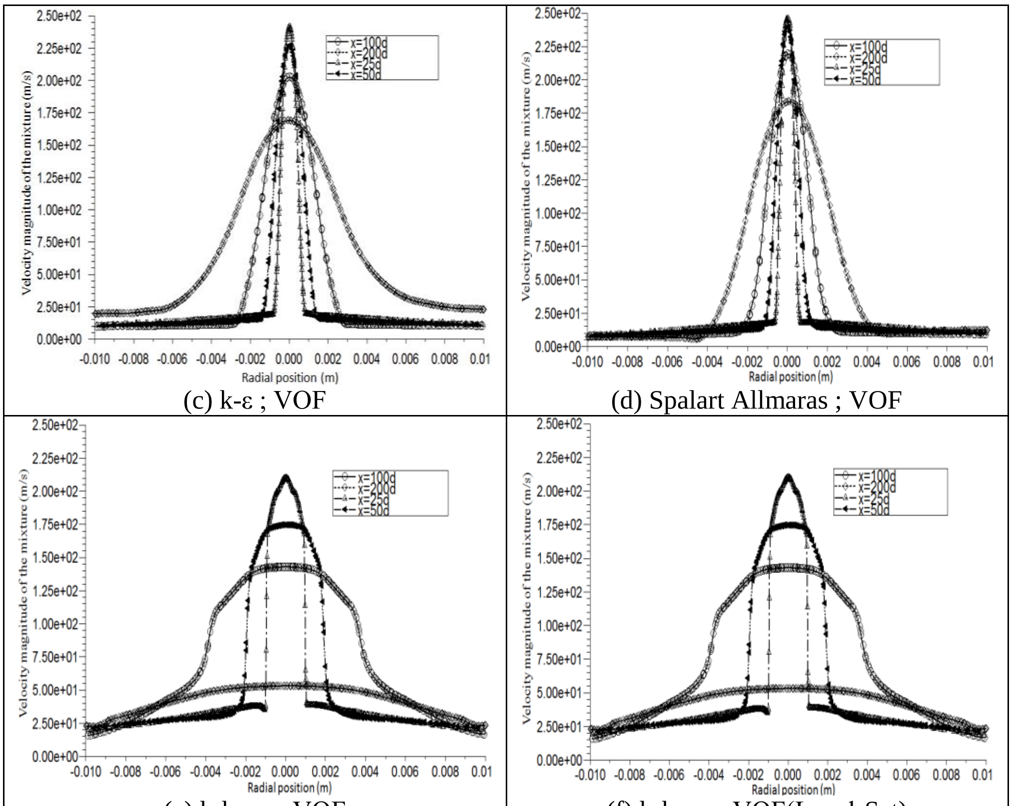
One can see on the curves which corresponding to the model k-k$_L$-ω that a very important radial evolution compared to other models, which provides information on the expansion of the jet advancing downstream. This last point is highlighted in the figure 2.



(a) k-ω ; VOF



(b) RSM ; VOF



(c) k-ε ; VOF (d) Spalart Allmaras ; VOF

(e) k-k_L-ω ; VOF (f) k-k_L-ω ; VOF(Level-Set)

Figure 4 : Averaged radial velocity field of the mixture at $t = 0.6ms$ after injection : $P_i = 800\ bar$ and $\rho_a = 25\ kg.m^{-3}$

2. The volume fraction field of the liquid at high injection pressure

After choosing the appropriate models to study the behavior of the dynamic spray: the k-k_L-ω for turbulence models and the coupling VOF_Level-Set model for two-phase flow, we did the study by using these models at high pressure injection. In figure 5, we visualized the volume fraction field of the liquid by using injection and ambient conditions ($P_i = 1500\ bar$, $\rho_a = 25\ kg.m^3$) that are the same conditions of the experiment **[19].** The images of simulation are agree with the experiment result **[19]**.

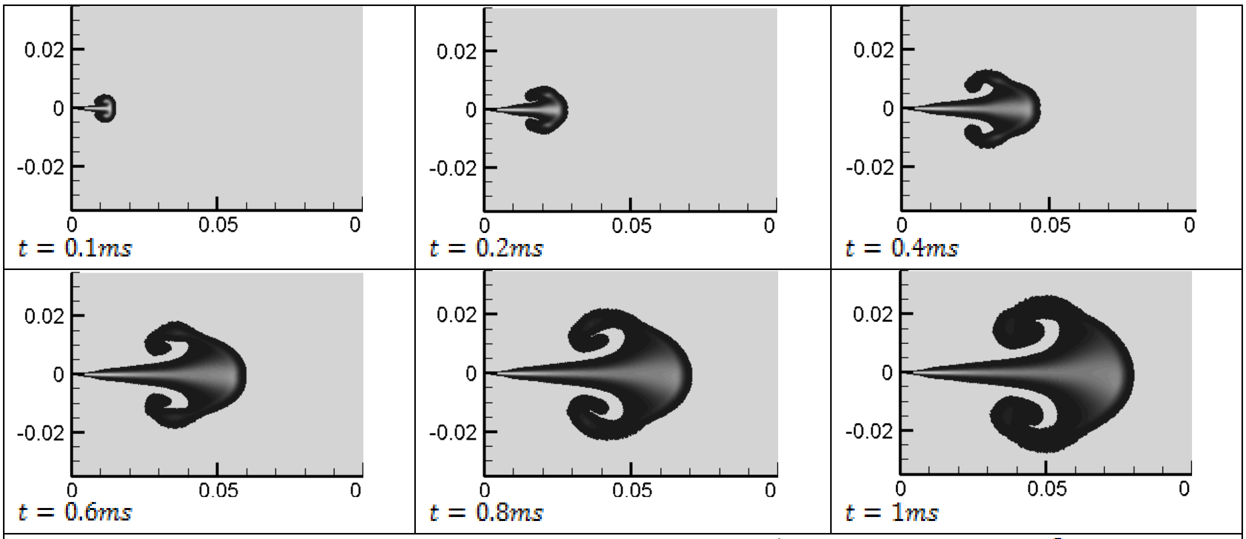


Figure 5 : Volume fraction field of the liquid : ($P_i = 1500\ bar$; $\rho_a = 25kg.m^{-3}$)

So, the k-k_L-w and the coupling "VOF_Level-Set" are the appropriate models for modeling the spray diesel at high injection pressure.

IV. CONCLUDING REMARKS

In this work, we studied the dynamic of the diesel spray in a combustion chamber of a Diesel engine of direct injection during the injection spray, at high injection pressure, in two dimensions. The penetration of a Diesel spray in a pressure chamber has been investigated numerically. The penetration length of the spray has been calculated at high pressure injection, with different turbulence models and two models of two-phase flow.
For the results, we visualized the evolution of the Diesel spray using the volume fraction field of the liquid and the longitudinal and radial evolution of the averaged velocity of the mixture. We drew the curves of the evolution of the penetration length of the spray. The comparison with the results of the experiment and the computation filtered gives an argument to prefer the $k\text{-}k_L\text{-}\omega$ model then the other models of turbulence for characterizing the behavior spray motion. Finally, we demonstrated the effect of two-phase models. We used the VOF models and its coupling with Level-Set, and we can remark a little difference on the conic form of the spray, which is clearly identified when we use the coupling VOF_Level-Set model.

REFERENCES

Journal Papers:

[2] A. Khalid, T. Yatsufusa, T. Miyamoto, J. Kawakami, and Y. Kidoguchi. "Analysis of Relation between Mixture Formation during Ignition Delay Period and Burning Process in Diesel Combustion". SAE Paper 2009 32 0018, pp.1 10, 2009.
[4] R. Lebas, G. Blokkeel, P. Beau, and F. Demoulin. "*Coupling vaporization model with the Eulerian–Lagrangian Spray Atomization (ELSA) model in diesel*". Engine conditions, SAE Technical Paper 2005-01-0213, 2005.
[6] C.W. Hirt and B. D. Nichols. "*Volume of Fluid Method for the Dynamics of Free Boundaries*". Journal of Computational Physics 39, 201-225 (1981).
[5] R. Lebas, T. Menard, P.A. Beau, A. Berlemont, and F.X. Demoulin. "*Numerical simulation of primary break-up and atomization: DNS and modelling study*". International Journal of Multiphase Flow 35 (3), 247–260.
[7] A.A. Oner, M.S. Akoz, M.S. Kirkgoz, and V. Gumus. "*Experimental Validation of Volume of Fluid Method for a Sluice Gate Flow*". Advances in Mechanical Engineering, Volume 2012 (2012), Article ID 461708, 10 pages.
[9] B. E. Launder and D. B. Spalding." *Lectures in Mathematical Models of Turbulence*". Academic Press, London, England. 1972.
[10] V. Yakhot and S. A. Orszag. "*Renormalization Group Analysis of Turbulence I Basic Theory*". Journal of Scientific Computing,. 1(1). 1–51. 1986.
[11] T.-H. Shih, W. W. Liou, A. Shabbir, Z. Yang, and J. Zhu. "*A New - Eddy-Viscosity Model for High Reynolds Number Turbulent Flows - Model Development and Validation*". Computers Fluids. 24(3). 227–238. 1995.
[12] D. C. Wilcox. "*Turbulence Modeling for CFD*". DCW Industries, Inc. La Canada, California. 1998.
[13] F. R. Menter. "*Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications*". AIAA Journal. 32(8). 1598–1605. August 1994.
[14] F. Menter and Y. Egorov. "*The Scale-Adaptive Simulation Method for Unsteady Turbulent Flow Predictions. Part 1: Theory and Model Description*". Journal Flow Turbulence and Combustion. 85. 113–138. 2010.
[15] M. Gibson and B. E. Launder. "*Ground Effects on Pressure Fluctuations in the Atmospheric Boundary Layer*". J. Fluid Mechanic 86. 491–511. 1978.
[16] B. E. Launder. *Second-Moment Closure: Present... and Future*? Inter. J. Heat Fluid Flow. 10(4). 282–300. 1989.
[17] B. E. Launder, G. J. Reece, and W. Rodi. "*Progress in the Development of a Reynolds-Stress Turbulence Closure*". J. Fluid Mech. 68(3). 537–566. April 1975.
[18] D. Keith Walters and Davor Cokljat. "*A three-equation eddy-viscosity model for Reynolds-Averaged Navier-Stokes simulations of transitional flows*". Journal of Fluids Engineering. 130. December 2008.
[19] D.Verhoven, J. Vanhemelryck, and T. Baritaud (1998). "*Macroscopic and ignition characteristics of high-pressure sprays of single-component fuels*". SAE technical paper 981069.

Thesis:

[1] J. Yon. "*Jet Diesel haute pression en champ proche et lointain : Etude par imagerie*". Doctorate Thesis, Rouan University, 2003.
[8] P. Spalart and S. Allmaras. "*A one-equation turbulence model for aerodynamic flows*". Technical Report AIAA-92-0439. American Institute of Aeronautics and Astronautics. 1992.
[20] P. Beau. "*Modeling atomization of a liquid jet-Application to Diesel sprays*". Doctorat Theses, University of Rouen (2006).
[21] L. Martinez. "*Simulation aux grandes échelles de l'injection de carburant liquide dans les moteurs à combustion interne*". Thèse soutenue Le 15 septembre 2009, Délivré par l'Institut National Polytechnique de Toulouse (INP Toulouse)

Books:

[3] A.H. Lefebvre. Atomization and Sprays (Taylor and Francis 1989).