Simulation of diesel Spray Flow with Different Break-up Models

M.H. Djavareshkian[1] and A. Ghasemi[2]

[1]Department of Mechanical Engineering
University of Tabriz, IRAN
< *djavaresh@Tabrizu.ac.ir* >

Abstract

The objective of this work is to simulate spray flow with different break-up models and investigate the effect of these models on DI diesel engine combustion and performance. In this simulation, the 3-Dimensional Naiver-Stokes equation is solved with SIMPLEC algorithm. All the simulations were carried out by the use of FIRE CFD tool. Results were validated with available experimental data for OM_355 DI diesel engine for mean cylinder pressure. There have been good agreements between experiments and the CFD calculations.

1. Introduction

In the last decade 3D-CFD has been successfully established for three dimensional simulations of fluid flow, mixture formation, combustion, and pollutant formation in internal combustion engines. In direct injected engines the accuracy of the simulation results and hence their contribution to design analysis and optimization strongly depends on the predictive capabilities of the models adopted for simulation of the injector flow, spray formation and propagation characteristics.

The present article provides an overview of the proper boundary conditions and models required for successful simulation of the spray formation/propagation characteristics in direct injected diesel engines. Individual model results are validated against selected experimental data. Finally, an outlook on future developments in IC engine spray modeling is given. For all cases presented in this study the CFD code FIRE is used for simulation of the relevant injector flow and spray formation and propagation processes [1-7].

2. Equations

The basic equations, which describe conservation of mass, momentum and scalar quantities, can be expressed in Cartesian tensor form as

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_j)}{\partial x_j} = 0 \qquad (1)$$

$$\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j - T_{ij})}{\partial x_j} = S_i^u \qquad (2)$$

$$\frac{\partial (\rho \phi)}{\partial t} + \frac{\partial (\rho u_j \phi - q_j)}{\partial x_j} = S^\phi \qquad (3)$$

The stress tensor and scalar flux vector are usually expressed in terms of basic dependent variable. The stress tensor for a Newtonian incompressible fluid is

$$T_{ij} = -p\delta_{ij} + 2\mu \vec{D} \qquad (4)$$

The scalar flux vector usually given by the Fourier-type law:

$$q_j = \Gamma_\phi \left(\frac{\partial \phi}{\partial x_j}\right) \qquad (5)$$

3. Spray Model

Currently the most common spray description is based on the Lagrangian discrete droplet method [8]. While the continuous gaseous phase is described by the standard Eulerian conservation equations, the transport of the dispersed phase is calculated by tracking the trajectories of a certain number of representative parcels (particles). A parcel consists of a number of droplets and it is assumed that all the droplets within one parcel have the same

Physical properties and behave equally when they move, break-up, hit a wall or evaporate. The coupling between the liquid and the gaseous phases is achieved by source term exchange for mass, momentum, energy and turbulence. Various sub-models account for the effects of turbulent dispersion [9], coalescence [10], evaporation [11], wall interaction [12] and droplet break up [13].

4. Break-Up Modeling

The atomization of IC-engine fuel sprays can be divided into two main processes, primary and secondary break-up. The former takes place in the region close to the nozzle at high Weber numbers. It is not only determined by the interaction between the liquid and gaseous phases but also by internal nozzle phenomena like turbulence and cavitation. Atomization that occurs further downstream in the spray due to aerodynamic interaction processes and which is largely independent of the nozzle type is called secondary break-up.

The classic break-up models like TAB (Taylor Analogy Break-up), RD (Reitz and Diwakar) and WAVE do not distinguish between the two processes [14]. The parameters of these models are usually tuned to match experimental data further downstream in the region of the secondary break-up. Originally, these parameters are supposed to depend only on nozzle geometry, in reality they also account for numerical effects.

Other models like ETAB (Enhanced TAB), FIPA (Fractionnement Induit Par Acceleration) or KH-RT (Kelvin Helmholtz - Rayleigh Taylor) treat the primary break-up region separately [14]. Hence, they in principle offer the possibility to simulate both break-up processes independently. The correct values for the additional set of parameters, however, are not easy to determine due to the lack of experimental data for the primary break-up region.

Despite the sometimes tedious tuning of these model parameters the use of break-up models is generally advantageous compared to the initialization of measured droplet distributions at the nozzle orifice. In the first approach the droplets are simply initialized with a diameter equal to the nozzle orifice (blob injection), the droplet spectrum automatically evolves from the subsequent break-up processes. The latter approach gives satisfying results only as long as injection pressure and droplet Weber numbers are low.

5. Injector Flow Coupling

The knowledge of the flow characteristics of the injected fuel at the nozzle exit is a key issue for a successful simulation of the spray primary break-up and hence the spray propagation characteristics in diesel engines. Recent modelling effort has lead to the successful coupling of the local flow conditions at the injector exit with advanced primary break-up models that account for injector flow induced turbulence as well as cavitation effects on the primary spray break-up processes [16]. The subsequent aerodynamic break-up processes are again modelled with the well established secondary break-up models.

The mathematical model used for injector flow calculations is based on a multi-fluid formulation of the relevant conservation laws [17].

The fuel vapor distribution in the nozzle hole clearly reveals the three-dimensional nature of the injector flow, with the secondary flow motion strongly determining the location and shape of the cavitation induced fuel vapor distribution that finally leads to the formation of two distinct cavitation zones at the nozzle hole exit.

The extension of the cavitation induced fuel vapor containing region is strongly depending on the injection pressure and the chamber back-pressure conditions as well as geometrical details of the injector.

Primary/secondary break-up modeling that accounts for the competing effects of turbulence, cavitation and aerodynamic induced break-up processes is based upon the spatially and temporally resolved injector flow data at the nozzle exit. In [16] the turbulence induced break-up is accounted for by solving a transport equation for the turbulent kinetic energy and its dissipation rate within the liquid fuel core.

The impact of the collapsing cavitation bubbles on the primary break-up is modeled via additional source terms in the turbulence model. The turbulence and cavitation induced break-up competes with the aerodynamic one until at a certain distance downstream of the nozzle exit the aerodynamic break-up processes become dominant.

It is evident that the maximum turbulence / cavitation induced break-up intensity is observed very close to the nozzle exit and can be attributed to the nozzle flow induced and (to a minor extent) to the cavitation collapse induced turbulent velocity fluctuations.

Due to dissipation of the turbulent fluctuations, however, the turbulence induced break-up rate is significantly reduced with increasing distance from the nozzle exit until it becomes negligible at about 2.5 mm downstream of the nozzle tip. The aerodynamic break-up rates show the opposite behaviour, i.e. they are very low immediately at the nozzle exit but increase significantly with increasing distance from the nozzle, where the compact liquid core has already been significantly disintegrated due to primary break-up mechanisms. Finally, even at the spray axis high aerodynamic break-up rates can be identified, indicating complete fragmentation of the compact spray core.

Elevated injection pressure levels lead to higher injection velocities and hence increased turbulence levels which directly lead to higher turbulence induced break-up rates. Increased chamber back-pressure levels, however, affect mainly the aerodynamic break-up mechanisms via the impact of higher gas densities and hence elevated interaction forces between ligaments / droplets and gaseous phase [16].

6. Eulerian Dense Spray Modeling

As shown in the previous parts fuel sprays in today’s IC-engine applications are usually modeled adopting the Lagrangian treatment of representative parcels of droplets tracked in the surrounding gas flow field. This method is especially suitable for dilute sprays, but has shortcomings with respect to modeling of dense sprays where particle interactions are strongly influenced by collisions and parcels have to be rearranged and redistributed very often.

Further problems are reported connected with bad statistical convergence [18] and also with dependence of the propagation of the spray on grid size [19].

An alternative approach is based upon adopting an Eulerian/Eulerian method treating different size classes of the spray droplets as separate, interpenetrating phases and solving conservation equations for each of them. The model under development is based on an Eulerian multiphase approach that has been derived from ensemble averaging of the conservation equations [20]. For each phase mass, momentum and energy conservation equations are solved as well as corresponding equations for turbulent kinetic energy and turbulent energy dissipation. Within each computational cell the droplet phases are characterized by a certain volume fraction. Thus all exchange processes related to droplet size or specific surface of the droplet phases depend on the flow regime and have to be modeled additionally. For the flow configuration considered here this concerns momentum transfer via drag and lift forces as well as mass transfer from secondary break-up, evaporation and collisions.

At the present state of implementation of the momentum transfer models the drag force takes into account effects of Reynolds number and volume fraction as well as deformation of the droplets. A first approach for turbulent dispersion force to treat interactions between gas phase turbulence and the droplet phases is also included. The models for lift forces cover Saffman and Magnus force.

7. Results

Figure 1 shows the comparison of mean pressure in the cylinder for present calculation and experimental data. As can be seen, the agreement between two results is close

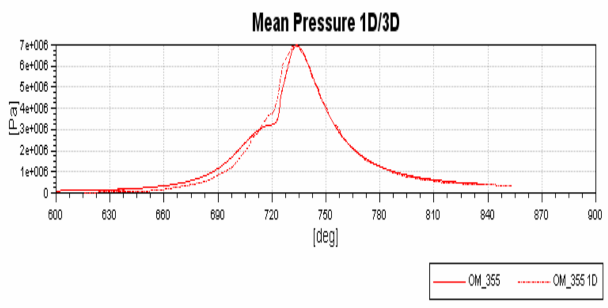


Figure 1 Comparison of Cylinder pressure for Model (Continuous) and experiment (dashed)

The introduction of break-up models has considerably simplified the simulation of spray processes. In the past a number of different approaches have been presented and it is not easy to decide which one to choose for a specific simulation task. It turns out that practically all the break-up models are capable of reproducing measured data, as long as model constants are properly chosen.

One quantity characterizing the average droplet is size of a spray and thus the success of spray break-up is the Sauter mean diameter (SMD). Figure 2 demonstrates the Sauter diameter and the calculated penetration by different break-up models.

According to [23], the time-dependent development of the spray penetration length can be divided into two phases. The first phase starts at the beginning of injection (*t* = 0, needle begins to open) and ends at the moment the liquid jet emerging from the nozzle hole begins to disintegrate ($t = t_{break}$). Because of the small needle lift and the low mass flow at the beginning of injection, the injection velocity is small, and the first jet break-up needs not always occur immediately after the liquid leaves the nozzle. During the second phase ($t > t_{break}$), the spray tip consists of droplets, and the tip velocity is smaller than during the first phase. The spray tip continues to penetrate into the gas due to new droplets with high kinetic energy that follow in the wake of the slower droplets at the tip (high exchange of momentum with the gas) and replace them. Experimental investigations have shown that the transition from a pure turbulent to a cavitating nozzle hole flow results in an increase of spray cone angle and in a decrease of penetration length. Strongly cavitating nozzle flows produce larger overall spray cone angles and smaller penetration lengths than non-cavitating ones.

The spray penetration increases with time due to the effect that new droplets with high kinetic energy continuously replace the slow droplets at the spray tip.

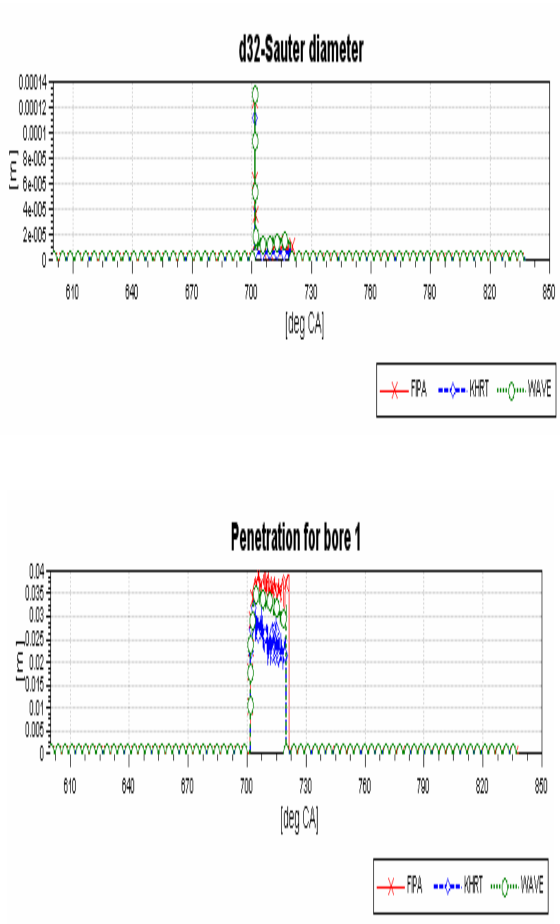


Figure 2 Comparison of different break-up models; a) Sauter Mean Diameter, b) spray penetration

Figure 3 presents the effect of different break-up models on the amount of liquid mass remaining after injection. If standard WAVE model with blob injection (=> initial droplets have the diameter of the nozzle orifice) is used for the simulation, it often happens that there is hardly any fuel vapor close to the nozzle. This is due to the fact that the droplets are still very large at the beginning and therefore hardly evaporate.

The KH-RT model with the lower penetration exhibits the most amount of liquid remaining.

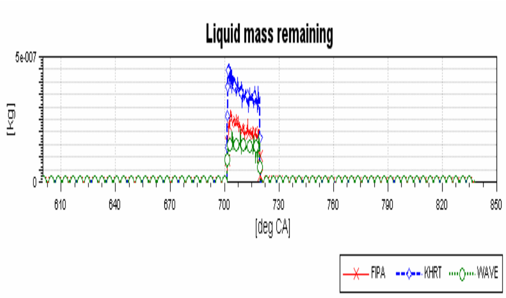


Figure 3 Liquid mass Remaining

Figure 4 represents the effect of different break-up models on the mean cylinder pressure. The results of the FPA and WAVE models are similar.

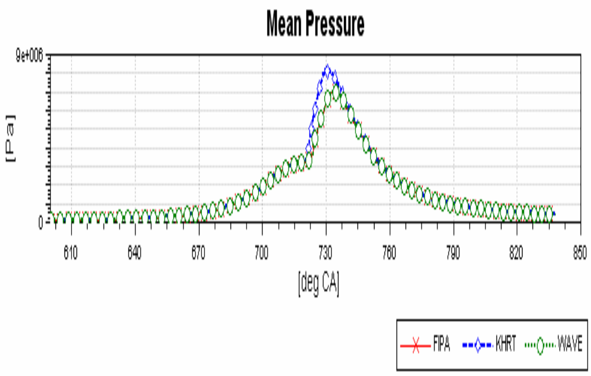


Figure 4 Comparison of Cylinder pressure for different Breakup models.

Figure 5 represents the effect of different break-up models on the combustion rate of heat release.

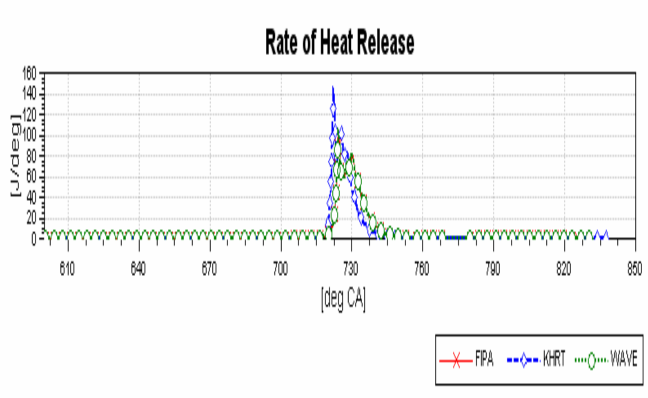
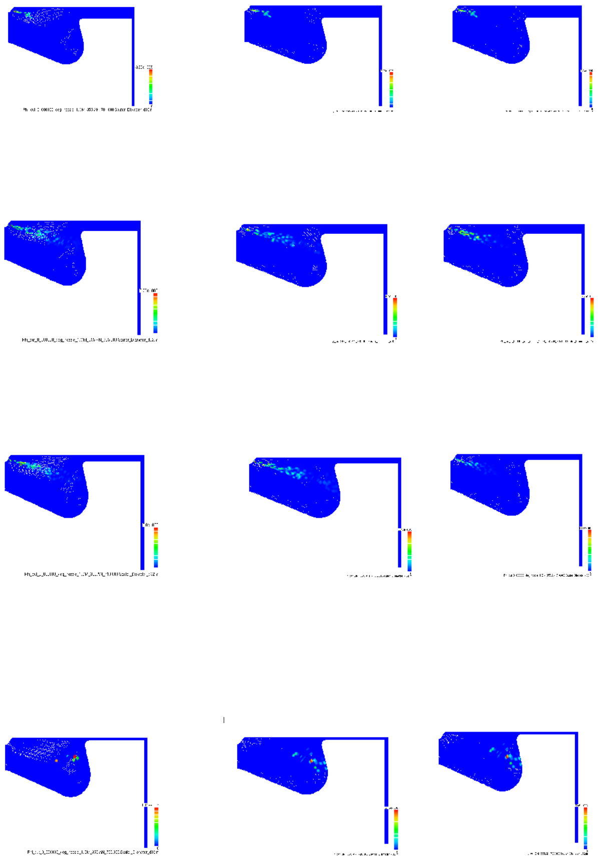


Figure 5 Comparison of Heat release rate for different Breakup models.

Figure 6 describes the variation of the Sauter mean Diameter (SMD) distribution during the injection process in different break-up models. For all models its obvious that the SMD has grater amounts near the nozzle hole.



a) b) c)

Figure 6 Comparison of SMD in four crank angles (701.6, 705, 710 and 720?CA) for different Breakup models: a) WAVE b) FIPA c) KH-RT

Further effects as, e.g., from virtual mass or Basset force have been neglected for the present application of droplet flow in a gaseous environment as is supported by the analysis of Sommerfeld [21].

Regarding mass transfer between the phases basic models for evaporation, secondary break-up and collisions have been implemented. Evaporation mass transfer is described according to differences of vapor pressure within the droplet phase and in the gas flow. For secondary breakup rate approaches from the standard WAVE and FIPA breakup models have been implemented. The collision model takes into account coalescence as well as secondary breakup after collision according to a collision Weber number criterion. In general the exchange terms are formulated in a modular way to allow an easy coupling of additional models for the different interphase exchange processes. Matching of turbulence model and adaptation of turbulent dispersion force for the spray application is performed presently.

The model has already been applied to Diesel injection test cases using simplified but typical conditions. Effects of inlet conditions, various drag formulations and basic functionality of the secondary break-up, evaporation and collision models have been tested successfully [22].

8. Conclusions

In the present article the spray flow has been simulated with different break-up models and the effect of these models on DI diesel engine combustion and performance was investigated. All the simulations were carried out by the use of FIRE CFD tool. Results were validated with available experimental data for OM_355 DI diesel engine for mean cylinder pressure. There have been good agreements between experiments and the CFD calculations.

9. REFERENCES

[1] Tatschl, R., Wieser, K., Reitbauer, R., Multidimensional Simulation of Flow Evolution, Mixture Preparation and Combustion in a 4-Valve Gasoline Engine, COMODIA 94, Yokohama, 1994

[2] Tatschl, R., Fuchs, H., Brandst?tter, W. Experimentally Validated Multi-dimensional Simulation of Mixture Formation and Combustion in Gasoline Engines, IMechE C499/050/96, 1996

[3] Tatschl, R., Riediger, H., PDF Modeling of Stratified Charge SI Engine Combustion, SAE 981464, 1998

[4] Tatschl, R., Pachler, K., Winklhofer, E., A Comprehensive DI Diesel Combustion Model for Multidimensional Engine Simulation, COMODIA 98, Kyoto, 1998

[5] Tatschl, R., Riediger, H., Bogensperger, M., Multidimensional Simulation of Spray Combustion and Pollutant Formation in a Medium Speed Marine Diesel Engine, 1998 FISITA World Automotive Congress, Paris, 1998

[6] Tatschl, R., Wiesler, B., Alajbegovic, A., Kuensberg Sarre, Ch., Advanced 3D Fluid Dynamic Simulation for Diesel Engines, Thermofluidynamic Processes in Diesel Engines, Valencia, 2000

[7] Tatschl, R., Riediger, H., v. Künsberg Sarre, Ch., Putz, N., Fast Grid Generation and Advanced Physical Modelling – Keys to Successful Application of CFD to Gasoline DI Engine Analysis, ImechE C587/042/2000, 2000

[8] Dukowicz, J.K., A Particle-Fluid Numerical Model for Liquid Sprays, Journal of Computational Physics, Vol. 35, pp. 229-253, 1980

[9] Gosman A.D. and Ioannides, E., Aspects of Computer Simulation of Liquid-Fueled Combusters, J. Energy, 7, pp. 482 490, 1983

[10] O'Rourke, P.J., Modeling of Drop Interaction in Thick Sprays and a Comparison with Experiments, IMechE - Stratified Charge Automotive Engines Conference, 1980

[11] Dukowicz, J.K., Quasi-steady Droplet Phase Change in the Presence of Convection, Los Alamos Report LA-7997-MS, 1979

[12] Naber, J.D., Reitz, R.D., Modeling Engine Spray / Wall Impingement, SAE 880107, 1988

[13] Liu, A.B. and Reitz, R.D., Modeling the Effects of Drop Drag and Breakup on Fuel Sprays, SAE 930072

[14] v. Künsberg Sarre, Ch., Tatschl, R., Spray Modeling / Atomization – Current Status of Break-Up Models, ImechE Seminar on Turbulent Combustion of Gaseous and Liquids, Lincoln, 1998

[15] Habchi, C., Baritaud, T. et al., Modeling Atomization and Breakup in High-Pressure Diesel Sprays, SAE 970881

[16] Tatschl, R., Kuensberg Sarre, Ch., Alajbegovic, A, Winklhofer, E., Diesel Spray Break-Up Modeling Including Multidimensional Cavitating Nozzle Flow Effects, ILASS Europe 2000, Darmstadt

[17] Drew, D.A., Passman, S.L., Theory of Multi-component Fluids, Springer, New York, 1998

[18] Krüger, Ch., Validierung eines 1D-Spraymodells zur
Simulation der Gemischbildung in direkteinspritzenden Dieselmotoren, Dissertation RWTH Aachen, M?rz, 2001

[19] Abraham, J., What is Adequate Resolution in the Numerical Computation of Transient Jets?, SAE
970051
[20] Alajbegovic, A., Drew, D.A., Lahey, R.T., An Analysis of Phase Distribution and Turbulence in Dispersed Particle/Liquid Flows, Chem. Eng. Comm., Vol. 174, pp. 85-133, 1999
[21] Sommerfeld, M., Modellierung und numerische Berechnung von partikelbeladenen turbulenten Str?mungen mit Hilfe des Euler/Lagrange Verfahrens, Shaker Verlag, Aachen, 1996
[22] v. Berg, E., Alajbegovic, A., Tatschl, R., Krüger, Ch., Michels, U., Multiphase Modeling of Diesel Sprays with the Eulerian/Eulerian Approach, 17th ILASS Europe Conference, Zürich, 2001
[23] Hiroyasu H, Arai M (1990) Structures of Fuel Sprays in Diesel Engines. SAE-paper 900475