FLOW ANALYSIS OF PISTON HEAD GEOMETRY FOR DIRECT INJECTION SPARK IGNITION ENGINE

Abdul Rahim Shar Anuar, Mohd Farid Muhamad Said*, Nur Adila Mohamad Shafie, Azhar Abdul Aziz, Henry Nasution

Automotive Development Centre (ADC), Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, 81310 UTM Johor Bahru, Johor, Malaysia

Graphical abstract

Particle Traces Colored by temperature (k). (Time=6.3333e-03)   Jun 06, 2015
Crank Angle=720.00(deg)   ANSYS Fluent 15.0 (3d, dp, pbns, dynamesh, spe, ske, transient)
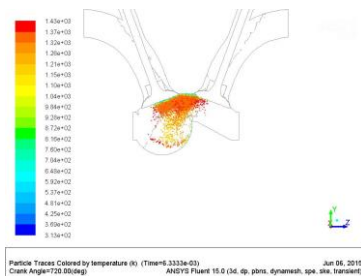
Abstract

Constructors of gasoline engines face higher and higher requirements as regards to ecological issues, and increase in engine efficiency at simultaneous decrease in fuel consumption. Satisfying these requirements is possible by the recognition of the phenomena occurred inside engine cylinder, the choice of suitable optimal parameters of fuel injection process, and the determination of geometrical shapes of the combustion chamber and piston head. The aim of this study is to simulate flow in Fuel Direct-Injection engine with different geometrical shapes of piston head. Designing piston head shapes was done by referring to existing motorcycle, Demak 200cc-single cylinder using SolidWork and ANSYS software. The parameter investigated are shallow and deep bowl design of piston head. In term of fuel distribution throughout the combustion chamber, engine model that has deeper bowl (Model 2) shows better fuel distribution than model of shallow bowl as it manages to direct the fuel injected towards the location of spark plug. Total kinetic energy of Model 2 is about 20% higher than Model 1. Therefore, engine with deeper bowl is chose as the best model between the two models as it can create a richer mixture around the spark plug.

Keywords: Direct-Injection, CFD simulation, piston head geometry, stratified combustion.

Abstrak

Pereka enjin gasolin berdepan dengan permintaan tinggi yang bersandarkan kepada isu-isu ekologi, kecekapan enjin yang tinggi serta pengunaan bahanapi yang rendah. Bagi memenuhi keperluan ini, adalah perlu untuk mengenal pasti fenomena yang terjadi di dalam silinder enjin, pilihan parameter optimal yang sesuai oleh proses pancitan bahan api, dan penentuan bentuk geometri kebuk pembakaran serta puncak omboh. Tujuan kajian adalah untuk mensimulasi aliran dalam enjin Pancitan Terus Bahan Api dengan bentuk geometri puncak omboh yang berbeza. Merekabentuk bentuk puncak omboh dilakukan dengan merujuk kepada motosikal sedia ada, bersilinder tunggal 200cc, Demak, dengan menggunakan SolidWork dan perisian ANSYS. Parameter yang dikaji ialah bentuk mangkuk yang dangkal dan dalam pada puncak omboh. Bagi bentuk aliran bahan api diseluruh kebuk pembakaran, model enjin dengan mangkuk lebih dalam (Model 2) menunjukkan pembahagian bahan api adalah lebih baik berbanding model dengan mangkuk yang dangkal, kerana ia berupaya menghalakan bahan api yang dipancit kearah lokasi palam pencucuh. Tenaga keseluruhan kinetik bagi Model 2 adalah 20% lebih tinggi berbanding Model 1. Maka, enjin dengan mangkuk lebih dalam dipilih sebagai model terbaik antara dua model tersebut kerana ia mampu memberi campuran lebih kaya di sekitar palam pencucuh.

Kata kunci: Pancitan-terus, simulasi CFD, geometri puncak omboh, pembakaran berstrata.

1.0 INTRODUCTION

Gasoline Direct Injection (GDI) engine is proven to be more advantageous compared to any other conventional Spark Ignition (SI) engines. Two combustion modes that are homogeneous combustion mode and stratified combustion mode introduced in the GDI engines help to improve the accuracy of Air-Fuel (AF) ratio during dynamics operation, and decrease the fuel consumption and CO_2 emission [1-4]. With the injector installed inside the combustion chamber and its non-throttle operation, engine can achieved higher power output.

Aside from the in-cylinder direct injection, GDI engine has another special feature that is the unique piston top surface shape. The piston top surface shape plays an important role of determining the behavior of air-fuel mixture inside the combustion chamber. Commonly, the piston surface of GDI engine is equipped with a piston bowl, and the bowl design is mostly determined by its bowl radius, bowl depth, bowl width, and bowl location relative to the spark plug [5-6].

The piston top surface is designed in such way that it will be compatible for both homogeneous combustion mode and also the stratified combustion mode. For the homogeneous combustion mode, the piston top surface plays the role to create a homogeneous mixture of the fuel and air before the combustion. Whereas, for stratification combustion mode, the piston top surface has the responsibility to form a stratified-charge rich fuel cloud around the spark plug [7-8].

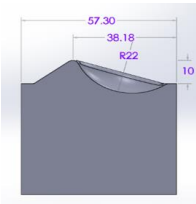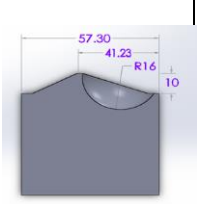## 2.0 METHODOLOGY

2.1 Building Models

Figure 1 Piston design model

For this study, a total of two models were built with each of the models having different types of parameters. The variation of parameters in this project mainly focused on the bowl radius and the bowl position on the piston top (Figure 1). The models were built based on common GDI engine piston design by using SolidWork software. In term of the piston design measurement, it is adjusted so that the piston is compatible with the Demak engine with respect to the piston bore and stroke.

2.2 Creating Flow Volume

After combining the piston with the cylinder head, a cavity exist inside the combustion chamber. For the simulation purpose, the cavity inside the combustion chamber was extracted to obtain the flow volume (Figure 2). The flow volume serves as the flow path of the mixture inside the combustion chamber. The flow volume can be created by using the Combine feature in SolidWork software.
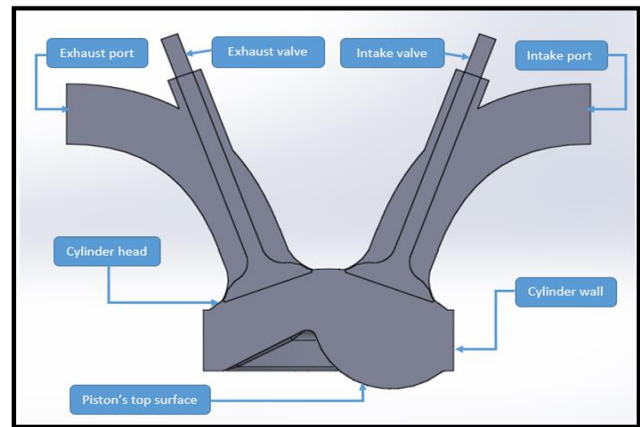

Figure 2 Flow volume and the required parts for the flow volume

2.3 Setting Up IC Engine Properties

The engine properties are defined in the ICE properties after IC engine analysis has been selected from the analysis systems toolbox. In the ICE properties, input data such as the simulation type is included. There are three types of simulation provided for the ICE engine simulation which are Cold Flow simulation, Port Flow simulation and Combustion simulation. Out of the three simulation types, the Combustion simulation is chose as the simulation type of this project since spray injection has to be included in the study. Basic engine properties such as the engine connecting rod length, crank radius, engine speed, minimum valves lift and the valves lift profile are also defined as shown in Table 1.

Table 1 Basic engine properties

No.	Property	Value
1	Simulation type	Combustion simulation
2	Combustion simulation type	Full Engine Full Cycle
3	Connecting rod length	90 mm
4	Crank radius	30 mm
5	Engine speed	2000 rpm
6	Minimum valves lift	0.2 mm

2.4 Decomposing Geometry

The flow volume generated is then imported to the ANSYS Fluent software. The first step before the simulation process is carried out is to decompose the computational geometry. When a model is decomposed, the model imported will be divided into smaller volumes where these volumes are compulsory as the mesh requirement in the meshing process [9, 10]. For that purpose, the geometry of the model has to be designed in such way that each small volume can be generated during the geometry decomposition.

Before decomposing the model, some parts of the model need to be defined first such as the inlet, outlet, intake valve, exhaust valve, intake valve seat and exhaust valve seat. Since the scope of the study is focused on the fuel injection during the compression stroke, the model is decomposed at 644 cad just before the fuel is injected into the combustion chamber during the compression stroke.

2.5 Meshing & Grid independence study

Once the model has been decomposed, meshing process is done. Mesh is generated individually based on the small volumes of the computational geometry created when the model is decomposed [9,11]. For the analysis, dynamic mesh is conducted.

Grid independence test was done in order to find the minimum number of mesh cell that can give good result from the simulation. It is important to determine the right total number of mesh cell to ensure that it is neither too low until causing high deviation from the right result, nor too high that can cause long computational time. The grid independence test computed for original piston has been performed at different number of mesh cell ranging from 580,000 to 900,000.

2.6 ICE Solver setting

ICE solver setting is divided into several parts where in each part, some settings are required for the model.

Basically, the ICE solver setting is configured to set the relevant input data required for the simulation. Some example of the input data (Figure 3) are the engine type, fuel type, fuel injection, boundary conditions of the analysis and the type of result required at the end of the simulation (Table 2 & 3).
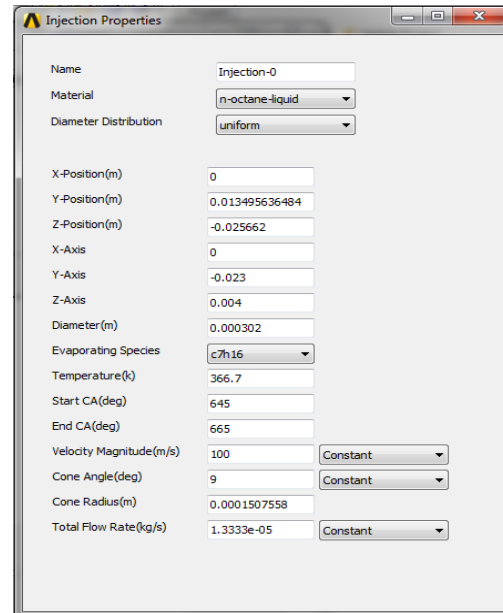
Figure 3 Data input for injection setting [12]

Table 2 Temperature of the combustion chamber wall [8]

Part	Zone	Boundary condition
Head	cyl-head, invalve1–ch, and exvalve1–ch	485 K
Piston	piston	485 K
Liner	cyl-tri	500 K
Exhaust valve	exvalve1–ib, exvalve1–ob, and exvalve1–stem	777 K
Exhaust port	exvalve1–port and exvalve1–seat	485 K
Intake valve	invalve1–ib, invalve1–ob, and invalve1–stem	400 K
Intake port	invalve1–port and invalve1–seat,	313 K

Table 3 Temperature and pressure of the mixture inside combustion chamber [8]

Part	Zone	Pressure	Temperature
Exhaust port	fluid-exvalve-1–port, fluid-exvalve-1–vlayer, fluid-exvalve-1–ib	0.5 MPa	1070 K
Inlet port	fluid-invalve-1–port, fluid-invalve-1–vlayer, fluid-invalve-1–ib	0 Pa	313 K
Chamber	fluid-ch	1 MPa	1070 K

2.7 Running the Simulation

ANSYS Fluent set the relevant number of time-steps and iterations to be computed for the simulation process to complete. For every iteration, 30 time-steps are calculated. As an optional choice, to decrease the amount of time consumed in the simulation, the continuity of the calculation is increased to 0.1 and the number of time-step is increased to 1. The total iteration required for the simulation to complete is 3280 iterations where each of the iteration takes a maximum of 50 time-steps. Once the setting is done, the last step is to compute the simulation. The simulation process can take days to complete depends on the number of iterations provided.

3.0 RESULTS AND DISCUSSION

3.1 Swirl Ratio

Swirl is defined as the rotational movement of air around the cylinder vertical axis. As one of the parameter used to quantify the in-cylinder fluid motion, swirl influence the heat transfer, combustion quality and emission in addition to affecting the mixing of air-fuel and combustion process [7]. Together with tumble, great intensity of these two parameters in induced flow during intake stroke will result in high turbulence in engine which can be retained during compression stroke [13, 14]. In reality, the nature of swirl phenomenon inside an engine is very difficult to be determined, yet to be predicted. Previously mentioned Flow Bench test is one of the methods frequently used to investigate the swirl in engine at steady state. In the measurement of swirl inside operating engine, swirl ratio is used to quantify swirl. Swirl ratio is defined as:

$$R_s = {\omega_s}/{2\pi N}$$

where R_s is swirl ratio, ω_s is angular velocity of rotating flow at swirl axis, and N is engine operating speed [12]. In CFD simulation using ANSYS IC Engine, swirl ratio can be automatically generated by simulation by inserting

the right command for swirl ratio. ANSYS IC Engine deduced the swirl ratio as:

$$R_s = \frac{L._{sa}}{I._{sa}}\Big/\frac{2\pi N}{60}$$

where $L._{sa}$ is magnitude of fluid angular momentum with respect to swirl axis, $I._{sa}$ is moment of inertia of fluid mass about swirl axis, and N is engine operating speed (revolution per minute) [12].

In this study, extensive simulation works have been carried out. As depicted in Figure 4, Model 1 shows about 5% higher swirl intensity compared to Model 2. This is because Model 2 has higher surface area compared to Model 1 due to its larger piston bowl radius and also the depth of the piston bowl which is deeper than Model 2. This high surface area creates high friction to the mixture flow when it comes in contact with the cylinder wall especially at the piston bowl region which in turn resisting the swirl motion inside the combustion chamber of Model 2.
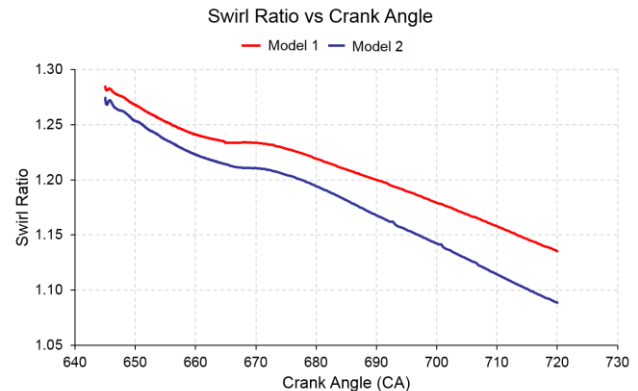
Figure 4 Swirl ratio generated in the combustion chamber of model 1 and model 2

3.2 Tumble Ratio

When piston approaches top-dead-centre (TDC) at the end of compression stroke, mixture inside engine undergoes radially inward or transverse motion called squish. Tumble is the secondary rotational flow as a result of squish motion when piston located nears TDC. Tumble is also defined as rotational flow occurred about circumferential axis near the piston bowl outer edge [2, 8,15]. By means of experimental methods, tumble ratio is usually measured using steady flow rig at selected valve lift, but tumble ratio value are deemed specific on tumble rig design. Thus, different data of tumble ratio from experiment with different rigs cannot be compared directly. In order to quantify the tumble in internal combustion engine, tumble ratio is the parameter discussed in this study.

In ANSYS IC Engine, tumble ratio is automatically computed under right simulation command. CFD

simulation by ANSYS IC Engine computed tumble ratio as:

$$R_t = \frac{L \cdot ta}{I \cdot ta} \Big/ \frac{2\pi N}{60}$$

where $L_{\cdot ta}$ is magnitude of fluid angular momentum with respect to tumble axis, and $I_{\cdot ta}$ is moment of inertia of fluid mass about tumble axis [7]. In addition to tumble ratio, ANSYS IC Engine introduce another parameter which is the cross tumble ratio which involve the computation of rotational flow at the axis perpendicular to tumble axis which also known as cross tumble axis.

Based on simulated results depicted in Figure 5, the tumble ratio generated in combustion chamber of all the models starts with a negative value and continuously decreasing. However, the negative value does not indicate that the tumble intensity is decreasing, but it indicates the direction of the tumble motion of the mixture which is directed to the exhaust side of the combustion chamber. The figure shows that the intensity of the tumble motion is increasing in a certain direction along the increasing of crank angle degree.

The magnitude tumble intensity of Model 1 is about 15% higher than Model 2 (Figure 5). The major factor that contributes to the turbulent intensity is the piston bowl design where with the right design, the piston bowl can help to promote the tumble motion. The right piston bowl design can also help to determine the direction of the mixture throughout the combustion chamber.
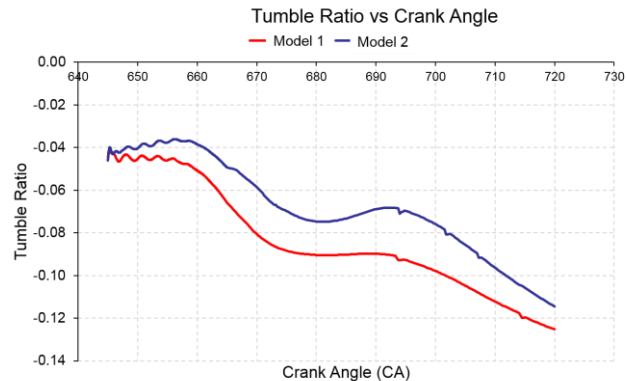
Figure 5 Tumble ratio generated inside the combustion chamber of Model 1 and Model 2

3.3   Total Kinetic Energy (TKE)

The TKE of Model 2 is 20% higher compared to Model 1 (Figure 6). Due to deeper bowl depth and curvier piston bowl, the piston bowl of Model 2 tends to create vortices with much higher speed than speed of vortices created by piston bowl of Model 1 during the compression stroke. These high speed vortices, in time, are colliding against each other to create high turbulent intensity inside the combustion chamber.
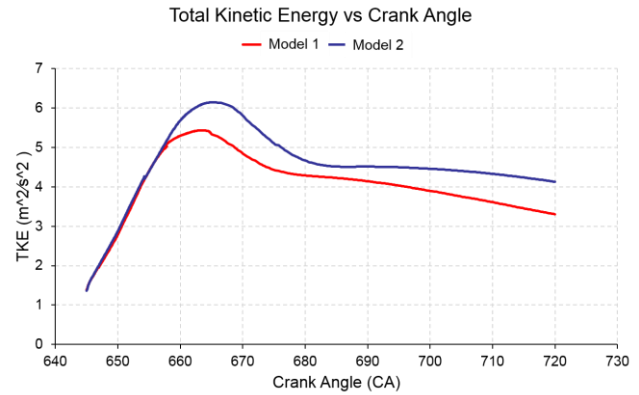
Figure 6 Graph of mass-average turbulent kinetic energy (TKE) of the two models vs the crank angle

3.4   Pressure

The pressure generated inside the combustion chamber of Model 1 is slightly higher than the pressure in Model 2 (Figure 7). This slight difference is mainly due to the difference of compression ratio for both model, which is caused by different parameters of the piston models that have different bowl radius and bowl position. Model 1 has a compression ratio of 9:1 whereby the compression ratio for model 2 is 8.5:1. The bowl radius is affecting the clearance volume, V_c of both models and thus the compression ratio.
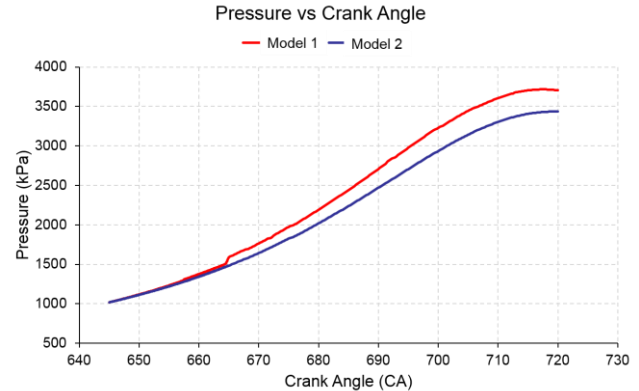
Figure 7 Generated pressure in combustion chamber of Model 1 and Model 2

3.5   Fuel Particle Traces

Model 1 fails to create rich mixture around the spark plug which is the most important requirement for stratified combustion (Figure 8). Furthermore, this piston design will end up increasing the fuel consumption and produces much lower power output due to unevenly fuel distribution.

For Model 2, when fuel is injected, it travels along the curve part of the piston bowl and in the end it is directed to the center of the combustion chamber where the spark plug is located (Figure 9). This behavior is preferable for stratified combustion since the fuel injected will form a rich mixture around the spark plug. However, there is also a portion of the fuel is being directed back to the intake side. Because of this, the consumption of fuel during combustion cannot be fully optimized.
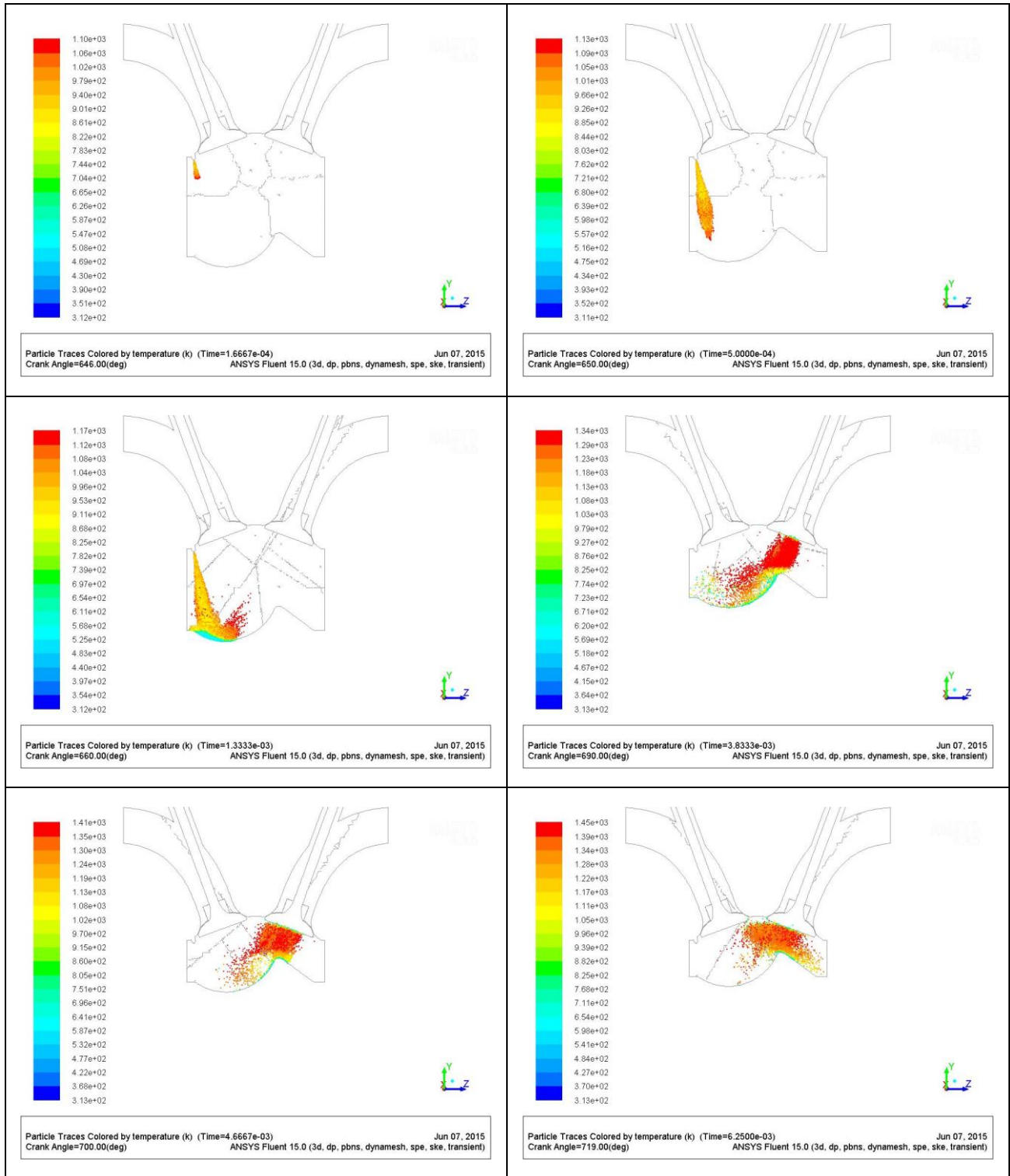
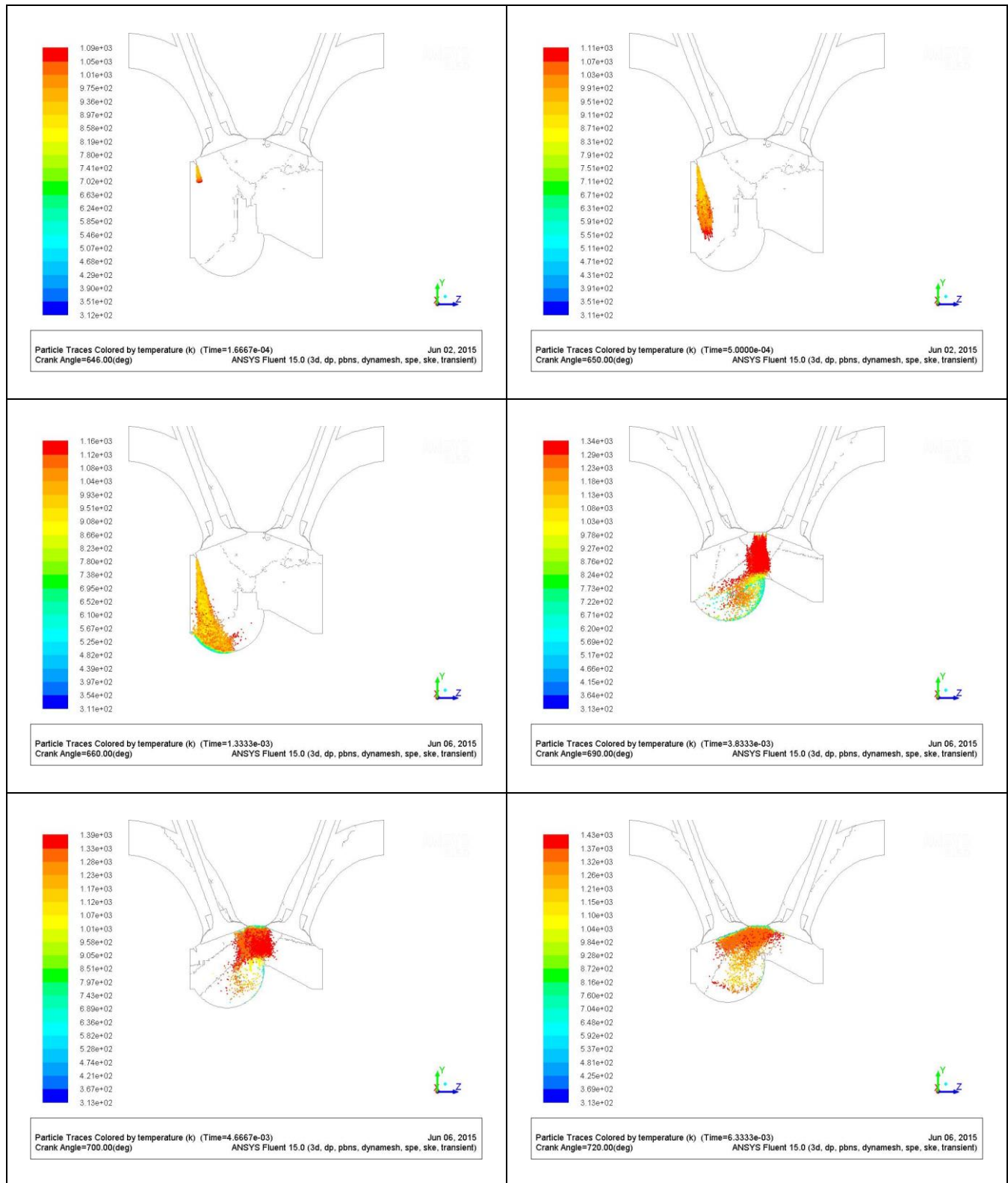**Figure 8** Fuel particle traces of Model 1

Figure 9 Fuel particle traces of Model 2

4.0 CONCLUSION

Based on the results obtained from the simulation, both models have their own advantages and disadvantages. For Model 1, the result indicates that it give higher swirl and tumble intensity compare to Model 2. Whereas Model 2 has higher TKE value than Model 1. However, the result of swirl ratio, tumble ratio

and TKE does not really show a significant difference between the two models. For the fuel distribution throughout the combustion chamber, Model 2 is better than Model 1 since the piston bowl of Model 2 directs the fuel axially towards the center of the combustion chamber where the spark plug is located. This behavior of Model 2 design that have deeper bowl is much preferable for stratified combustion mode.

Acknowledgement

References

[1]   Zhao, H. 2010. *Advanced Direct Injection Combustion Engine Technologies and Development*. Cambridge: Woodhead Publishing Limited.

[2]   Heywood, J.B. 1988. *Internal Combustion Engine fundamentals*. United States: McGraw-Hill, Inc.

[3]   Paul, B. and Ganesan, V. 2010. Flow Field Development in a Direct Injection Diesel Engine with Different Manifolds. *International Journal of Engineering, Science and Technology*. 2(1): 80-91.

[4]   Oh, H. and Bae, C. 2013. Effects of the Injection Timing on Spray and Combustion Characteristics in a Spray-guided DISI Engine under Lean-stratified Operation. *Fuel*. 107: 225 – 235.

[5]   Xu, Z., Yi, J., Curtis, E. and Wooldridge, S. 2009. Applications of CFD Modeling in GDI Engine Piston Optimization. *SAE Technical Paper*. 2009-01-1936.

[6]   Pathak, Y.R., Deore, K.D. and Maharu, P.V. 2014. In Cylinder Cold Flow CFD Simulation of IC Engine using Hybrid Approach. *International Journal of Research in Engineering and Technology*. 3(8):16-21.

[7]   Priscilla and Meena, P. 2013. A Comprehensive Study on In-cylinder IC Engine due to Swirl Flow. *International Journal of Engineering Research & Technology*. 2(7):1156-1161.

[8]   Abianch, O.S., Mirsalim, M. and Sabet, A.S. 2009. Investigation of Swirling and Tumbling Flow Pattern of Spark Ignition Engine. *The Journal of Engine Research*. 14: 27-34.

[9]   Lakshman, A., Karthikeyan, C.P. and Padmanabhan, R. 2013. 3D In-Cylinder Cold Flow Simulation Studies in an IC Engine using CFD. *International Journal of Research in Mechanical Engineering*. 1(1): 64-69.

[10]   Hepkaya, E., Karaaslan, S., Uslu, S, Dinler, N. and Yucel, N. 2014. A Case Study of Combustion Modelling in a Spark Ignition Engine using Coherent Flame Model. *Journal of Thermal Science and Technology*. 34(2): 111-121.

[11]   Czyz, A. and Pletrykowski, K. 2014. CFD Model of the CNG Direct Injection Engine. *Advances in Science and Technology Research Journal*. 8(23): 45-52.

[12]   ANSYS Fluent. 2015. *Internal Combustion Engine Tutorial Guide*. ANSYS.

[13]   Barbouchi, Z. and Bessrour, J. 2009. Turbulence Study in the Internal Combustion Engine. *Journal of Engineering and Technology Research*. 1(9): 194-202.

[14]   Pfeffer, T., Bühler, P., Meier, E.D., and Hamdani, Z. 2002. Influence of Intake Tumble Ratio on General Combustion Performance, Flame Speed and Propagation at a Formula One Type High-speed Research Engine. *SAE 2002 World Congress*. Detroit, Michigan.

[15]   Han, S.B., Chung, Y.J. and Lee, S. 1995. Effect of Engine Variables on the Turbulent Flow of a Spark Ignition Engine. *Journal of Mechanical Science and Technology*. 9(4): 492-501.