ANALYSIS OF TEMPERATURE, PRESSURE AND SOOT DENSITY ON A SINGLE CYLINDER DIESEL ENGINE

M.H. Mohd Hafidzal[1,2*], N.N. Mohamad Isa[2], A. Roslizar[1,2], M.Z. Akop[1,2], M. S. Zakaria[1,2], J. Ridhwan[1,2]

[1]Centre for Advanced Research on Energy,
Universiti Teknikal Malaysia Melaka, 76100 Durian Tunggal,
Melaka, Malaysia

[2]Faculty of Mechanical Engineering, Universiti Teknikal Malaysia Melaka,
76100 Durian Tunggal, Melaka, Malaysia

ABSTRACT

In order to develop and improves the efficiency of the diesel engine, various of works and researches had been done. The most commonly research studies are internal combustion engine and the emmisions. This study focused on temperature, pressure and soot in a single cylinder diesel engine. Studies using experimental methods require a lot of cost, manpower, time and high technology equipment. Therefore, the simulation such as Computational Fluid Dynamic (CFD) can be used to conduct a study on the model engine by entering a few geometry settings and types of simulations to be performed. By using CFD FLUENT simulations, this study aims to investigate the distribution of temperature, pressure and soot formation in diesel internal combustion engine. In this study, CFD simulations were conducted on an internal combustion diesel engine model in 2D by using the eddy-dissipation model and non-premixed combustion. All engine parameters and dimensions have been taken from the actual engine model-KM186 KIPOR diesel engine. Results of simulations conducted have found that the maximum pressure obtained was 71.78MPa pressure with a temperature derived at 918K. Both the pressure and maximum temperature obtained from the crank angle of 720° (+0° ATDC).Soot formation is increased dramatically immediately after igniting fuel into the combustion chamber. During combustion, the average value of the maximum soot density is 0.023284 kg/m3 at 732° (+12° ATDC).

KEYWORDS*: Internal combustion engine, Temperature, Pressure, Soot and After Top Dead Centre (ATDC)*

* Corresponding author email: hafidzal@utem.edu.my

1.0 INTRODUCTION

A development in the automotive sector today, especially in diesel engine has resulted in competition among vehicle manufacturers in order to making quality vehicles and always has improved. Soot is among the negative effect that produced from the diesel engine. The soot influence the lifelong of lubricant oil (Mohd Hanafi et al., 2013; Mahmood, 2011). Temperature and pressure are the main factors contributing to soot formation. Therefore, this study focusing on the analysis of the temperature, pressure and soot. There are various methods and ways that can be done and one of the methods is by running a computer simulation of the engine to be studied. Some researchers used kiva-3V software to simulate soot formation (Bonatesta, 2007). However, this study used a CFD simulation of ANSYS Fluent Use of computer simulations such as CFD have a lot of advantages in terms of time, cost and manpower compared with experimental methods (Pang, 2009). One of the CFD a simulation is often done on the vehicle is simulated on the model of an internal combustion diesel engine. CFD simulations will run on engine model by analyzing various aspects such as geometry, combustion processes occurring in the combustion chamber, fuel, chemical reactions, taking into account the mathematical methods involved and the type of gas being released (Canonsburg, 2012). Therefore, this study will conduct the computational simulation of CFD on the diesel internal combustion engine focus on the temperature and pressure distribution also the soot emission in the diesel engine cylinder combustion chamber.

2.0 SIMULATION SET-UP

In this study, a single-cylinder of 4-stroke diesel engine with a compression ratio 19:1 was used. All the parameters and dimensions of the engine based on KIPOR KM-186 diesel engine model. The non-premixed combustion and eddy dissipation model of ANSYS FLUENT CFD Software were set up This study focuses on the simulation of a diesel internal combustion engine where the type of geometry used in the form of 2D including the part of piston head and cylinder where it is consists of 3697 cells, 7472 faces and 3800 nodes. This simulation will be conducted starting from the phase of the compression stroke until power cycle. Based on the results generate from simulation, pressure and temperature distribution also the soot emission expected to be obtained. Table 1 shows the characteristic of the engine.

Table 1. Internal Combustion (IC) engine parameter

Engine parameter	Value
Crank Shaft Speed (RPM)	3000
Starting Crank Angle (deg)	360
Crank Period (deg)	720
Crank Angle Step Size (deg)	0.5
Piston stroke (mm)	72
Crank radius (mm)	36
Connecting rod length (mm)	140

Models setup, standard k-ε turbulent model was enabled to allow the eddy-dissipation setting on Species Model since this equation suitable for turbulent compressible flow (Sureshbabu, 2013). Auto-ignition model was enabled to follow by discrete phase modelled setting (Kongre, 1010). For the diesel combustion models it is typically a compressible turbulent flow where diesel-air fuel is chosen as a fuel in Material database (Karunanidhi, 2013). In this study, the injection parameter and specification are shown in Table 2 and the single injection type was chosen. At the solution setup, since the simulation uses the pressure-based segregated logarithm Pressure-Implicit with Splitting of Operation (PISO) scheme is selected.

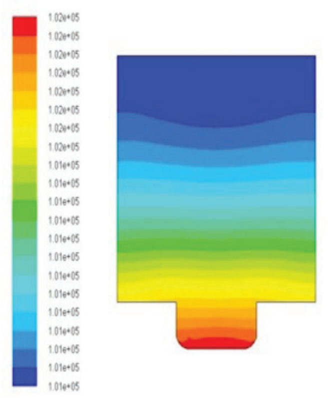
Table 2. Parameter for Injection Point Properties

Injection Parameter	Value
X-position	43
Y-position	5
Diameter (mm)	0.268
Temperature (K)	341
Start Crank Angle (deg)	710
Stop Crank Angle (deg)	725
Flow Rate (kg/s)	0.001044
Y-velocity (m/s)	468

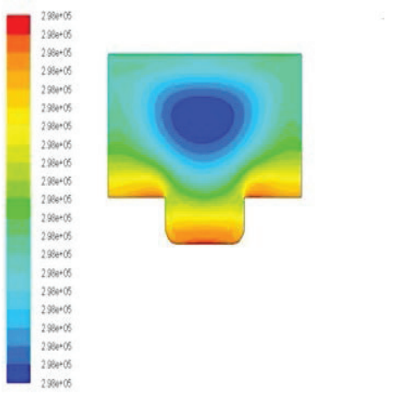
3.0 RESULT AND DISCUSSION

The modelled CI diesel engine - single cylinder was simulated by FLUENT using the model of Eddy Dissipation and non-premixed combustion that focus on the second stroke cycle of CI diesel engine operation where combustion operation occurs starting from the compression stroke cycle at 540° Crank Angle (CA) until 900° Crank Angle (CA).

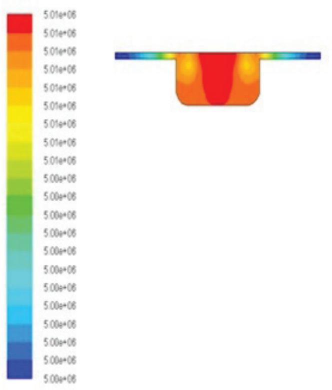
Figure 1 shows the contour of the static pressure process and Figure 2 shows the contour of static temperature in combustion chamber start from 550° CA to 900° CA. At 550° CA (+10° ABDC) where the piston located than near to bottom dead centre (BDC), the piston start to compress the air until it reaches near to TDC which is 720° CA. At 710° (-10° ATDC) during the compression stroke fuel injection through the combustion chamber begin at high speed of 468 m/s. Diesel, liquid fuel jet, then atomized into a droplet, then evaporates become a fuel vapour that mixed with air within combustion proportions where the power stroke cycle begins. At this point, temperature and pressure are above fuel ignition point until some short delays of 0.36 s pre-exponential period. An auto-ignition of non-uniform fuel air mixture initiates the combustion process where the pressure and temperature reached a maximum peak pressure of 720° CA (0° ATDC). The flame produced from the combustion then will spread rapidly through the remains injection mixed-air fuel to burn. Mixing of the air-fuel and burning gasses combustion lead to the expansion process to start and push the piston down back to Bottom Dead Centre (BDC) at 900°CA.



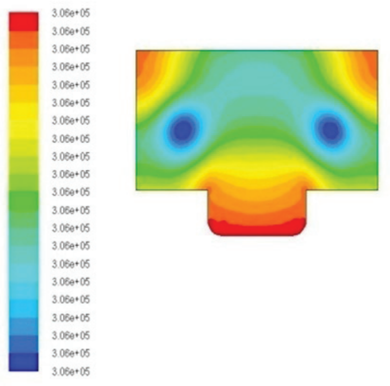
(a) 550 ° CA (+10 ° ABDC)



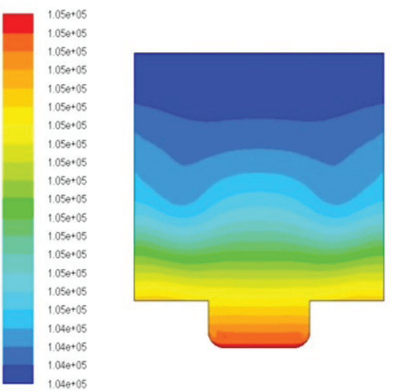
(b) 630º CA (+90 ° ABDC)



(c) 720 ° CA (0 ° ATDC)

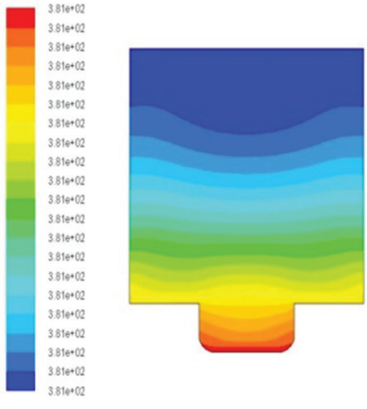


(d) 810 ° CA (+90 ° ATDC)

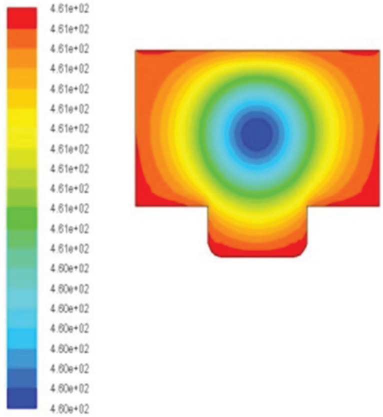


(e) 900 ° CA (BDC)

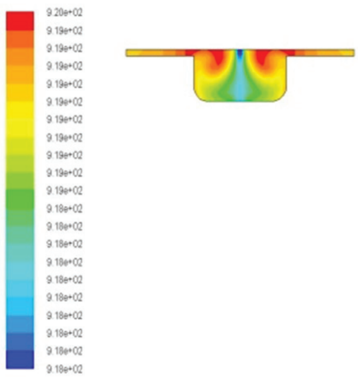
Figure 1. Contours of static pressure



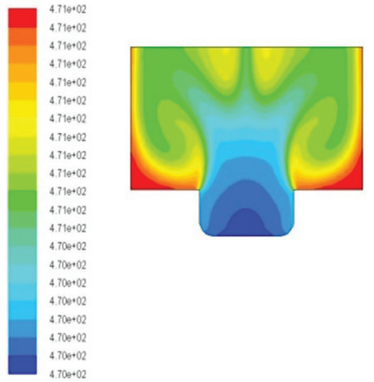
(a) 550 ° CA (+10 ° ABDC)



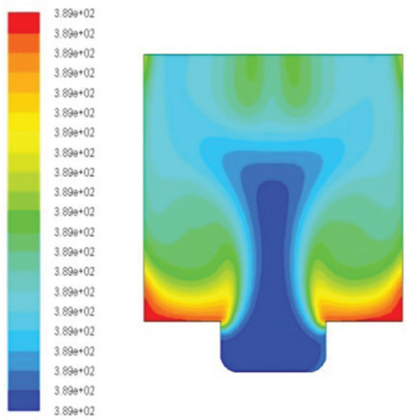
(b) 630° CA (+90 ° ABDC)



(c) 720 ° CA (0 ° ATDC)



(d) 810 ° CA (+90 ° ATDC)



(e) 900 ° CA (BDC)

Figure 2. Contours of static temperature

Based on the simulation data, the value of temperature and pressure for each every step of crank angle can be determined by plot the graph during the simulation. Figure 3 shows the graph of pressure against flow time where from the graph it showed that the value of the pressure is increased from 540°(0° ABDC) at 70,150Pa until it reached a maximum peak pressure TDC which is 720° CA at 7178911Pa (≈72 bar) where the combustion occurred. After the pressure reached a peak pressure, the pressure will gradually decline because of the high pressure push piston back to BDC.

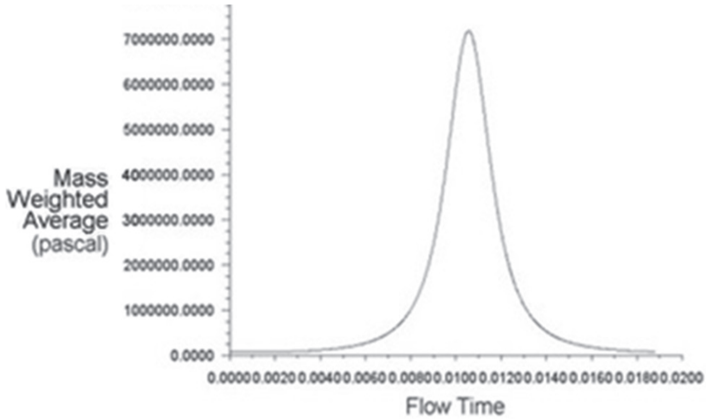


Figure 3. Graph of pressure against flow time

Another graph of temperature against flow time also has been generated from as shown in Figure 4. The pressure showed that the temperature gradually increases start at BDC which is 383 K until it reached a maximum peak temperature at 919 K. At this peak pressure, the fuel is directly injected through the piston bowl to perform combustion in the combustion chamber.

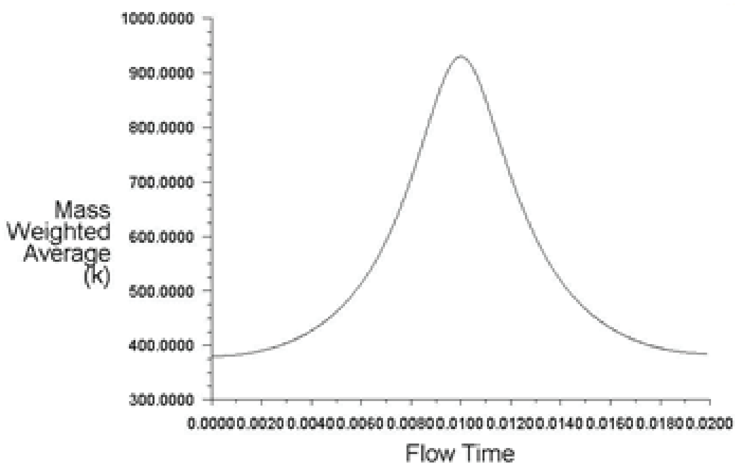


Figure 4. Variation of temperature against flow time

To validate the data obtained from the simulation, the result obtains from the simulation is compared to the result obtained by using the simulation of MATLAB. Figure 5 shows the result of pressure against CA by using the Fluent and MATLAB simulation. There are slightly different between the values of pressure by using these both simulations. For the value of peak pressure, the simulation by using MATLAB got the value of 7,150 kPa compared to the result obtain by using Fluent which is 7,179 kPa with the difference value about 0.4%.

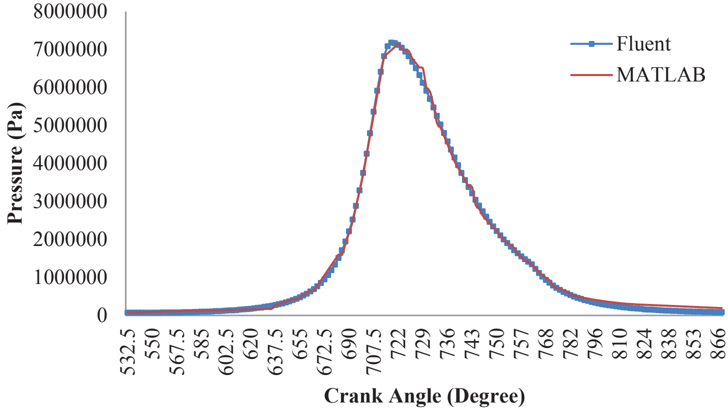


Figure 5. Graph of Pressure against crank angle (CA)

Figure 6 shows the variation of temperature against crank angle. The maximum peak pressure obtained from MATLAB simulation is 1004K with the difference value of 9.3%. This difference value can be accepted since it is less than 10% according to the experimental standard. The difference between the temperature value by using MATLAB and Fluent simulation is because of the some different settings before the simulation start. For MATLAB this simulation based on the equation and the some parameter of injection and the type of fuel did not consider.

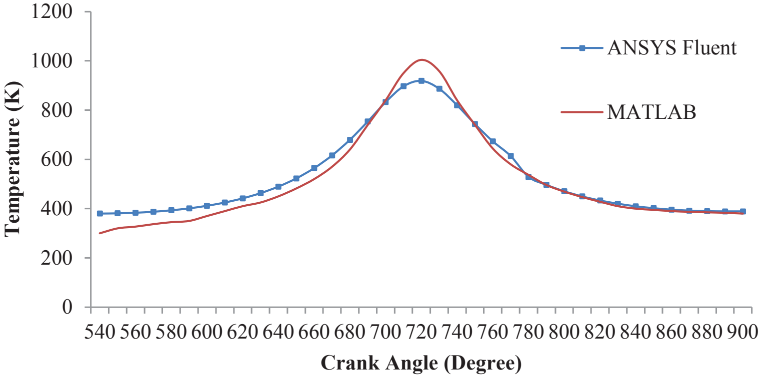


Figure 6. Effect of temperature against crank angle

Figure 7 shows the effect of soot formation density in the combustion chamber against the crank angle by considering diesel air fuel as the fuel and oxygen as an oxidant. Soot formation starts while the fuel injections begin and mixed with hot compressed air. The soot formation increased dramatically once the injection begins at 710º CA (-10º BTDC) with average soot density of 0.00788 kg/m3 until it reached its maximum soot density at 732° (+12° ATDC) with the maximum soot average density of 0.023284 kg/m^3. After soot density reached the maximum density at 732° (+12° ATDC), the amount of soot density will gradually decline until the combustion repeated for th next power stroke cycle.

Soot start to form at 710° CA (-10° BTDC) where the pressure is 5,910,713 Pa and temperature is 899 K. The soot reached the maximum density at 732° (+12° ATDC) with the pressure at 7,046,078 Pa and temperature of 960 K. According to previous studies, the formation of soots occur at a temperature of 1000 K to 2800 K and pressure of 5000 kPa up 10000 kPa (Heywood, 1988). Therefore, the results of the simulation Fluent does not contradict compared to the earlier results although the temperature is a little different but it is open from huge difference and the pressure is on budget pressures on previous studies.

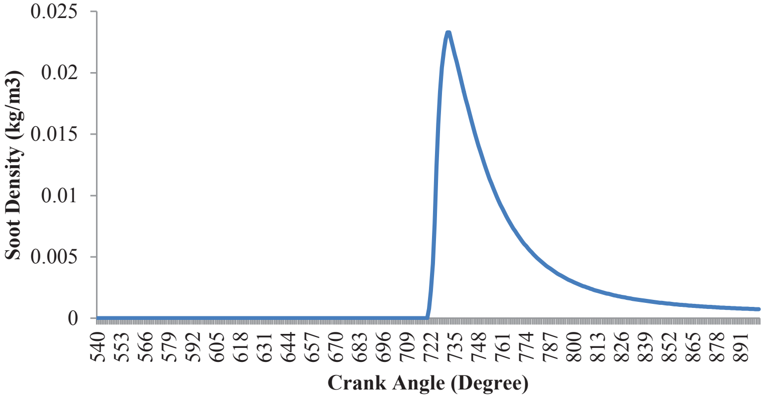


Figure 7. Graph of average soot density against crank angle

4.0 CONCLUSION

From this study of the 2D single diesel engine cylinder by using computational fluid dynamic, the results which are the pressure and temperature distribution during combustion in the combustion chamber, the amount of soot emission from the combustion and maximum peak pressure and temperature are obtained. Fluent simulation results from the model of the engine, the pressure graph shows that the maximum static pressure is 7178911 Pa at 720° CA (+0° ATDC) and the maximum static temperature of 913K at 720° CA (+0° ATDC). Comparison of the temperature and pressure are performed by using MATLAB simulation and it shows the differences value in the maximum static pressure are 0.4% and the difference between the static temperatures is 9.7%. This difference can be accepted based on standard experimental comparison of below 10%. On the density of the production of soot, soot production increased dramatically starting at 710° CA (-10° BTDC) to 732° (+12° ATDC) with the maximum average density of soot that of 0.023284 kg/m^3. The maximum value of the average density of soot, the pressure is at 5,910,713 Pa and the temperature is at 899 K.

In conclusion, the objective of this study is to determine the temperature and pressure of the ANSYS simulation as a whole has been achieved in terms of the movement of the figure or the fuel combustion process. Based on a computer simulation model of a real engine like CFD, it helped in facilitating the study of the combustion engine and to study the ability of the engine model. This is because, time and cost can be reduced and if the study requires repeated testing CFD simulations can be used repeatedly by simply changing a few parameters.

ACKNOWLEDGEMENT

The authors gratefully acknowledge contributions from the members of the Green Technology Vehicles (GTeV) and financial support from Universiti Teknikal Malaysia Melaka and The Ministry of Higher Education, Malaysia (MoHE) under Dana Pembudayaan Penyelidikan (RAGS), grant no.: RAGS/2013/FKM/TK01/02/B00041.

REFERENCES

Bonatesta, F., Rocca, A. L., Shayler, P.J., & Wahab, E. (2007) The Influence of Swirl Ratio on Soot Quantity and Distribution in the Cylinder of a Diesel Engine, Third European Combustion Meeting ECM.

Canonsburg, T. D. (2012). *Internal Combustion Engines in Workbench Vol. 15317*, 724–746.

Karunanidhi, S. G., Melvinraj, C. R., P, S. D. K., & Rao, G. S. (2013). CFD Studies of Combustion in Diesel Engine, 3(4), 827–830.

Kongre, U. V, & Sunnapwar, V. K. (2010). CFD Modeling and Experimental Validation of Combustion in Direct Ignition Engine Fueled with Diesel, 1(3), 508–517.

Mohd Hanafi, M. H., Wan Mohd, F. W. M., Roslizar, A., Zakaria, M. S., Saadun, M. N. A., & Rafeq, S. A. (2013). Analysis of Soot Particle Movement in Diesel Engine under the Influence of Drag Force. *Applied Mechanics and Materials, 393*, 275–280. doi:10.4028/www.scientific.net/AMM.393.275.

Mahmood, W. M. F. W. (2011). Computational Studies of Soot Paths to Cylinder Wall Layers of a Direct Injection Diesel Engine. PhD Tesis, Department of Mechanical, Materials and Manufacturing Engineering, The University of Nottingham.

Pang, K. M., Ng, H. K., & Gan, S. (2009). *Light-Duty Diesel Engine Modelling with Integrated Detailed Chemistry in 3-D CFD Study*, (December), 7–8.

Sureshbabu, G., Jagadeesh, S. D. V. S., Saicharan, U. B., & Praneeth, P. R. S. (2013). Analysis of a Single Cylinder Combustion Engine Using CFD, (5), 164–167.