Simulation of High Pressure Fuel Injection Influence to the Diesel Engine Processes

ADRIAN SABĂU
Department: Naval Mechanical Engineering
Maritime University of Constantza
Address: Mircea cel Batran Street, no. 104, Contantza
ROMANIA
adrian.sabau@cmuwedu.eu, http:/mail.imc.ro

Abstract - Computational simulation for the combustion process in a DI diesel engine was performed through a code made by the author [4]. Two conditions of different injection pressure were examined in the simulation. The injection pressures are 1500 bar and 500 bar. The combustion phenomenon was modeled as a combined process of formation of a combustible mixture and a chemical reaction. The rate of mixture formation was assumed to be dependent on the turbulence characteristics and the concentration of species in each computational cell. The rate of chemical reaction is described as an Arrhenius equation. The result of the simulation agrees with the experimental result qualitatively, and the effect of injection pressure on the combustion process is well predicted.

Key-words: spray, turbulence, injection, heat release, injection pressure, chemical reactions.

1 Introduction

In this paper, we are interested in problems, such as high-pressure fuel injection in an internal combustion engine, in which the spray carries sufficient momentum to entrain and set into motion the surrounding gas. In turn, the motion of the gas in the vicinity of the particles reduces the resistance to their motion and allows the spray to penetrate much further than would otherwise be the case. It is important, therefore, to account for the interaction between the particles and the gas. This interaction is of course always present, but it is particularly significant whenever the particles are sufficiently small so that the coupling of a particle to the gas is strong.

The procedure is to represent the spray by discrete particles, rather than by continuous distributions. This amounts to a statistical (Monte Carlo) formulation of the problem, since the finite number of particles used represents a sample of the total population of particles. The developed method [5] is of multi-dimensional type and it solves the specific combined equation systems of:

- Compressing eddying flow by using:
  - SGS (Subgrid scale turbulent viscosity) model for turbulence;
  - wall logarithm law for the turbulent boundary layer;
  - altered Reynolds formula for the caloric boundary layer;
  - approximating the fluid as Newtonian to compute its viscosity
- Chemical reactions of fuel combustion by using:
  - A kinetic equation of fuel combustion;
  - Splitting equations of combustion products treated on equilibrium;
  - Extended Zeldovich mechanism solved kinetically for NO.
- The flow and evaporation of liquid jet of particles by using:
  - The general equation of jet simplified in stochastic approach;
  - The equation of drops’ evaporation as deducted by O’Rouke;

The formula is bi-dimensional in space and thus allows a plane and axial symmetrical approach of the geometry of the combustion chamber. The axial symmetrical formula, the most often case met in practice, allows us to take into account the swirl movement as well, enhancing thus the spatial resolution and partially implementing the third geometric dimension.

Computational simulation for the diesel combustion is an active area of research. For practical applications, there are some requirements in the computational code. They are accuracy of the simulation, ability of dealing with several input
variables such as geometry of the combustion chamber, operating conditions, and fuel specifications.

2 Fuel Spray Model

It will be assumed that no particle coalescence or particle breakup occurs. This implies that the particles are sufficiently dispersed that particle collisions are infrequent. The initial breakup of liquid sprays or jets is not considered. It is assumed that initial conditions for the particles are known. That is, the initial particle size distributions, positions, and velocities are independently specified.

This leads to two sets of equations, one set for the gas and the other for the particles. These equations will be coupled primarily by two mechanisms, the displacement of gas by the volume occupied by the particles and momentum interchange between particles and the gas.

2.1 Spray equations

\[
\frac{\partial f}{\partial t} + \nabla \cdot (fu_p) + \nabla \cdot (F_p) + \frac{\partial}{\partial r_p} \left( J F_p \right) + \frac{\partial}{\partial T_p} \left( J F_p \right) = \dot{Q} \quad (1)
\]

Where \( u_p, F_p, R_p \) and \( T_p \) is the velocity, force, rate of radius variation and the temperature of an arbitrary droplet in the spray at \( x \) position [1].

2.2 Continuity equation for the gas

\[
\frac{\partial \theta}{\partial t} + \nabla \cdot \theta u_g = 0 \quad (2)
\]

Where \( \theta \) is the void fraction, or the fraction of the volume occupied by the gas, and \( u_g \) is the gas velocity. The presence of the void fraction in this equation accounts for the displacement effect of the particles.

2.3 Momentum equation

\[
\frac{\partial}{\partial t} \mu_g + \nabla \cdot \mu_g u_g = \frac{\theta}{\rho_g} \nabla p + \frac{\rho_g}{\rho_g} \nabla u_g + \frac{1}{\rho_g} M_p \quad (3)
\]

Where \( g \) is the acceleration of gravity, \( p \) is the pressure, \( \mu_g \) is the kinematic viscosity, and \( M_p \) is the term defining momentum exchange with the particles, per unit volume. An alternative form of this equation can be obtained by subtracting out the continuity equation [3]:

\[
\frac{\partial}{\partial t} u_g + u_g \cdot \nabla u_g = g - \frac{1}{\rho_g} \nabla p + \frac{1}{\rho} \nabla \cdot \mu_g \nabla u_g + \frac{1}{\rho} M_p \quad (4)
\]

This is the form of the equation used. In a turbulent flow, the gas equations of the previous section are written in terms of the mean velocity \( u_g \). For particles, gas turbulence is important as a mechanism for diffusion; and it is convenient to write the instantaneous, rather than averaged, equations for the particles. To do this, we define the instantaneous gas velocity, \( U_g = u_g + u_g' \), where \( u_g' \) is the turbulent component of the gas velocity.

2.4 Particle momentum equation:

\[
u_{pk} = \frac{dx_{pk}}{dt} \quad (5)
\]

\[
m_k \frac{d}{dt} u_{pk} = m_k g - \frac{m_k}{\rho_k} \nabla p + D_k \left[ U_g U - u_{pk} \right] ; \quad (6)
\]

Where \( x_{pk} \) is the particle position, \( u_{pk} \) is its velocity, \( m_k \) is its mass, and \( \rho_k \) is its density. The notation \( D_k \) is used to denote the drag function, evaluated using the velocity \( U_g \), which is the coefficient in the force acting on the particle due to its motion through the gas. It will be convenient to abbreviate the notation to \( D_k \) when referring to the drag function evaluated at the mean gas velocity.

It is more convenient to consider the effect of turbulence on the particles to be due to a force \( f_{pk} \), in which case the momentum equation is written [3]:

\[
f_{pk} = D_k \left[ U_g U - u_{pk} \right] - D_k \left[ u_g - u_{pk} \right] , \quad (7)
\]

\[
m_k \frac{d}{dt} u_{pk} = m_k g - \frac{m_k}{\rho_k} \nabla p + D_k \left[ u_g - u_{pk} \right] + f_{pk} \quad . \quad (8)
\]

The terms in the gas equations (2 – 4) dependent on the particles have not yet been defined. Taking ensemble averages, we can write:

\[
\theta = \frac{1}{4} \sum_k \left\langle \frac{4}{3} \pi x_k^3 \delta(x-x_{pk}) \right\rangle , \quad (9)
\]

\[
M_p = - \sum_k D_k \left[ U_g U - u_{pk} \right] \delta(x-x_{pk}) . \quad (10)
\]

For modeling purposes, it is not possible to deal with the large number of droplets, so that a sampling
technique be employed in which each single particle represents a characteristic group of particles. This is equivalent to the following distributions function:

\[ f(r, x, u_p, t) = \sum_k N_{pk} \delta(r - r_k) \delta(x - x_k) \]

\[ \delta(u_p - u_{pk}) = -\frac{\Delta N}{\Delta r \Delta x \Delta u_p} \]  

(11)

where \( N_{pk} \) is the number of identical particle by particle \( k \), \( \Delta N \) is the number of particles in the volume \( \Delta r \Delta x \Delta u_p \).

The droplet size distribution is:

\[ f_r(r) = \frac{6}{D_{32}^2} \exp \left( -\frac{6r}{D_{32}} \right) \]  

(12)

where \( D_{32} \) is Sauter mean diameter which is consider to varying very little.

Eddy viscosity was estimated using a value appropriate to a turbulent gaseous jet:

\[ \mu_g = 0.0161 \sqrt{0.25 \pi d^2 V^2} \]  

(13)

where \( d \) is the orifice diameter and \( V \) is the droplet injection velocity.

No attempt was made to calculate particle diffusion accurately. Particle turbulence was modeled using the assumption, that the fluid turbulence is isotropic and has a Gaussian distribution in velocity. Given the turbulent kinetic energy \( k \), is \( k = 0.1 u_g^2 u_g \), and turbulent velocity is:

\[ u_g^t = k^{0.5} \text{sgn}(X) \cdot \text{erf}^{-1}(|X|) \]  

(14)

where, as before, \( X \) and \( Y \) are random variables selected for a uniform distribution in the range \(-I < X, Y < I\). To complete the description, the relevant turbulence time scale \( \tau \) is required. The velocity \( u_g^t \) is assumed to act for a time equal to \( \tau = \Delta t \). An elementary analysis suggests that the produces particle diffusion corresponding to:

\[ D_T \approx 0.66 \left( \frac{D_k}{m_k} \right)^2 q \tau^3 \]  

(15)

The resulting particle concentration, which is Gaussian, forms a distribution function for the location of the particle. Particle positions are randomly selected from within this distribution, such that on each time step the diffusional increment in particle position is

\[ \Delta x_{pk} = (4D_t \Delta t)^{0.5} \text{sgn}(X) \text{erf}^{-1}(|X|) \]  

(16)

3 Chemical Models

The model involves twelve species and the consequent equations are:

A stoichiometric, irreversible, kinetic equation (22), a single-step hypothetical fuel combustion \( (C_nH_mO_r, \text{given by elemental analysis where } m, n \text{ and } r \text{ may be not integers);} \)

\[ C_xH_nO_r + \left( n + \frac{m}{4} - \frac{r}{2} \right) O_2 \rightarrow \frac{79}{21} (n + \frac{m}{4} - \frac{r}{2}) N_2 \rightarrow \text{, (17)} \]

Three partial equilibrium reversible equations expressing the extended Zeldowich mechanism (18) for NO evaluation;

\[ \begin{aligned}
N_2 + O &\leftrightarrow k_{11/2} NO + N \\
N + O_2 &\leftrightarrow k_{13/4} NO + O \\
N + OH &\leftrightarrow k_{5/6} NO + H
\end{aligned} \]  

(19)

Six reversible equilibrium equations for main combustion products dissociation [2]:

\[ \begin{aligned}
H_2 &\leftrightarrow 2H \\
O_2 &\leftrightarrow 2O \\
N_2 &\leftrightarrow 2N \\
O_2 + H_2 &\leftrightarrow 2OH \\
O_2 + 2H_2O &\leftrightarrow 4OH \\
O_2 + 2CO &\leftrightarrow 2CO_2
\end{aligned} \]  

(20).

The chemical source term in continuity equation is given by equation (6), and the chemical heat release term in the energy equation is given by equation (21) [4].

\[ \dot{\rho}_c^c = W_t \sum_r (b_r - a_r) \dot{\omega}_r \]  

(21)

\[ \dot{Q}_c = \sum_r q_r \dot{\omega}_r \]  

(22)
where \( a_{kr} \) and \( b_{kr} \) are stoichiometric coefficients, \( W_k \) is the molecular weight, \( \dot{w}_r \) is the rate of progress of \( r \)-th reaction, \( q_r \) is the negative of the heat of reaction at 0º K. If \( r \) is a kinetic reaction, then \( \dot{w}_r \) is computed by equation (23) which is an equilibrium reaction and \( \dot{w}_r \) is implicitly determined by the condition expressed by equation (24):

\[
\dot{w}_r = k_{fr} \prod_k (\rho_k / W_k)^{a_{fr}} - k_{br} \prod_k (\rho_k / W_k)^{b_{fr}}, \quad (23)
\]

\[
\prod_k (\rho_k / W_k)^{b_{fr} - a_{fr}} = K'_c(T), \quad (24)
\]

where: \( k_{fr} \) and \( k_{br} \) are the rate coefficients for reaction (generalized Arrhenius form) \( r \), \( a_{fr}' \) and \( b_{fr}' \) are orders of the reaction and \( K'_c(T) \) is the concentration equilibrium constant.

### 4 Computational Conditions

The model presented in the paper was used for numerical simulation of the T684 engine manufactured by "Tractorul" Plant of Brasov, a four stroke automotive engine specifications are shown in Table 1. The calculating conditions are shown in Table 2. The difference of injection pressure is given through the input parameters of fuel injection shown in Table 3. The engine has a centrally located 4-holes nozzle and an accumulator type fuel injection system. A periodicity was assumed. So, the calculation was performed for a part of the combustion chamber which includes one fuel spray.

#### Table 1 Engine specifications

<table>
<thead>
<tr>
<th>Specification</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bore [mm]</td>
<td>105</td>
</tr>
<tr>
<td>Stroke [mm]</td>
<td>115</td>
</tr>
<tr>
<td>Compression ratio</td>
<td>17.5</td>
</tr>
<tr>
<td>Piston bowl [mm]</td>
<td>60</td>
</tr>
<tr>
<td>Injection nozzle [mm]</td>
<td>4 holes, diameter=0.25</td>
</tr>
</tbody>
</table>

#### Table 2 Calculation data

<table>
<thead>
<tr>
<th>Specification</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Engine speed [rpm]</td>
<td>1400</td>
</tr>
<tr>
<td>Injection quantity [g/st]</td>
<td>55</td>
</tr>
<tr>
<td>Injection timing TDC [deg.]</td>
<td>-5</td>
</tr>
<tr>
<td>Injection pressure [bar]</td>
<td>1500 and 500</td>
</tr>
</tbody>
</table>

#### Table 3 Data for fuel spray

<table>
<thead>
<tr>
<th>Injection pressure [bar]</th>
<th>Injection Duration [deg.]</th>
<th>Sauter Radius [mm]</th>
<th>Initial Velocity [m/s]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1500</td>
<td>11.7</td>
<td>0.025</td>
<td>374</td>
</tr>
<tr>
<td>500</td>
<td>17.8</td>
<td>0.045</td>
<td>215</td>
</tr>
</tbody>
</table>

Simulation is done from -20 TDC degrees to 60 TDC degrees. The calculus is done under the hypotheses that the reactions are de-coupled and the pressure is constant.

The reaction speed for the equilibrium conditions, \( \dot{w}_r \), (where \( r \) is the number of chemical reactions in equilibrium conditions) is computed by the use of an iterative algorithm. Thus, every reaction is relaxed until the equilibrium constant given by equation (23) becomes equal to the value resulted from equation (24), with an acceptable error.

The temporal differentiate is based on ICE (Implicit Continuous-fluid Eulerian) algorithm which is a partial implicit method. This iterative technique joins the continuity and moment equations and solves them simultaneously by using the state equation; the energy equation is solved explicit apart from the other two. To move forward in time some values several steps in time are needed. This approach is in direct connection to the spatial digitization based on ALE method [6].

The code is written in Matlab [2].

### 4 Results and Discussion

Figure 1-3 show the predicted spray evolution for 1500 bar injection pressure and Figure 4-6 for 500 bar. Concentrations of vaporized fuel O2 and NOx is shown as representative species in parallel with spray evolution, Figures shows a section includes the axis of fuel injection. The concentration of fuel increases in the injection duration only near the fuel spray axes, and is low at the wall. After the end of injection, the concentration of fuel decreases rapidly. The concentration of product species increases around the fuel spray. Strong turbulence is formed along the
axes of fuel spray in the injection duration (from -1 to 3 TDC degree for 1500 bar, from -4 to 15 TDC degree for 500 bar). From these observations, the combustion process seems to occur mainly around the axes of fuel spray. Therefore, the fuel spray characteristics such as the penetration and the shape have essential effects on the combustion process in this kind of simulation for DI diesel engines.

The results of the simulation well agree with the observed combustion process qualitatively, and the simulation well predicts several features of the effect of high pressure fuel injection on the combustion process. When the fuel injection pressure is high, the flame develops more rapidly than in the ordinary pressure condition. In the high pressure condition, the flame development near the wall starts at -3 TDC degrees; in the ordinary injection pressure, it starts at 3 TDC degrees.

Corresponding result of the experiment is shown in Fig. 8. The high pressure condition gives a higher peak pressure and earlier peak pressure timing. These features are commonly found in both the simulation and the experiment.

Quantitatively, the simulations agree with the experiment in the peak pressure timing, however, the peak pressure in the simulation is higher than that observed in the experiment. The predicted pressure is about 4 bar higher than experimental result at injection timing (-5 TDC degrees).

At first, the author thought the cause of higher peak pressure was due to simplified model for chemical reaction without thermal dissociation. However, it was found that the peak pressure was not varied even if the chemical equilibrium is counted, though the peak temperature decreases about 150 K when chemical equilibrium is counted. In addition, the simulation gives higher pressure in the combustion chamber before the combustion process initiates. Therefore, there will be other causes of the discrepancy in the predicted pressure and the observed pressure. One possible cause is a blow-by gas in the compression stroke. In the simulation, the effect of blow-by gas was simply neglected. However, there is some blow-by gas in real engine systems.
Figure 9 and fig. 10 show the predicted rate of heat release (ROHR) and the experimentally obtained ROHR respectively. There are some fluctuations on the ROHR curves Fig. 9.

The experimentally obtained ROHR curves were calculated from the pressure indicator obtained in only one cycle of the engine operation, and the fluctuations in Fig. 9 is not essential. Due to these fluctuations, it is difficult to discuss precisely about the propriety of the combustion model, however, the essential effect of the injection pressure on the ROHR seems to be well predicted at least qualitatively. The ROHR in the injection duration increases when the fuel injection pressure increases. In general, the constants used in the equations must be optimized through the comparison of the prediction and the observation. This kind of optimization will be the next step of the study.

For the condition of high pressure fuel injection, a chemical equilibrium was also tested. The ROHR was not significantly varied whether the chemical equilibrium is counted or not.

5 Conclusions

A computational simulation for a DI diesel engine was performed through the code. Two conditions of the fuel injection pressure were tested. The difference of the fuel injection pressure is given in the input parameters of diameter and velocity of initially injected fuel droplets. From the comparison of the results of experiment and simulation, it was concluded as follows.
1. As fuel injection pressure increases, flame develops rapidly, and the timing of flame development near the wall becomes earlier. These are well predicted by the simulation.
2. It was suggested that the combustion process in the injection duration of DI diesel engines is strongly dependent on the behavior of the fuel spray and the characteristics of the turbulence formed by the fuel spray.
3. When injection pressure increases, the peak pressure in the combustion chamber increases and the timing of the peak pressure becomes earlier. Qualitatively, the simulation well predicts these observed effects of high pressure fuel injection. Quantitatively, predicted pressure in the combustion chamber is higher than that observed in the experiment.
4. Results largely depend on the constants of the models and for this reason they have to be closely analysed and carefully interpreted as they may carry a subtle mistake and this would lead to wrong conclusions;
5. The performances of the programme are limited by the models used and for this reason the cases and values studied must be carefully selected so that we do not surpass their applicability range.

As mentioned above, the simulation well predicts the effect of high pressure fuel injection in many aspects qualitatively. Fine computational grids and appropriate submodels for the fuel spray are essentially important in this kind of simulation.

References: