ISSN 1996-3343

Asian Journal of Applied Sciences

http://knowledgiascientific.com

Knowledgia SCIENTIFIC
A Place to Publish Outstanding Research

OPEN ACCESS
Asian Journal of Applied Sciences
ISSN 1996-3343
DOI: 10.3923/ajaps.2017.102.115

Research Article
Computational Analysis of Flow Characteristic in Inlet and Exhaust Manifolds of Single Cylinder Spark Ignition Engine

[1]Hussain H. Al-Kayiem, [2]Hasanain A. Abdul Wahhab and [2]A. Rashid A. Aziz

[1]Department of Mechanical Engineering, Universiti Teknologi PETRONAS, 32610 Seri Iskandar, Perak, Malaysia
[2]Centre for Automotive Research and Electric Mobility, Universiti Teknologi PETRONAS, 32610 Seri Iskandar, Perak, Malaysia

Abstract
Background and Objective: The gas hydrothermal behaviour is strongly influencing the engine performance. The flow behaviour may be modified through the unavoidable installation of a flow restrictions and modifiers into the intake and exhaust systems and consequently the engine performance is adversely altered. The objective of this study was to perform a numerical modeling and analysis to predict the flow characteristics in the inlet and exhaust manifolds of a four-stroke single cylinder gasoline internal combustion engine. **Materials and Methods:** The flow at the intake and exhaust was simulated using ANSYS FLUENT employing boundary conditions at valves timing at critical points of opening and closing. A user define function has been coded to relate the valves and piston motions to generate the moving mesh inside the cylinder. **Results:** The results demonstrated some reversal flow cases, at certain positions of the valves. At crank angle around 570° the flow was reversed from the cylinder to the manifold, while at crank angle of around 370°, the flow was reversed from the manifold to the cylinder. **Conclusion:** The results obtained from the simulation code be used as guide to improve the understanding of the hydrothermal behaviour of the fluids in the manifolds and might be utilized to improve the manifold design.

Key words: Engine performance, flow in manifolds, gas hydrothermal, computational fluid dynamic, SI engine

Received: December 30, 2016 Accepted: April 13, 2017 Published: June 15, 2017

Citation: Hussain H. Al-Kayiem, Hasanain A. Abdul Wahhab and A. Rashid A. Aziz, 2017. Computational analysis of flow characteristic in inlet and exhaust manifolds of single cylinder spark ignition engine. Asian J. Applied Sci., 10: 102-115.

Corresponding Author: A. Rashid A. Aziz, Centre for Automotive Research and Electric Mobility, Universiti Teknologi PETRONAS, 32610 Seri Iskandar, Perak, Malaysia Tel: 006-01136377102

Competing Interest: The authors have declared that no competing interest exists.

Data Availability: All relevant data are within the paper and its supporting information files.

INTRODUCTION

The intake and compression strokes are one of the most significant processes which influences the behaviour of air flow structure inside cylinder. The best design of intake manifolds produce a good conditions needed for the fuel injection during the compression stroke. Due to the high velocity inside the Internal Combustion Engine (ICE) during intake operation, all in cylinder flows are typically turbulent[1]. More so, exhaust manifolds are a part of IC engines, it is used to collect and carry the exhaust gases away from the cylinder head and send it to the exhaust tail, with a minimum back pressure. The exhaust manifold plays an important role in the performance of an engine system. Generally, a proper intake and exhaust manifolds design leads to reduce the fuel consumption and exhaust emissions[2,3]. In today's world, major objectives of engine designers are to achieve the twin goals of best performance and lowest possible emission levels, not only to have more engine power, but also to be more environmentally friendly[4].

Heat transfer is an important phenomenon in both the intake and exhaust systems of an internal combustion engine[3]. On the intake side, it increases the intake air temperature which reduces the volumetric efficiency and the tolerance to engine knock and causes higher chemical reaction rates leading to increased NOx emissions[3]. It also improves engine performance and emissions through enhancing fuel evaporation and charge mixing in the engine intake ports and cylinders. On the exhaust side, heat transfer robs the flow of available energy. With stricter emission regulations, the ability to get the after treatment system to the proper temperature during cold starts is crucial. Losing energy will cause this system to take longer to reach its maximum effectiveness and result in more tailpipe emissions[5]. Exhaust gas should be kept at high temperature in the exhaust pipe, especially at low engine rotation conditions when engine starts, because, the catalyst located at the end of the exhaust pipe will absorb more pollutant under high temperature conditions[2,4]. Exhaust gas should also be led from the piston chambers to the exhaust manifold smoothly, to maximize the engine power especially at high engine rotation conditions. Such design is used to be performed by trial and error through many experiments and analysis by engineers[4]. Therefore, an automated design optimization is desired to reduce technical schedule and cost risks for new engine developments. The efficiency of an SI engine depends on several complicated processes including induction, mixture preparation, combustion and exhaust flow[6].

Air motion within the engine cylinder is one of the major factors that control the fuel/air mixing and combustion process in spark ignition engines. It also has a significant impact on heat transfer. Both the bulk air motion and the turbulence features of the flow are important to produce the homogeneity structure of air flow come into cylinder[7,8]. Intake generated swirl usually persists through the compression, combustion and expansion stroke and it can greatly enhances the mixing of air and fuel to give a homogeneous mixture in the very short time[8]. It is also a main mechanism for very rapid spreading of the flame front during the combustion process[9-11]. Many researchers worked in this area to investigate experimentally as well as computationally to reveal on the flow phenomenon in the cylinder of internal combustion engine[12-14]. Cheeda *et al.*[12] worked on design and CFD analysis of a regenerator for a turbo-shaft helicopter engine. While, Hasse *et al.*[14] worked on numerical investigation of cyclic variations in SI engines using a hybrid URANS/LES modeling approach. The 3D CFD analysis of an abnormally rapid combustion phenomenon in downsized gasoline engines is studied by Reveille and Duparchy[15] and Kurniawan *et al.*[16], who focused on a particular abnormally rapid, yet non-destructive and seemingly stable combustion phenomena which have been recognized on low speed mid to high load operating points when performing aggressive downsizings on various engines. Also, many studies have been carried out on estimation of flow characteristic in exhaust manifolds of internal combustion engines using a four-stroke variable compression ratio single cylinder gasoline engine[3,8]. Rathnaraj[17] worked on the predictions obtainable using the conformal and indirect interfaces for Conjugate Heat Transfer (CHT) analysis in an exhaust manifold. Satish *et al.*[18] approved thermo mechanical fatigue analysis of stainless steel exhaust manifolds.

In this study the researchers focused on the effects of analysis for thermodynamics parameters: Temperature, pressure and velocity on various parts of the engine, the effect of changing temperature on the intake swirl generation, the analysis of the flue gases of various fuels in exhaust manifold. It is clear that few works have carried out done in field of determining the behaviour of these parameters in the inlet and exhaust manifolds and along the length of the engine cylinder.

The objective of this study was to present a developed numerical procedure and CFD simulation of the flow characteristic in the inlet and exhaust manifolds of internal combustion engines.

MATERIALS AND METHODS

Spark ignition engine: This study was performed using experimental data of a four stroke single cylinder SI engine[8] to build the computational model is shown in Fig. 1. Single cylinder four strokes type was (GR0306/000/037A), the detailed specification of the base engine selected for the simulation is given in Table 1.

Computational procedure: The methodology adopted for the present study was as follows. Flow through the inlet and exhaust manifolds were simulated to study the in-cylinder flow field, which included the following steps:

- Solid modeling of the inlet and exhaust manifolds and cylinder geometry with valves. Figure 2 show the engine's combustion chamber, air intake and exhaust valves
- Mesh generation
- Solution of the governing equations with appropriate boundary conditions
- Comparison of the simulated results with the available results in the experimental tests[8]

This study was expected to explore the potential of using CFD tool for analysis of flow characteristic in the inlet and exhaust manifolds. The CFD package included user interfaces to input problem parameters and to examine the results. The three-dimensional model of the single cylinder for SI engine with intake and exhaust systems was developed by using the pre-processor CFD software. The number of cells used in this model of one case (crank angle 540°) under study was 778,567. Mesh refinement investigation was carried out to optimize the number of cells used. It was found that increasing the number of cells to 815,222 and 868,555 had no effect on the results accuracy. Hence, 778,567 was selected to be the optimum number of cells that can be used in the simulation. The SI engine cylinder with valves and inlet and exhaust manifolds and the mesh for cases under study is shown in Fig. 3.

Governing equation: The CFD code contains the numerical solutions of the fundamental governing equations of fluid dynamics namely the continuity, momentum, energy, species and turbulent equations. The FLUENT software package was used to accomplish this job. This software flow solver is a finite volume, pressure based, fully implicit code solving the 3D Navier-Stokes equations governing fluid flow and associated physics considering incompressible fluids.

Table 1: Single cylinder engine's parameters[8]

Model	Single cylinder
Bore	90 mm
Stroke	85 mm
Connecting rod length (L)	160 mm
Crank radius (r)	47.5 mm
Displacement (Vd)	541 cm^3
Compression ratio	9
Fixed spark advance	10° spark ignition
Intake valve opens (BTDC)	54°
Intake valve close (ATDC)	22°
Exhaust valve opens (BTDC)	22°
Exhaust valve close (ATDC)	54°

BTDC: Before top dead center, ATDC: After top dead center

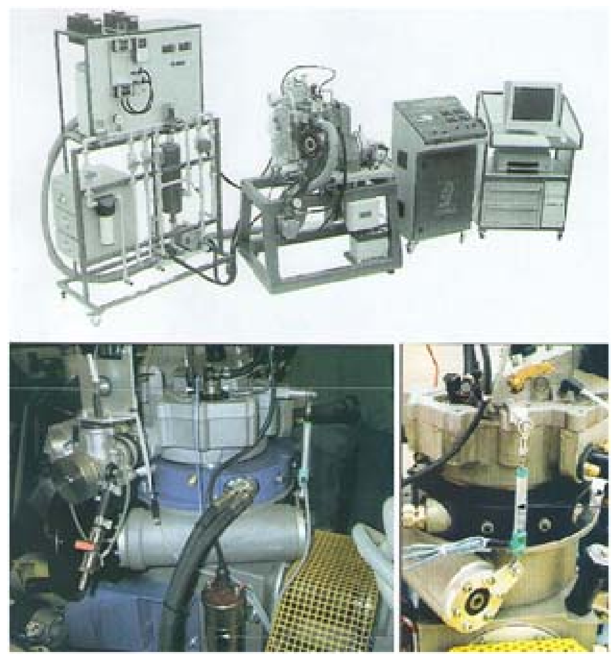

Fig. 1: A single cylinder four strokes type (GR0306/000/037A)[8]

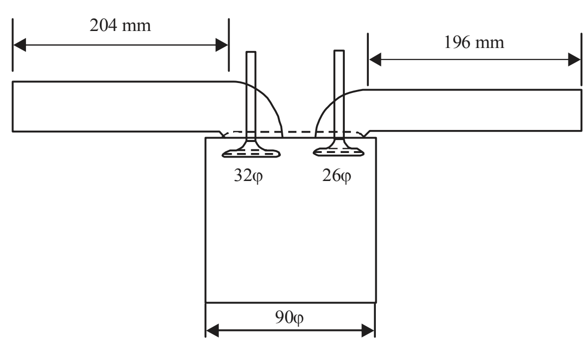

Fig. 2: Engine's combustion chamber, air intake and exhaust valves

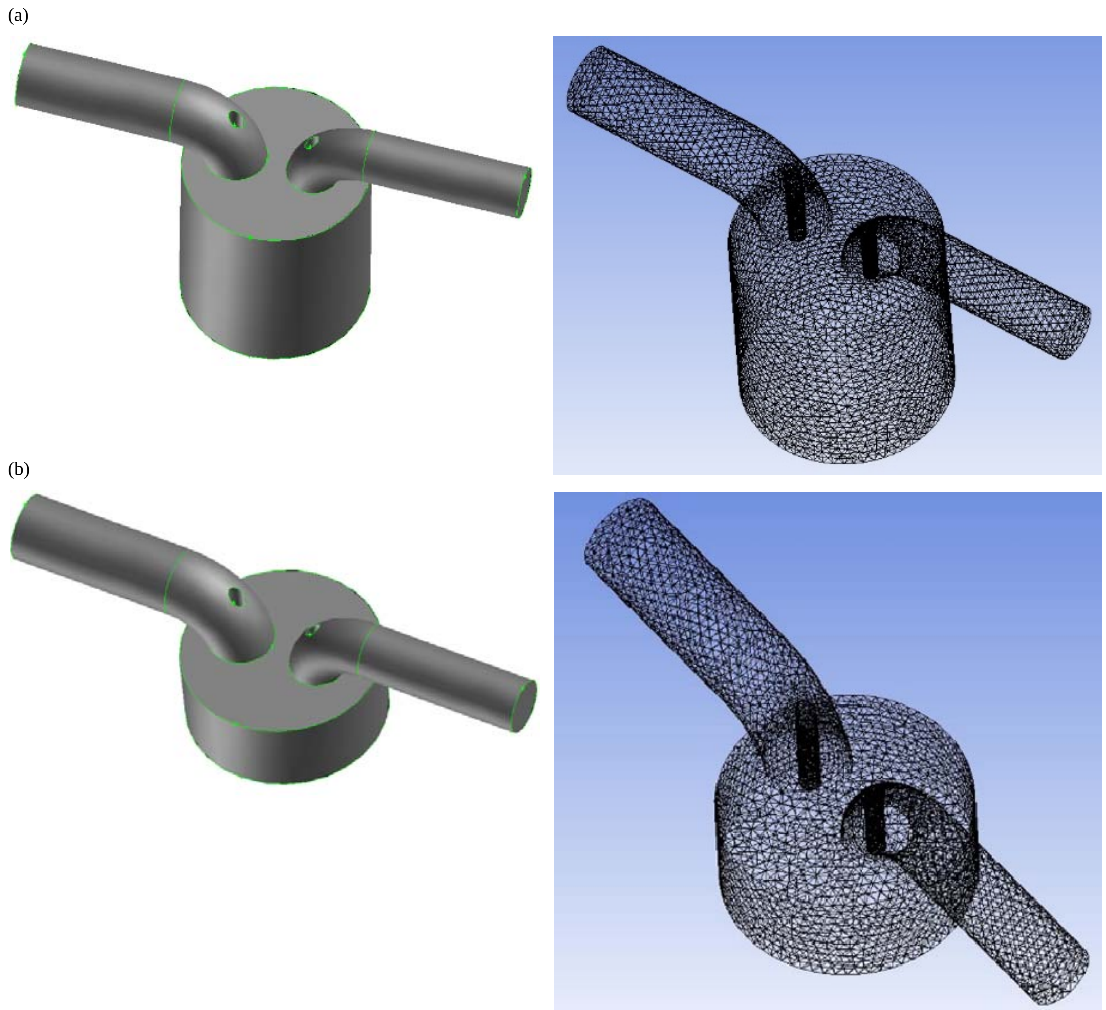

Fig. 3(a-b): SI engine cylinder with inlet and exhaust manifolds and the mesh for cases under study with crank angle (a) 540° and (b) 404°

Continuity equation: It is based on the principle of conservation of mass. Net mass flow out of control volume equal to time rate of decrease of mass inside control volume:

$$\frac{\partial \rho}{\partial t} + \nabla(\rho u) = S_m \quad (1)$$

Momentum equation: It is based on the law of conservation of momentum, which states that the net force acting in a fluid mass is equal to change in momentum of flow per unit time in that direction:

$$\frac{\partial(\rho \bar{u})}{\partial t} + \nabla.\left(\rho \overline{\overline{uu}}\right) = -\nabla P + \nabla.\sigma + \rho g + S_{mom} \quad (2)$$

Energy equation: It is based on the principle that total energy is conserved. Total energy entering control volume equal to total energy leaving control volume:

$$\frac{\partial(\rho E)}{\partial t} + \nabla.\left(\rho \bar{u} E\right) = P\nabla.\bar{u} + \nabla.k\nabla T + \mu\varnothing + S_E \quad (3)$$

where, ∂ is partial derivative, ρ is mass density, u is velocity, S is source, t is time, φ is viscous dissipation, g is gravitational acceleration, T is temperature, E is specific internal energy, μ is viscosity, k is thermal conductivity and mom is momentum.

Boundary conditions: Pressure and temperature were used as boundary condition at both the inlet and the exhaust

Table 2: Properties initial boundary condition of pressure to cases under study

Crank angle (°)	Inlet flow pressure (kPa)	Exhaust flow pressure (kPa)	Pressure inside cylinder (kPa)
414	105	-	82
540	105	-	97.7
187	-	101	206.8
306	-	101	124.2
316	105	101	110.8
404	105	101	86.7

Table 3: Initial boundary conditions of engine temperature

Temperature (K)	Value
Inlet valve temperature	523
Exhaust valve temperature	923
Cylinder wall temperature	458
Piston face temperature	573
Spark plug temperature	873

manifolds. Attached boundaries were specified on the coincident cell face near the cells above/below the valve. No slip wall boundary condition in conjunction with logarithmic law of wall was used. Walls were considered to be same experimental values as discussed by Pulkrabek[19]. Table 2 gives the initial boundary conditions of pressure for the cases under study.

Typical temperature values found in an SI engine operating at normal steady state conditions were used by Pulkrabek[19]. The initial boundary conditions of the engine temperatures are shown in Table 3.

Statistical analysis: The numeric procedure included design function code has been developed to solve the moving boundary problem during the up/down motion of the piston. The ANSYS FLUENT version 14 software has been utilized to simulate the flow field in the intake and exhaust systems for the engine dimensions and timings used in the experimental study were similar with Al-Khishali *et al.*[8].

RESULTS AND DISCUSSION

Many cases have been studied using fluent program where the crank angle are the main parameter that affects the flow in pipes and inside the cylinder. Pressure, temperature and flow pattern are the outputs of the programs. Results of certain cases were presented at full intake valve open case, partially inlet valve open case, full open of exhaust gas valve, partially exhaust gas valve open. Overlapping results, where one valve is fully open the other is partially open for inlet stroke and exhaust stroke have been presented as well.

Flow characteristic in inlet manifold: The pressure, temperature and velocity variations in the cylinder with respect to crank angle during gas exchange cycle for an engine speed (2500 rpm) are shown in Fig. 4a-c. It is observed that most graphs exhibit almost similar characteristics between numerical and experimental magnitudes at engine speed (2500 rpm). This clearly illustrates the unsteady nature of the flow in the inlet pipes of the engine. From the pressures obtained from fluent, it is shown that it has similar values to those from the experimental data[8,20]. The former has a better representation for both temperature and velocity which will give the designer a better tool to improve the performance by altering the dimensions and type of pipes to be used in the

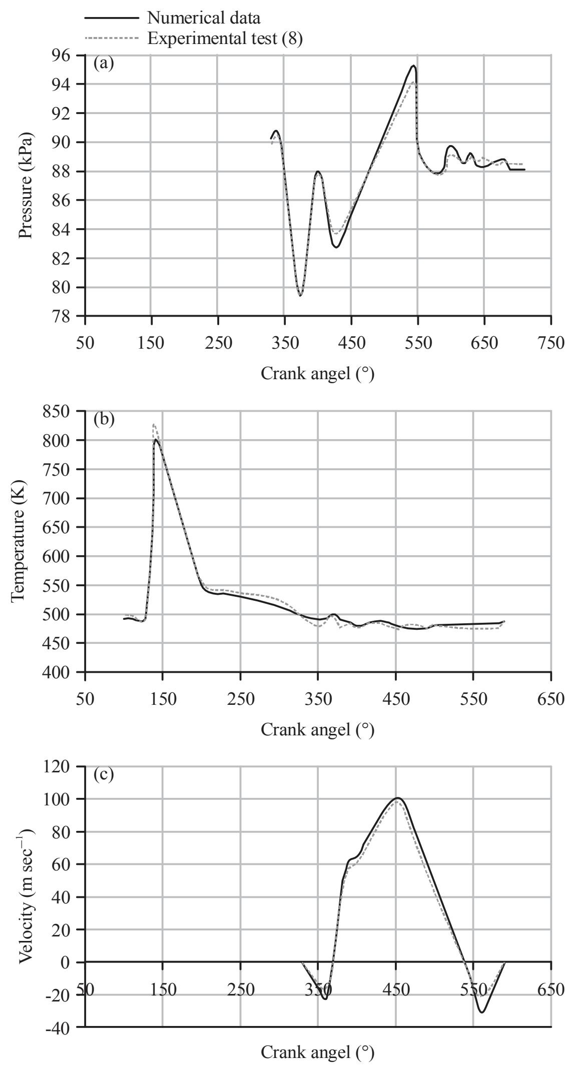

Fig. 4(a-c): (a) Pressure, (b) Temperature and (c) Velocity distribution in the cylinder with respect to crank angle during gas exchange in the inlet manifold

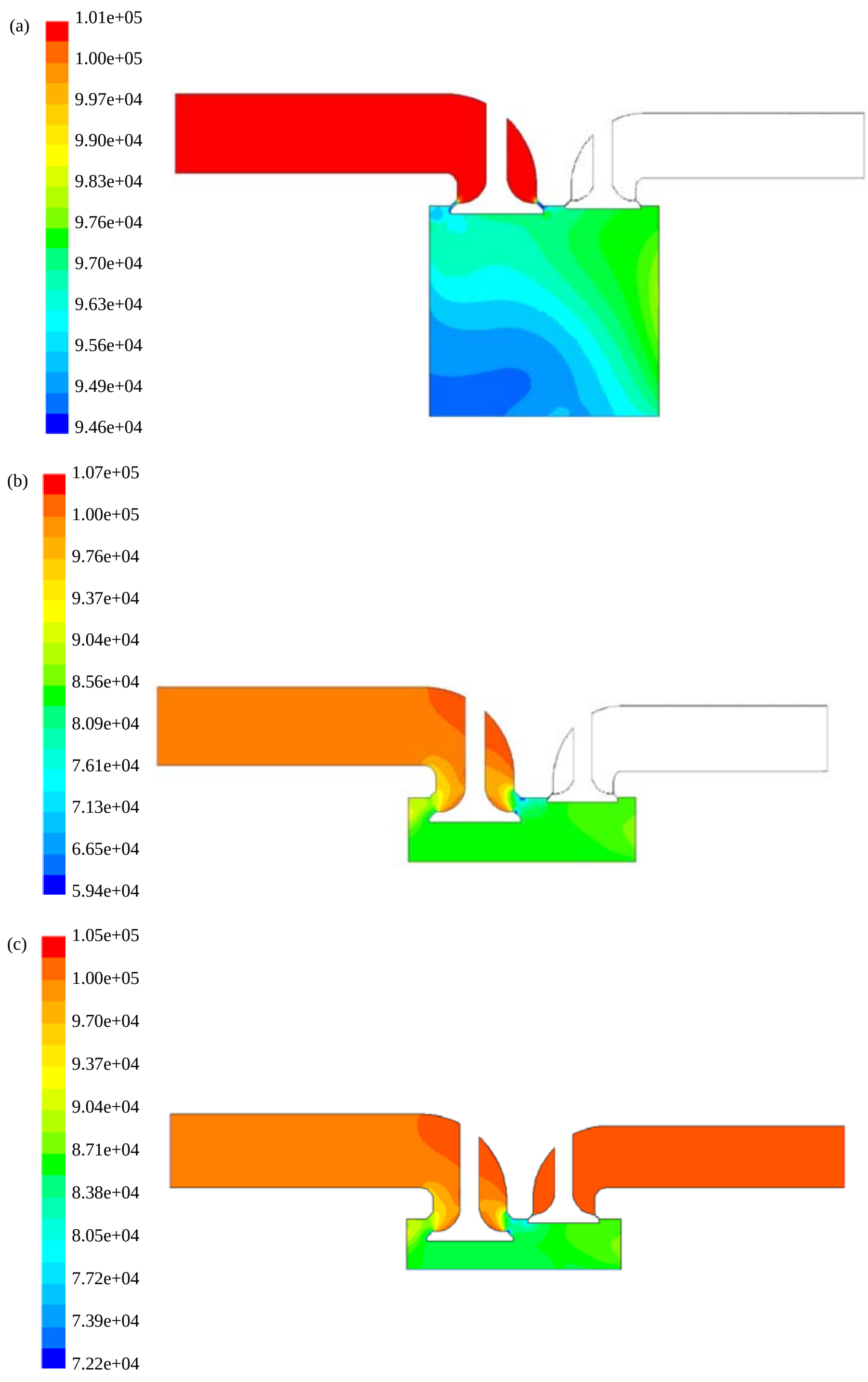

Fig. 5(a-c): Pressure distribution in the inlet manifold for cases under study with crank angle (a) 540°, (b) 414° and (c) 404°

actual engine, as well as to alter the valve shape and dimensions for the same reasons and the angles of the valves on cylinder for flow optimization if required.

The sudden increase in the pressure at first just after inlet valve open was observed. This is because the pressure inside the cylinder is slightly higher than the pipe pressure and also the air valve opens before (TDC) causing a compression wave to propagate towards the valve and results in a pressure rise at that point (Fig. 5). After piston movement changes direction, a depression will promptly develop in the cylinder. This results in a flow of gas from the inlet pipe into the cylinder.

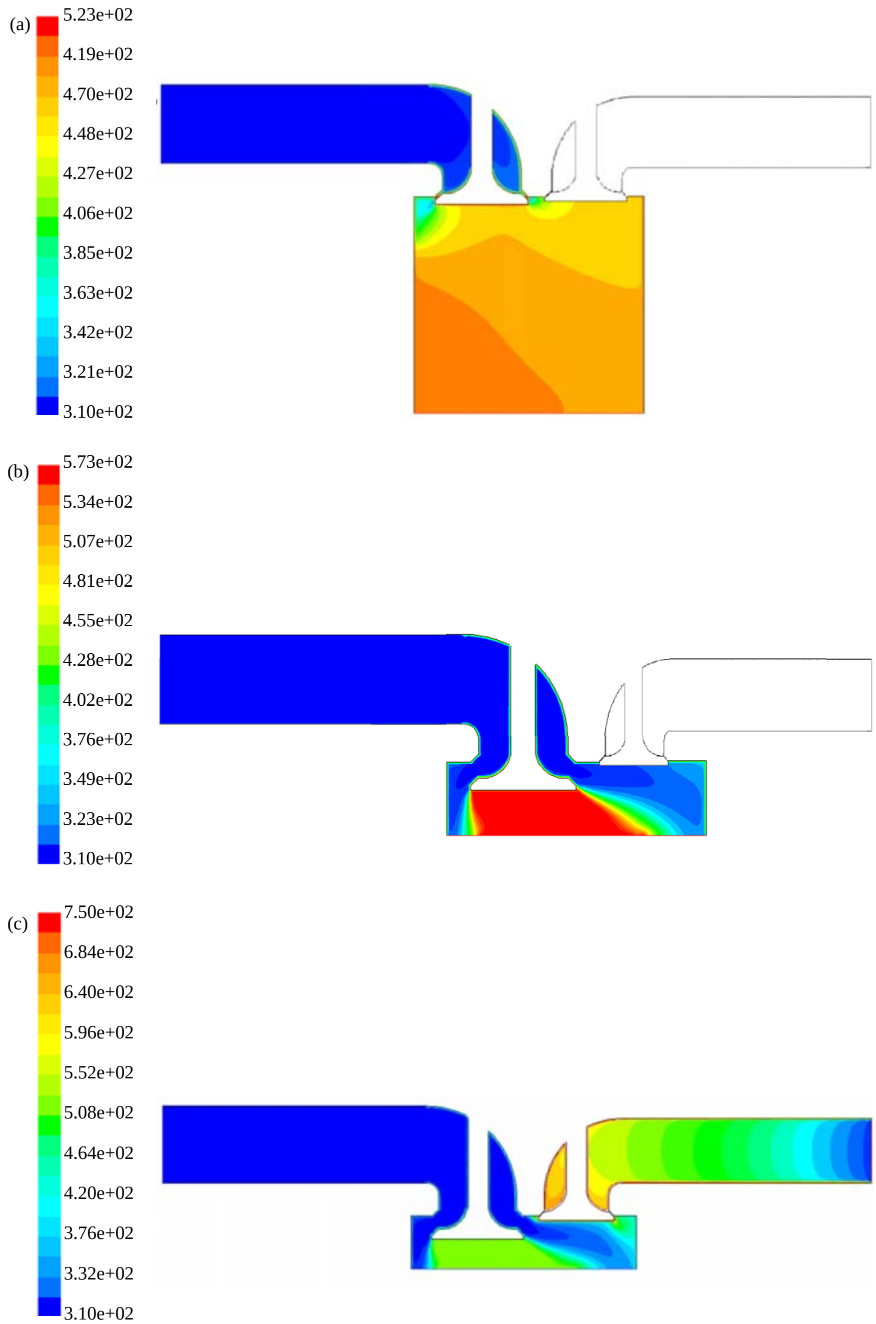

Fig. 6(a-c): Temperature distribution in the inlet manifold for cases under study with crank angle (a) 540°, (b) 414° and (c) 404°

The instantaneous temperature through the inlet pipe, with respect to crank angle, for engine speed (2500 rpm) is shown in Fig. 6. It has a similar trend as the pressure since temperature is pressure dependent. While, the instantaneous velocity through the inlet pipe, with respect to crank angle, for engine speed (2500 rpm) is shown in Fig. 7. The sinusoidal nature of the velocity profile at each pipe end is again due to the wave action. Some back flow is indicated on the curve in Fig. 4c. The velocity profile in the inlet pipe can be divided into three parts: When the inlet valve is closed near BTC, overlap near TDC (low velocity region) and when inlet valve is open (high velocity region). On other hand, Fig. 4c shows that the variation of the flow with the experimental and CFD results, using a 204 mm length pipe intake manifold. These results shows quiet similar trends to those shown in Fig. 7. Likewise, the differences between the experimental and CFD simulation results are reasonable. Results presented show the later to have smaller manifold losses. A comparison effort was carried

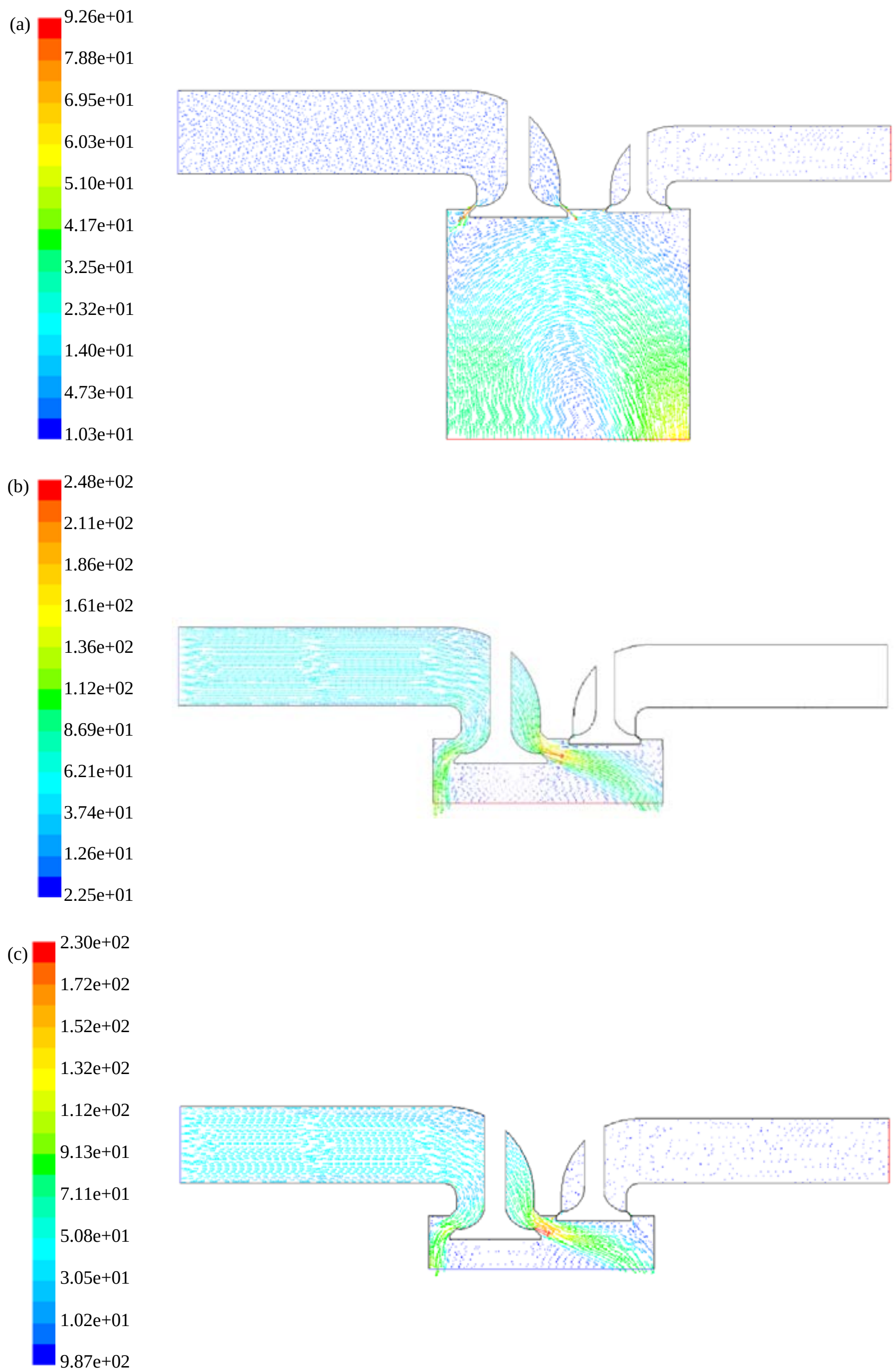

Fig. 7(a-c): Velocity distribution during the inlet manifold for cases under study with crank angle (a) 540°, (b) 414° and (c) 404°

out to investigate the validity of this numerical flow simulation with CFD results by Matsui *et al.*[21] and a good agreement was achieved.

Flow characteristic in exhaust manifold: The effect of back pressure on engine is the pressure that tries to force the exhaust back into the cylinder. In extreme cases too much backpressure can damage the engine. Back pressure rise is a common phenomenon observed for after-treatment strategies utilization in exhaust systems. Possible causes and remedies are required to control exhaust back pressure phenomenon plays vital role, so this study would be useful for automotive sector to understand this important controllable backpressure as an engine operating variable. All the feasible

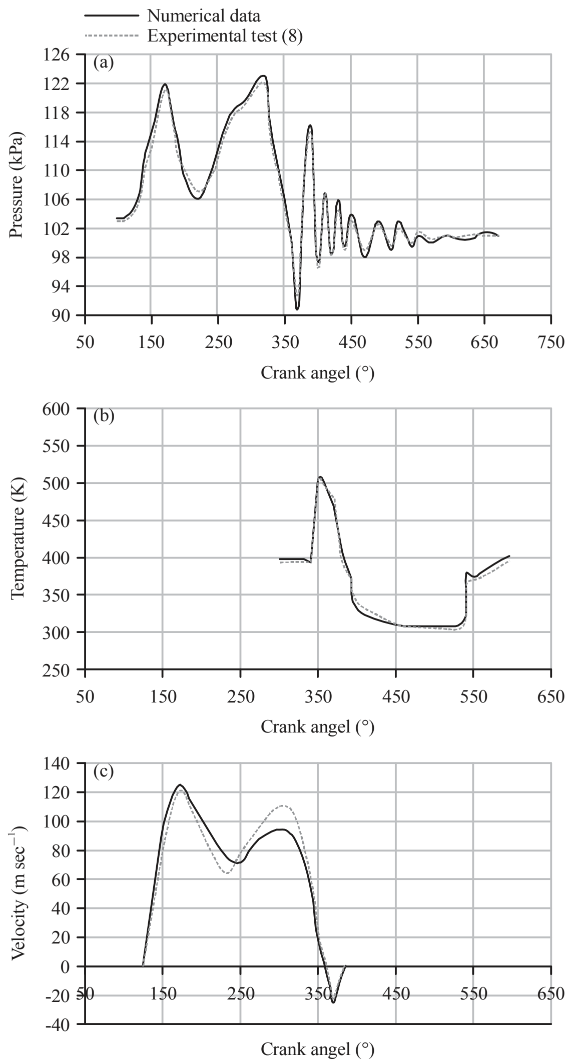

Fig. 8(a-c): (a) Pressure, (b) Temperature and (c) Velocity distribution in the cylinder with respect to crank angle during gas discharge in the exhaust manifold

ways are to be studied for keeping the backpressure value at minimum level, irrespective of engine type and operating conditions. The results obtained from fluent code on pressure, temperature and velocity for the boundary conditions obtained from the experimental tests for engine speed (2500 rpm) show a much better agreement than the other mathematical program as shown in Fig. 8a-c. From the pressures obtained from fluent it is shown that there is good agreement with those from the experimental tests by Matsui *et al.*[21], Durat *et al.*[22] and Jain and Agrawal[23], the former has a better representation for both temperature and velocity which will give the designer a better tool to improve the performance by altering the dimensions and type of pipes to be used in the actual engine.

Pressure distribution in the exhaust manifold is shown in Fig. 9, it is noticed that the transient pressure is sharply reduced during Exhaust Valve Open (EVO) in the "Blow down" period and before Inlet Valve Open (IVO) the wave pattern is basically made up of many pressure pulses; these pulses combine to give a single pulse, this is because the number of pulses for a given engine is a function of piston movement (piston length), valve opening and engine speed. By the time the exhaust valve closes, the pressure has dropped to a value less than atmospheric pressure which is again a function of engine speed. The pressure increases again after reaching its minimum and positive pressure in the cylinder gradually builds up. The wave action plus the piston motion both help to draw in more fresh charge and improve the efficiency of the induction process.

The variation in the temperature in cylinder during the gas exchange period is shown in Fig. 10. It is observed that the sequence of events for the temperature variation in the cylinder coincides with that of the pressure variations. It also shows that after Exhaust Valve Open (EVO) the released temperature decreases rapidly and because of the wave action. The temperature will be sustained or slightly raised until Inlet Valve Open (IVO), when fresh air flows in and heat transfer takes place between the fresh charge and the residual gas. This will make the temperature in the cylinder decrease even more rapidly during the blow down period until after Exhaust Valve Closed (EVC), after this the temperature will gradually increase until Inlet Valve Closed (IVC).

The better representation for both temperature and velocity will give the designer a better tool to improve the performance by altering the dimensions and type of pipes to be used in the actual engine for flow optimization. When the gas enters in the combustion chamber, heat transfer by convection and radiation takes place leading to a decrease in the exhaust gas temperature, while a further heating of exhaust air as the valve is quite hot and adds energy to the leaving gases.

The distribution of velocity profile in the exhaust pipe is in agreement with experimental data, is shown in Fig. 11. It has a similar trend as the pressure since velocity distribution is pressure dependent. In all velocity vectors it is clear that the velocity close to the valve phase is small compared to the valve sides one and stem and goes up to 230 m sec^{-1} at the valve edges (near chocked flow). Figure 8 shows the variation of the flow velocity measured by the experiment and predicted by the simulation, using a 196 mm

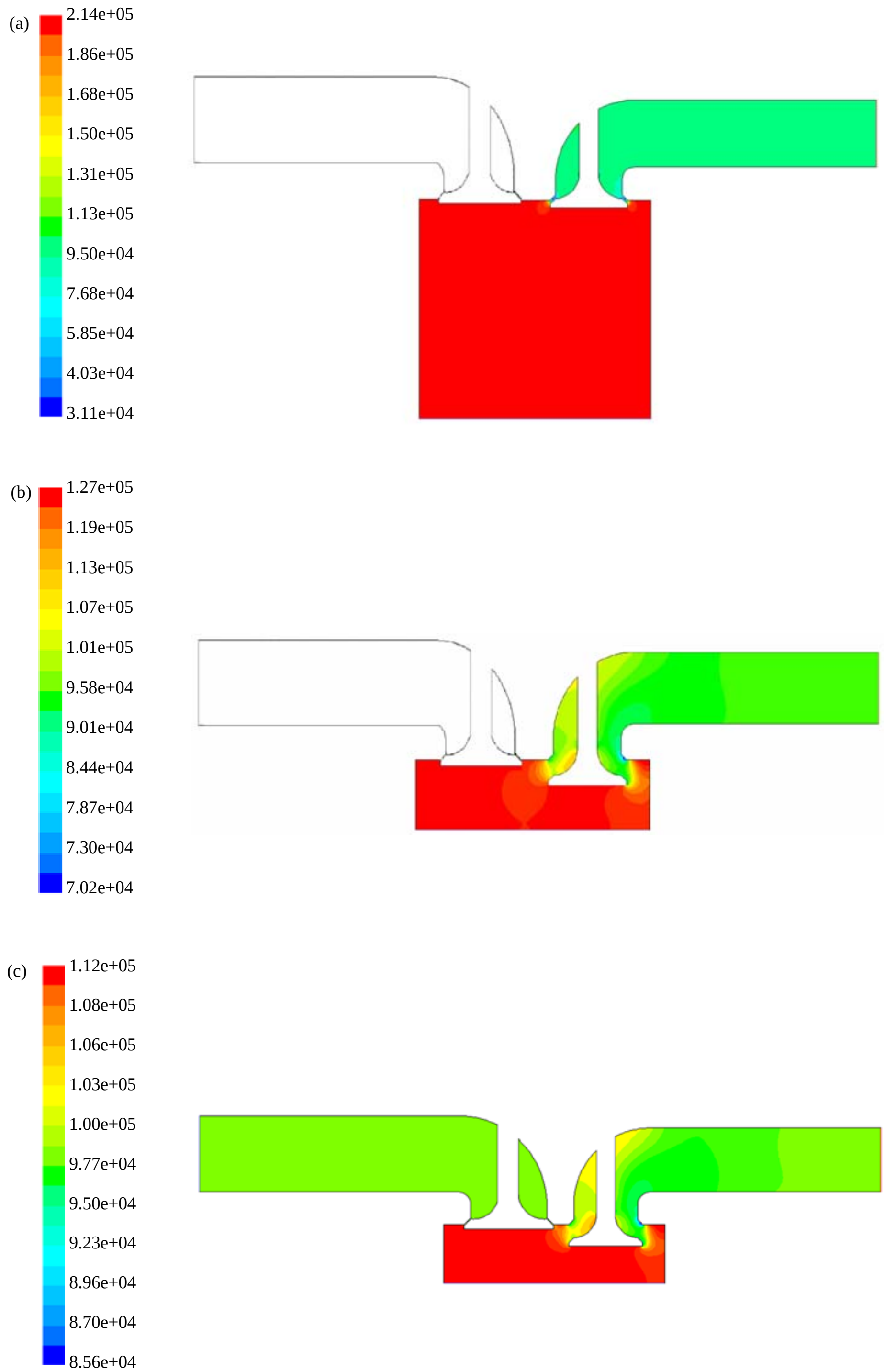

Fig. 9(a-c): (a) Pressure distribution during the exhaust manifold for cases under study with crank angle (a) 184°, (b) 306° and (c) 316°

length pipe exhaust manifold. The results shown quit similar trends to those shown in Fig. 11. Likewise, the differences between the experimental and CFD simulation results are reasonable. Considering the simulation results, as in Fig. 11, provides a guide to the designer of the engine manifolds by improving the geometrical shaping of the exhaust manifold. More so, these investigations shown the effect of back pressure where it is an undesired effect because as the back

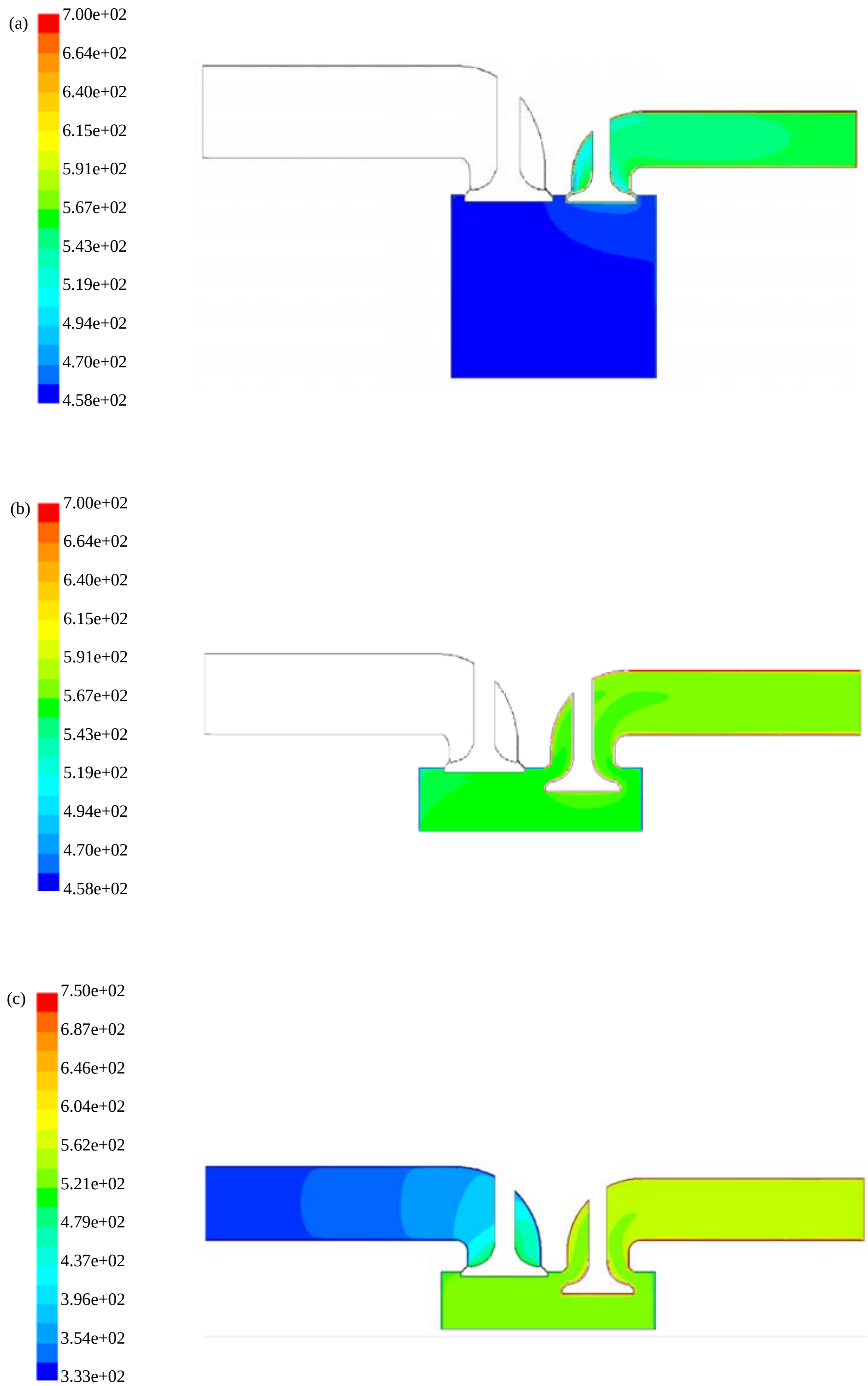

Fig. 10(a-c): Temperature distribution during the exhaust manifold for cases under study with crank angle (a) 184°, (b) 306° and (c) 316°

pressure increases so does the amount of gas residuals in the cylinder head. The increase in weight of these residuals will decrease the volume of the fresh charge, in turn increasing the temperature at the beginning of compression as well.

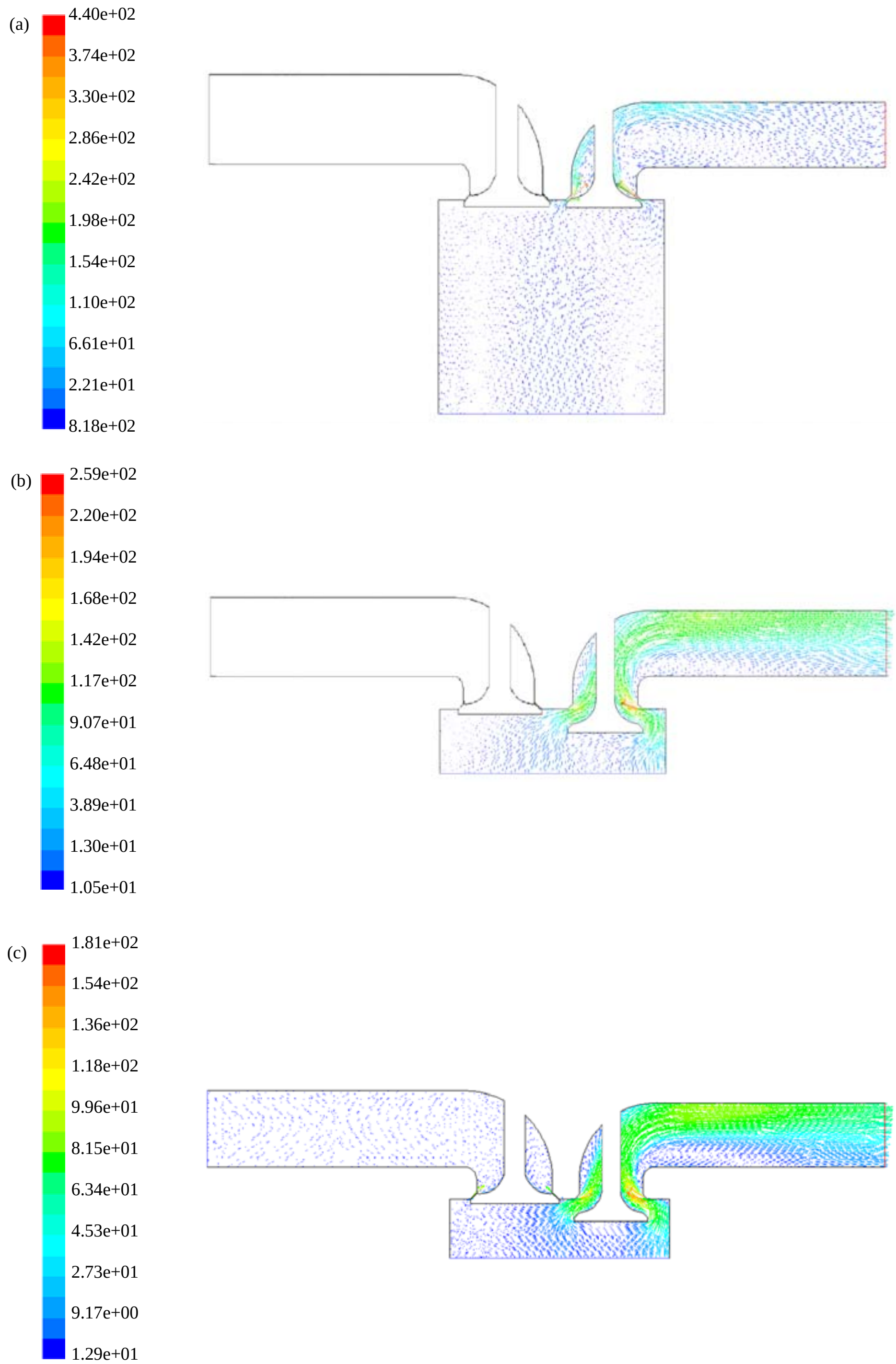

Fig. 11(a-c): Velocity distribution during the exhaust manifold for cases under study with crank angle (a) 184°, (b) 306° and (c) 316°

CONCLUSION

The numerically prediction of intake and exhaust manifolds flow characteristics of pressure, temperature and velocity demonstrated authenticity of experimental data for development of inlet and exhaust manifolds design to increase volumetric efficiency and decrease knock in the engine. At crank angle around 570 and 370°, reversal flows have been noticed to accrue at the inlet and exhaust seats. This behaviour of velocity distribution caused to back pressure at critical points in inlet and exhaust manifolds. Maximum velocity of flue gases is achieved near the outlet of exhaust manifold as the pressure at that position is low so the gases rush out of the exit of the duct. The temperature of flue gases in all the cases cools down as the gases proceed towards the outlet of the exhaust manifold.

SIGNIFICANCE STATEMENTS

- This study simulates the hydrothermal characteristics of the flow at inlet and exhaust manifolds, to provide added knowledge to the designer and researcher in the internal combustion field
- Results from the validated simulation presented clear prediction and visualization of the pressure, temperature and velocity distributions in the engine manifolds
- User design function code has been developed to solve the moving boundary problem during the up/down motion of the piston

ACKNOWLEDGMENT

The authors are obliged to the Universiti Teknologi PETRONAS for providing the facilities of centre for automotive research and energy management (CAREM). The financial support was provided by CAREM is highly appreciated.

REFERENCES

1. Umesh, K.S., V.K. Pravin and K. Rajagopal, 2013. CFD analysis and experimental verification of effect of manifold geometry on volumetric efficiency and back pressure for multi-cylinder SI engine. Int. J. Eng. Sci. Res., 3: 342-353.
2. Kanupriya, B., C. Akash, A. Akshay and S. Shiena, 2017. CFD analysis of exhaust manifold of SI engine and comparison of back pressure using alternative fuels. IOSR J. Mech. Civil Eng., 14: 23-29.
3. Pandey, K.M. and R. Bidesh, 2012. CFD analysis of intake valve for port petrol injection SI engine. Global J. Res. Eng. Mech. Mech. Eng., 12: 12-20.
4. Umesh, K.S., V.K. Pravin and K. Rajagopal, 2013. Experimental analysis of optimal geometry for exhaust manifold of multi-cylinder SI engine for optimum performance. Int. J. Automobile Eng. Res. Dev., 3: 11-12.
5. Pearson, R., M. Bassett, P. Virr and S. Lever, 2006. Exhaust system gas-dynamics in internal combustion engines. Proceedings of the ASME Internal Combustion Engine Division Spring Technical Conference, May 7-10, 2006, Aachen, Germany, pp: 187-196.
6. McKee, R.H., G. McCullough, G. Cunningham, J.O. Taylor, N. McDowell, J.T. Taylor and R. McCullough, 2006. Experimental optimisation of manifold and camshaft geometries for a restricted 600cc four-cylinder four-stroke engine. Proceedings of the SAE Small Engine Technology Conference and Exposition, November 13-16, 2006, San Antonio, TX.
7. Safari, M., M. Ghamari and A. Nasirtosi, 2003. Intake manifold optimization by using 3-D CFD analysis. Proceedings of the SAE Small Engine Technology Conference and Exposition, September 16-18, 2003, Madison, Wisconsin, USA.
8. Al-Khishali, K.J.M., M.A. Mashkour and E.S. Omaraa, 2010. Analysis of flow characteristics in inlet and exhaust manifolds of experimental gasoline combustion in a VCR engine. Eng. Technol. J., 28: 1416-1431.
9. Deger, Y., B. Simperl and L.P. Jimenez, 2004. Coupled CFD-FE-analysis for the exhaust manifold of a diesel engine. Proceedings of the ABAQUS Users' Conference, May 25-27, 2004, Boston, Massachusett, pp: 1-10.
10. Rezaei, R., S. Pischinger, P. Adomeit and J. Ewald, 2009. Evaluation of CI in-cylinder flow using optical and numerical techniques. Proceedings of the 9th International Conference on Engines and Vehicles, September 13-18, 2009, Capri, Naples, Italy.
11. Kurniawan, W.H., S. Abdullah and A. Shamsudeen, 2007. Turbulence and heat transfer analysis of intake and compression stroke in automotive 4-stroke direct injection engine. Algerian J. Applied Fluid Mech., 1: 37-50.
12. Cheeda, V.K., R.V. Kumar and G. Nagarajan, 2008. Design and CFD analysis of a regenerator for a turboshaft helicopter engine. Aerospace Sci. Technol., 12: 524-534.
13. Basha, S.M.J., P.I. Prasad and K. Rajagopal, 2009. Simulation of in-cylinder processes in a DI diesel engine with various injection timings. ARPN J. Eng. Applied Sci., 4: 1-7.
14. Hasse, C., V. Sohm and B. Durst, 2010. Numerical investigation of cyclic variations in gasoline engines using a hybrid URANS/LES modeling approach. Comput. Fluids, 39: 25-48.
15. Reveille, B. and A. Duparchy, 2009. 3D CFD analysis of an abnormally rapid combustion phenomenon in downsized gasoline engines. Oil Gas Sci. Technol. Rev. IFP, 4 431 444-10.2516/ogst/2009005.
16. Kurniawan, W.H., S. Abdullah and A. Shamsudeen, 2007. A computational fluid dynamics study of cold-flow analysis for mixture preparation in a motored four-stroke direct injection engine. J. Applied Sci., 7: 2710-2724.
17. Rathnaraj, J.D., 2012. Thermo mechanical fatigue analysis of stainless steel exhaust manifolds. IRACST-Eng. Sci. Technol., 2: 265-267.
18. Satish, S., M. Prithiviraj and S. Hari, 2012. Comparison of predictions obtained on an exhaust manifold analysis using conformal and indirect mapped interface. Proceedings of the International Congress on Computational Mechanics and Simulation, December 9-12, 2012, IIT, Hyderabad.

19. Pulkrabek, W.W., 1997. Engineering Fundamentals of the Internal Combustion Engine. Prentice Hall, New Jersey.
20. Ramasamy, D., K.A. Soon, H.W.G. Briggs and Z.A. Zainal, 2013. Variation of airflow pattern through dissimilar valve lift in a spark ignition engine. J. Chin. Inst. Eng., 36: 1083-1096.
21. Matsui, T.S.H., M. Tsubouchi and M. Katsurada, 2004. Evaluation of CFD tools applied to engine coolant flow analysis. Mitsubihi Motors Tech. Rev., 16: 56-60.
22. Durat, M., Z. Parlak, M. Kapsiz, A. Parlak and F. Ficici, 2013. CFD and experimental analysis on thermal performance of exhaust system of a spark ignition engine. J. Thermal Sci. Technol., 33: 89-99.
23. Jain, S. and A.B. Agrawal, 2013. Coupled thermal-structural finite element analysis for exhaust manifold of an off-road vehicle diesel engine. Int. J. Soft Comput. Eng., 3: 226-230.