Numerical Investigation on the Effect of Piston Bowl Geometry on Combustion Characteristics of a Heavy-Duty Diesel Engine

Oladapo M. Adeniyi*

Facility of Engineering and Environment, Northumbria University, Newcastle upon Tyne, United Kingdom

Article Info

Article history:
Received May 20th, 2019
Revised Jul 18th, 2019
Accepted August 14th, 2019

Keyword:
Heavy-duty diesel engine
CFD numerical modelling
Efficiency and emissions

Abstract

Today, due to increased efficiency of heavy-duty diesel engines in the global industry, discussion of the performance of these motors, including higher efficiency and lower emissions, is very important. Several methods exist to meet these demands by the diesel engine. Piston bowl geometry deformation strategy is a reliable method for achieving pollutants of lower nitrogen oxides (Nox) and soot and higher yield. This paper has used Converge software to model the CFD for the performance of a Caterpillar 3401 engine with three different piston bowl geometries and various depths and chamfers for one of the geometries. The compression, inlet temperature, and pressure ratio are assumed to be constant. The results of efficiency and pollution of the engine are presented at all stages for analysis, comparisons, and conclusions. The results show that the cylindrical piston bowl geometry has a proper performance in terms of efficiency and the pollutants produced.

1. Introduction

Due to higher compression ratio, diesel or compression ignition (CI) engines have higher efficiency than gasoline engines. Due to rising fuel prices and increasing attention to the problem of air pollution, they have recently been the focus of attention [1-3]. But due to the direct injection of fuel into the cylinder of the diesel engines near the high point of death, part of the cylinder has a rich mixture and the other parts have a poor mixture which results in the formation of soot and nitrogen oxides pollutants, respectively; they have been an obstacle to the development of the engine. One of the potentials for reducing HC and CO pollutants, thus increasing the efficiency of combustion, is the optimization of the piston bowl geometry design [4, 5].

Several studies are provided in the field of effects of geometric factors and fuel injection on the performance of the diesel engine by different methods [6-10]. Wang et al [9] investigated the effects of diesel engine piston bowl geometry, which works with biodiesel fuels, on combustion and pollutants. He studied three different piston bowl geometries and used Kiva code integrated by Chemkin to simulate the combustion process. According to the results, the optimal piston was dependent on speed. Results have shown that at low speeds, SCC has higher indicator work, cylinder pressure, combustion temperature, and consequently, releases more nitrogen oxides; and OCC has better performance at medium and high speed. Kokjohn et al [8] studied RCCI performance at different loads and two modes of single-injector and dual-injection fuel injections using Converge software code and experimental method. The results indicated that the dual-injection strategy has been suitable for low and medium loads, and single-injector strategy has been proposed for higher loads. Park [10] optimized the combustion chamber geometry and the operating conditions for compression ignition engines which work with dimethyl ether fuel. Also, Donateo et al [11] and Genzale et al [12] predicted the combustion process using a genetic algorithm and achieved the combustion chamber and piston bowl curve suitable for methane-fueled engines.

Using numerical methods, Rakopoulos et al [13] studied the piston geometry of a diesel engine and investigated three different piston geometries in three different speeds. Also, Jovanovic et al [14] compared two omega-shaped and cylindrical piston bowls and the spatial distribution of the kinetic energy and the flow pattern in the two-piston bowls in...
different angles of the crankshaft. Cao et al [15] examined the effect of injection time and piston bowl geometry on the pollutants in the PCCI engine and compared three different piston geometries, namely open bowl and vertical sidewall bowl and re-entry bowl. The results showed that open bowl had the highest amount of nitrogen oxides and carbon monoxide, and vertical side had the lowest amount of carbon monoxide and unburned hydrocarbons.

This study simulates combustion in a diesel engine using Converge software code and predicts the yields and pollutants resulted from combustion. The parametric study is initially conducted on three different piston bowl geometries in a fixed compression ratio with different speeds, and the impact of the chamfer size in one of the geometries on the yield and pollutants is studied. Finally, the effect of different depths of piston bowl of one of the geometries on the yield and pollutants is studied.

2. Modelling and Simulation of a Heavy-Duty Diesel Engine

Converge code uses integrated and fixed Cartesian grid and finite volume method. For the transfer of mass, momentum and energy, cell fluids were calculated using second-order central difference method. Time advance implicit method was used for the transient solution. In addition, the conserved form was used to solve the equations. PISO method was applied for coupling pressure and speed at transient solution. To ensure the accuracy of the pressure gradient, partial differential equations of mass, momentum and energy are solved simultaneously.

2.1. Geometry and grid generation

In this article, simulation of the closed cycle was done on the sector grid using periodic boundaries to improve computing efficiency. Computational grid is a sector grid of 60 degrees (one hole of the 6 injector holes used in Caterpillar engines). The sector grid is shown in Figure 1. The number of cells in the lower point of death is 30510 and the size of each cell is 3 mm. in the Converge software, Cartesian grid with organization with base cell size of 4.1 mm is used. Modification of the adaptive network (AMR) was used to correct the grid based on temperature gradient and the speed with minimum cell size of 0.25 mm.

![Figure 1. A section of simulated engine geometry](image)

2.2. Turbulence model

In this study, the spray model, and Lagrangian drop Euler fluid for spray model were used; in addition, for simulation, turbulence model of Reynolds Averaged Navier-Stokes (RANS) was used. In this model, the actual speed of \( u_i \) is decomposed to an average speed component of \( \bar{u}_i \), and a pendulous speed component of \( \dot{u}_i \) which is known as Reynolds decomposition.

\[
  u_i = \bar{u}_i + \dot{u}_i
\]  

Putting Reynolds decomposition in equations of constant density flows, equations with the average quantity and additional turbulence expressions that must be modelled, are obtained. Density weighting is done using Favre averaging which is expressed by Mode mark. Favre average speed is expressed as follows:

\[
  \bar{u}_i = \frac{\rho u_i}{\rho}
\]  

Reynolds decomposition is expressed as follows using Favre averaging quantities:

\[
  u_i = \bar{u}_i + u_i''
\]  

In this relation, \( u_i'' \) is the average speed component according to the Favre average speed. By averaging the Navier-Stokes equations, and placement in the Favre averaging decomposition to simplify the expressions, a form of equation...
similar to the laminar state was achieved, but with Favre averaging quantities for variables and an additional expression for describing the effect of the turbulence. The averaged form of the conservation equations of mass, momentum and energy are respectively as follows:

\[
\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial \bar{\rho} \bar{u}_i}{\partial x_i} = 0 \quad (4)
\]

\[
\frac{\partial \bar{\rho} \bar{u}_i}{\partial t} + \frac{\partial \bar{\rho} \bar{u}_i \bar{u}_j}{\partial x_j} = - \frac{\partial \bar{P}}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} \bar{\rho} \frac{\partial \bar{T}}{\partial x_i} \right] + \frac{\partial}{\partial x_j} \left( \bar{\rho} \bar{u}_i \bar{u}_j \delta_{ij} \right) + \frac{\partial}{\partial x_j} \left( \bar{\rho} \bar{u}_i \bar{e} \right) \quad (5)
\]

\[
\frac{\partial \bar{\rho} \bar{e}}{\partial t} + \frac{\partial \bar{\rho} \bar{u}_i \bar{e}}{\partial x_i} = -P \frac{\partial \bar{u}_i}{\partial x_i} + \bar{\rho} \bar{u}_i \delta_{ij} \bar{u}_j + \frac{\partial}{\partial x_j} \left( K \frac{\partial \bar{T}}{\partial x_j} \right) + \frac{\partial}{\partial x_j} \left( \rho D \sum_m \bar{h}_m \frac{\partial \bar{Y}_m}{\partial x_j} \right) + \frac{\partial}{\partial x_j} \left( \bar{\rho} \bar{u}_i \bar{e} \right) \quad (6)
\]

\[
\frac{\partial \bar{m}}{\partial t} + \frac{\partial \bar{m} \bar{u}_i}{\partial x_i} = \frac{\partial}{\partial x_j} \left( \bar{\rho} D \frac{\partial \bar{Y}_m}{\partial x_j} \right) + \frac{\partial}{\partial x_j} \left( \bar{\rho} \bar{u}_i \bar{Y}_m \right) \quad (7)
\]

Comparing the averaged form of the momentum equation with the form of laminar, we find that each averaged expression has a similar laminar expression. However, the averaged momentum equation has an additional expression called Reynolds stress tensor as follows:

\[
\tau_{ij} = -\bar{\rho} \bar{u}_i \bar{u}_j \quad (8)
\]

Furthermore, the averaged equations of energy and species have Reynolds turbulence expressions, respectively as follows:

\[
q_t = -\bar{\rho} \bar{u}_i \bar{e} \quad (9)
\]

\[
m_{m,t} = -\bar{\rho} \bar{u}_i \bar{Y}_m \quad (10)
\]

In these equations, \( q_t \) is the additional thermal charge in the energy equation, and \( m_{m,t} \) is the additional mass flux in the species equation of \( m \). Given that there is no available equation for Reynolds expression, the system cannot be solved. To solve the system of equations, for oscillating expressions of Reynolds, additional theory should be introduced. This is usually done by gradient approximation; between the penetration rate increased by the rate of molecular effusion and the turbulence that is proportional to the spatial gradient of transitional quantities, it uses a similarity that is as follows in the formula:

\[
-\bar{\rho} \bar{u}_i \phi'' = \frac{\mu_t}{\sigma_{\phi}} \frac{\partial \bar{\phi}}{\partial x_i} \quad (11)
\]

In this equation, \( \phi \) is the transitional scalar quantity, \( \mu_t \) is the turbulence viscosity, and \( \sigma_{\phi} \) is a number without turbulent dimension, that relates the \( \mu_t \) to turbulence proportionality coefficient for \( \phi \). Applying the gradient approximation, the Reynolds oscillating expressions in equation (3-22), are respectively as follows:

\[
-\bar{\rho} \bar{u}_i \phi'' = \mu_t \left( \frac{\partial \bar{u}_i}{\partial x_i} + \frac{\partial \bar{u}_i}{\partial x_j} \delta_{ij} \right) \frac{2}{3} \mu_t k \delta_{ij} - \bar{\rho} \bar{u}_i \phi'' = \bar{\rho} \alpha_x \frac{\partial \bar{e}}{\partial x_i} = \frac{\mu_t}{\rho \alpha_x} \frac{\partial \bar{e}}{\partial x_i} \quad (12)
\]

\[
-\bar{\rho} \bar{u}_i \bar{Y}_m'' = \rho D_t \frac{\partial \bar{Y}_m}{\partial x_i} = \mu_t \frac{\partial \bar{Y}_m}{\partial x_i} \quad (13)
\]

Where \( \alpha_x = \) Turbulent thermal penetration, \( D_t = \) Turbulent species penetration, and \( k = \) Turbulent kinetic energy which will be discussed then. In equation (14), \( pr_t \) is the Turbulent Prandtl number, i.e.:

\[
pr_t = \frac{\mu_t}{\rho \alpha_x} \quad (14)
\]

And \( Sc_t \) is Turbulent Schmidt number as follows:

\[
Sc_t = \frac{\mu_t}{\rho D_t} \quad (15)
\]

In all like simulations presented here, for \( Pr_t \) and \( Sc_t \), we use constant values of 0.9 and 0.78 that are traditional values for air.
The most common method to model the turbulent viscosity which is also used here is a two-equation model. In a two-equation RANS model, for the turbulence kinetic energy of $k$, and turbulence dissipation rate of $\epsilon$, two additional transport equations are solved and used to define the turbulence viscosity and guidance for each computational cell. Turbulence kinetic energy is defined as follows:

$$ k = \frac{1}{2} u_i^2 u_i' $$

(16)

Turbulence dissipation rate is a rate at which the smallest vortex turbulence kinetic energy is converted into molecular thermal energy. Then, the turbulence viscosity is obtained by values for $k$ and $\epsilon$ in the centre of each cell.

$$ \mu = \frac{C_{\mu} k^2}{\epsilon} $$

(17)

In the equation above, $C_{\mu}$ is the model constant. Then, using the definition of $k$, it is expressed as follows:

$$ \tau_{ij} = \mu_\tau \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_l}{\partial x_l} \right) $$

(18)

To simulation here, RNG turbulence model used was (RNG) $k-\epsilon$ renormalization model of Han and Reitz. Transport equation for $k$ is as follows:

$$ \frac{\partial \rho k}{\partial t} + \frac{\partial \rho u_i k}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \frac{\mu}{\rho} \frac{\partial k}{\partial x_i} + \frac{\partial}{\partial x_i} \left( \frac{k}{\rho} \frac{\partial \rho}{\partial x_i} \right) - \rho \epsilon + S_k \right) $$

(19)

Where $Pr_k$ is the Prandtl number which describes the viscous penetration ration to the penetration of $k$, and $S_k$ is the source expression that is used by the spray model to show the transition of turbulent kinetic energy from the gas phase to liquid particles. Transport equation for $\epsilon$ is as follows:

$$ \frac{\partial \rho \epsilon}{\partial t} + \frac{\partial \rho u_i \epsilon}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \frac{\mu}{\rho} \frac{\partial \epsilon}{\partial x_i} \right) + \frac{c_{\epsilon 3} \rho \epsilon}{Pr_k} \frac{\partial u_i}{\partial x_i} + \left( c_{\epsilon 1} \frac{\partial u_i}{\partial x_i} \tau_{ij} - c_{\epsilon 2} \rho \epsilon + c_{\epsilon S} S_j \right) \frac{\epsilon}{k} - \rho R $$

(20)

In RNG model, $R$ expression is expressed as follows:

$$ R = \frac{C_{\mu} \eta^3 (1 - \eta / \eta_0) \epsilon^2}{(1 + \beta \eta^4) k} $$

(21)

$\eta$ is as follows:

$$ \eta = \frac{k}{\epsilon} \left| S_j \right| = \frac{k}{\epsilon} \sqrt{2S_{ij} S_{ij}} $$

(22)

In equation $\epsilon$, $Pr_k$ describes the viscous penetration ratio to the penetration of $\epsilon$, and $c_{\epsilon 1}$, $c_{\epsilon 2}$, and $c_{\epsilon 3}$ are the model constants. The modified RNG model is $k-\epsilon$ standard model ($R = 0$) because the model constants can be obtained analytically from isotropic turbulence correlations, so, there is no need to adjust the model constants.

In turbulence models based on RANS, if the grid is properly resolved for showing the penetration rates of turbulence, a turbulent length scale is defined as follows:

$$ l_e = \frac{C_{\mu} k^3 / \epsilon}{2} $$

(23)

In this equation, $C_{\mu}$ is the model constant. In areas where combustion occurs to ensure accuracy in combustion rate, cell width should be less than $l_e$. In most cases, the cell distance of 0.25 mm can do this.

2.3. Combustion model of SAGE

SAGE model uses CVODES solver (it solves the initial value problems for the system of ordinary differential equations). This model solves the response rate numerically and is applicable to different regimes of combustion such as mixing-controlled, premixed, and ignition. A multistage chemical reaction mechanism can be written as follows:

$$ \sum_{m=1}^{M} V_{m,n} X_m = \sum_{m=1}^{M} V_{m,n} X_m $$

(24)
Where \( v'_{m,r} \) and \( v''_{m,r} \) are stoichiometric coefficients for reactants and products. If the number of species present in the reaction is \( m \) and the total number of reactions is \( R \), the net reproductive rate of \( m \) is obtained from the following equation where \( M \) is the total number of species.

\[
\omega_m = \sum_{r=1}^{R} v'_{m,r} q_r
\]

For \( m=1,2,\ldots,M \)

\[
v''_{m,r} = v''_{m,r} - v'_{m,r}
\]

(25)

\( M \) is the total number of species. \( q_r \) is the variable of progress rate for the reaction \( r \), which is obtained from the following equation:

\[
q_r = k_f \prod_{m=1}^{M} [X_m] \cdot v'_{m,r} - k_r \prod_{m=1}^{M} [X_m] \cdot v''_{m,r}
\]

(26)

In the equation above, \([X_m]\) is the molar concentration of \( m \) species, \( k_f \) and \( k_r \) are the coefficients of sweep reaction rate. In this model, the response rate is obtained from the Arrhenius equation of 

\[
\frac{d[X_m]}{dt} = \omega_m
\]

(28)

\[
\frac{dT}{dt} = \frac{V \cdot dP}{d} - \sum_{m} \left[ \tilde{h}_m \cdot \omega_m \right]
\]

(29)

\[
\frac{dP}{dt} = \frac{\sum_{m} \left[ [X_m] \cdot \tilde{c}_{p,m} \right]}{\rho}
\]

In the equations above, \( V, T, \) and \( P \) are volume, temperature, and pressure, respectively. \( \tilde{h}_m \) and \( \tilde{c}_{p,m} \) are specific molar enthalpy and specific molar heat under constant pressure of \( m \)th species. These equations are solved in each Converge time step, and reactive species are updated.

### 2.4. Multi-zone combustion model to accelerate chemical kinetics

To speed up the detailed chemical kinetics solution, the multi-zone model of Baba Dimopoulos et al. was used. In the two-dimensional zoning strategy, zoning is done based on two variables of temperature and equivalent ratio of cells. The average temperature and composition are determined for each zone in order to determine the mix thermodynamic state in that zone. Then, the SAGE chemistry solver is used on each zone. Chemical kinetic equations for a homogeneous closed volume reagent for each zone is written as follows:

\[
\frac{\partial T}{\partial t} \bigg|_{zone} = -\frac{1}{\rho \tilde{c}_{v}} \sum_{k=1}^{K} \hat{\omega}_k W_k \epsilon_k \bigg|_{zone}
\]

(30)

\[
\frac{\partial Y_k}{\partial t} \bigg|_{zone} = \frac{\hat{\omega}_k W_k}{\rho} \bigg|_{zone} \quad (k = 1,\ldots,K)
\]

(31)

Where \( \hat{\omega}_k \) is the production/consumption rate of \( k \), and \( W_k \) and \( \epsilon_k \) are molecular weight and specific internal energy of \( k \), and \( \tilde{c}_{v} \) is the specific heat at constant volume of gas mix. Progress equivalence ratio \( (\varphi) \) and total \( (\varphi^T) \) are defined as follows:
\[
\phi = \frac{2C^a_{-CO_2} + H^a_{-H_2O}}{O^a_{-CO_2-H_2O}}, \varphi' = \frac{2C^a + H^a}{O^a}
\]  

(32)

Where \( C^a \) and \( H^a \) are the number of carbon and hydrogen atoms, respectively; \( C^a_{-CO_2} \) and \( H^a_{-H_2O} \) are the number of carbon atoms, excluding CO2, and the number of hydrogen atoms with the exception of H2O. Multi-zone modelling accuracy increases by reducing the size of the zone, however, the cost of computing increases.

Converge uses the multi-zone mapping technique of Baba Dimopoulos and his colleagues. This mapping is based on the "ch" value of each cell in the zone. This value is defined as follows:

\[
ch_{cell} = 2C^a_{-CO_2} + H^a_{-H_2O}
\]  

(33)

The value of "ch" for all the cells and zones is calculated before the start of chemistry calculations in the zones. The sum of all values of "ch" of each cell will be equivalent to the value of "ch" of that zone in a specified zone. 

\[
m_{k,cell} = \frac{ch_{cell}}{ch_{zone}}m_{k,zone}
\]  

(34)

Converge uses the equation above to calculate all species except CO\(_2\), H\(_2\)O, O\(_2\) and N\(_2\). Converge also tries to maintain the atoms of carbon, hydrogen and oxygen in each cell by resetting the mass of CO\(_2\), H\(_2\)O and O\(_2\).

2.5. Emissions model

Soot pollution is predicted by Hiroyasu-NSC empirical model. \( M_s(g) \) soot formation in a computational cell is determined via a single-stage competition between the soot formation rate \( M_{sf}(g/l) \) and oxidation rate \( M_{sox}(g/l) \):

\[
\frac{dM_s}{dt} = M_{sf} - M_{sox}
\]  

(35)

Soot formation rate uses Arrhenius expression and its oxidation rate is based on Nigel and Strickland carbon oxidation model.

Finally, the following equation is used in Converge to update soot density in every computational cell and each time step:

\[
\rho_s^{n+1} - \rho_s^n = \left( \frac{SF \rho_{form}}{SO} - \rho_s^n \right) + \left[ 1 - \exp(-SO\Delta t) \right]
\]  

(36)

Where \( \rho_{form} \) is the density of soot formation species. \( SF \) and \( SO \) are defined as follows:

\[
SF = A_s P^{0.5} \exp\left( \frac{E_s}{RT} \right)
\]  

(37)

\[
SO = A_w \frac{6MW_s}{\rho_s D_s} \frac{1}{R_{total}}
\]  

(38)

Where \( M_s \), \( R_{total} \) and \( MW_s \) are soot mass, overall response rate, and molecular weight of carbon, respectively. Soot density \( (ps) \) and Soot particle diameter \( (Ds) \) are considered to be 2 g/cm\(^3\) and 0.025 Micro, respectively. In the equation of soot formation rate, there are two adjustable constants of Asf and Esf. For combustion strategies such as HCCI and RCCI, Asf 700 and Esf 1250 are considered. The Extended Zel’dovich model was used to predict the formation of NO\(_x\) pollutant.

3. Validation

3.1. Model specifications and comparison with experimental results

The 2.44-litre single-cylinder Caterpillar 3401 (SCOTE) heavy-duty diesel engine is selected for all modelling and simulations. Engine specifications and operating conditions used in the simulation are respectively listed in Table 1.
Table 1. Geometric specifications of the single-cylinder Caterpillar 3401 engine

<table>
<thead>
<tr>
<th>Engine Type</th>
<th>Caterpillar SCOTE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Diameter × stroke (cm)</td>
<td>16.51 × 13.72</td>
</tr>
<tr>
<td>Connecting rod length (cm)</td>
<td>21.16</td>
</tr>
<tr>
<td>Displacement volume (L)</td>
<td>2.44</td>
</tr>
<tr>
<td>Compression ratio</td>
<td>16.1:1</td>
</tr>
<tr>
<td>Rotation ratio</td>
<td>1.38</td>
</tr>
<tr>
<td>Piston bowl type</td>
<td>Mexican hat</td>
</tr>
<tr>
<td>Inlet valve opening (°ATDC)</td>
<td>-335</td>
</tr>
<tr>
<td>Inlet valve closing (°ATDC)</td>
<td>-143</td>
</tr>
<tr>
<td>Outlet valve opening (°ATDC)</td>
<td>130</td>
</tr>
<tr>
<td>Outlet valve closing (°ATDC)</td>
<td>-355</td>
</tr>
</tbody>
</table>

Common rail injector specifications

| The number of injector holes | 6 |
| Injectors hole diameter (μm) | 250 |
| Fuel spray angle (°) | 145 |

The results of the simulation in this part is obtained on the Caterpillar 3401 single-cylinder engine. These results are validated by the results reported by Tess et al. [16] in constant 9 bar. As shown in Figure (2), the results obtained are of high accuracy.

Figure 2. Comparison of the simulation results with the results of Tess et al. [16] in the combustion of the diesel engine

3.2. Independency of solution from the grid

To ensure the accuracy of the solution, it must be independent of the grid. To evaluate the independency of the solution from the grid, four networks with different sizes of 1.2, 1.3, 1.4, and 1.6 mm were selected. As Figure 3 shows, the results of pressure in the cylinder and heat release rate are virtually independent of the size of the selected grid which shows the accuracy of the solution and its independency on the size of the grid.
4. Results and discussions

Figure 4 shows three selected profiles of bowl including the stoke, bathtub and cylindrical bowls that are selected according to technical literature and engine specifications (Caterpillar). In the computational grid for different geometries about 12,000 was meshed for each geometry at the high point of death. As previously mentioned, the gap in the piston bowl geometry is considered. Parametric study was performed at three speeds of 800rpm, 1300rpm and 1800rpm. Comparison of the graphs of pressure in the cylinder and heat release rate in different rotations are shown for three geometries in Figure 5. Comparison results of the performance and pollution are also shown in Figure 6.

The results obtained showed that the bowl geometry has no significant impact on the pressure and heat release and therefore the combustion phase; but with increasing engine speed, the time of ignition is affected. High engine speeds provide less time for the occurrence of chemical reactions; this is a factor that reduces the pressure and temperature. It will also lead to less time for heat transfer, which increases pressure and temperature.

According to Figure 5, bathtub and cylindrical geometries produce the highest temperature and pressure inside the cylinder. As Figure 5 shows, at different speeds, the graphs of the pressure of these two geometries were overlapped, but at 1800rpm speed, cylindrical geometry is slightly higher than other geometries. Combustion chamber geometry has a significant impact on the formation of squish flow and thus the processes of evaporation and mixing. As it is clear, stoke and cylindrical geometries have relatively narrow input areas than the deeper part of the combustion chamber compared to the bathtub geometry. It considerably increases the velocity of the air going into the piston bowl, and make a powerful squish flow (because the input air rate is the same, and cross-sectional area in stoke and cylindrical geometries is less).
Figure 5. Comparison of the graphs of pressure and heat release rate for three different geometries in the diesel combustion and speeds of a) 800rpm, b) 1300rpm, and c) 1800rpm.
Moreover, less piston bowl depth lead to less heat dissipation (because the ratio of the surface is less than the volume), and consequently, the average temperature rises, and more nitrogen oxides are released. In the results obtained, bathtub and cylindrical geometries have shown the same behavior.

Figure 6 shows temperature contours on a page in accordance with the spray direction at different speeds for different profiles of piston bowl. Figure 7 shows the calculations of efficiency results of three geometries at 800-1300-1800rpm.
5. Conclusions

In this study, Converge software was used to simulate the combustion behaviour of diesel engines. In the first section, we numerically investigated the effect of three-piston bowl geometries (stoke, cylindrical and bathtub) on efficiency and pollutants. The second section examined the effect of chamfer size in stoke geometry at 1300rpm on the engine’s performance. In the third section, we explored the effect of different depths of cylindrical geometry on efficiency and pollutants, and the results are compared with each other; the results of this study are summarized below.

- In bathtub and cylindrical geometries, high engine speeds relatively increase pressure and temperature and also lead to more time for heat transfer. The behaviour of these two geometries are almost similar to each other, but in stoke (omega-shaped) geometry, maximum pressure and temperature are achieved at 1300rpm, and the result obtained in this speed for the stoke geometry is very close to two other geometries, but it faces pressure and temperature reduction at 1800rpm.
- Two pollutants of nitrogen oxides and soot are very important in diesel engine design, and as it is clear from the results, the impact piston bowl geometry on nitrogen oxides and soot is high.
- Cylindrical geometry has the best geometry and emission performance compared with the other two geometries (stoke and bathtub) at all speeds.
- Two bathtub and cylindrical geometries have similar behaviours, and only at 1300rpm, the stoke geometry gets close to these two geometries.

References


