ISBN 978-602-53531-8-5

Investigation of Flow Field Modeling on Gasoline Engine of Motor Cycles

Bambang Wahono[1,2], Yanuandri Putrasari[1,2], and Ocktaeck Lim[3,*]

[1] Graduate School of Mechanical Engineering, University of Ulsan, Ulsan, 44610, South Korea

[2] Research Center for Electrical Power and Mechatronics, Indonesian Institute of Sciences (LIPI), Jl. Cisitu No 21/154 D, Gd.20, Bandung 40135, Indonesia

[3]School of Mechanical Engineering, University of Ulsan, Ulsan, 44610, South Korea

*Corresponding author e-mail: otlim@ulsan.ac.kr

Abstract

Internal combustion engine is the best available source of power for transportation sectors. The big issue appears at the efficiency of these engines. Every effort made to improve these engines to obtain the maximum efficiency. The gasoline engine performance is improved by the proper design of inlet manifold, combustion chamber, exhaust manifold, etc. The goal of this study is to investigate the in-cylinder flow characteristic of gasoline engine of motor cycles using Computational Fluid Dynamics (CFD) at three different speed (1500 rpm, 2000 rpm and 2500 rpm) like tumble, swirl, turbulence during cold start condition. The model of the intake port was analyzed by using Converge. The mesh was generated using the polyhedral scheme which includes primarily of tetrahedral mesh elements. The pressure boundary conditions were used to define the fluid pressure at the inlet and outlet of port. The tumble is decreased two folds between the speeds 1500 and 2000 rpm, and down one time in tumble between 2000 and 2500 rpm. The swirl in negative axis is increased two folds between the speeds 1500 and 2000 rpm, and increased a half time between 2000 and 2500 rpm. The results indicate that the CFD model can be used as a tool to investigate deeply the effect of various parts of port for optimization like manifold, spray, film, mixture formation and combustion. Finally, this study gives an effect to reduce the number of experiments to be carried out for arriving at the final optimized system.

Keywords: Gasoline Engine, In-Cylinder, CFD, Speed, Converge

1. Introduction

In the last decades, internal combustion engine is the best available reliable power source for many sectors and applications especially in transportation sector. The big issue arises at the internal combustion engine is engine efficiency performance and low emission. Every effort made to improve the internal combustion engines tends to get the maximum efficiency performance and low emission. This is demanded to protect the environment and energy conservation. The performances of the internal combustion engines are obtained by proper design of intake manifold, exhaust manifold, combustion chamber, piston etc. In other side, to achieve the maximum efficiency performance and low emission, it is important to improve the combustion characteristics, that means it reduces the pollutant or unburned elements in exhaust gas by achieving a homogeneous mixture with controlled in-cylinder flow like swirl and tumble flow (Li et al., 2001; Nonaka et al., 2004).

The Swirl flow is defined as a rotation of the charge about the cylinder axis and the tumble flow is defined as a rotation orthogonal to the cylinder axis (Lumley, 1999). These two flows occur simultaneously inside the cylinder. It should be noted that there is a range of ideal levels of swirl, tumble, flow speed and turbulent kinetic energy (TKE). Small levels of these variables lead to slow flame propagation, poor mixture formation resulting in reduced efficiency. But very high levels of those can lead to increased heat transfer to walls, reduced volumetric efficiency and even flame quenching and incomplete burning (Li et al., 2001; Hill and Zhang, 1994; Aita et al., 1991; Wheeler et al., 2013). The flow field model inside an engine especially in cylinder can be visualized by several ways such as using experimental techniques like Laser Doppler Anemometry (LDA) (Li et al., 2001; Le Coz et al., 1990; Fansler and French, 1988) and Particle image velocimetry (PIV) (Li et al., 2001; Rouland et al., 1997; Choi et al., 1999; Reuss et al., 2000). These techniques are good but expensive (Kurniawan et al., 2007). Other choice to visualize the flow field model is using numerical techniques such as CFD. It can lead to very accurate results, which can help detecting problems in the engine design and accelerate the project phase.

A CFD analysis is based on the continuity, Navier-Stokes and energy equations along with some modelling for turbulence. The turbulence models most used industrially are based on the Reynolds Averaged Navier Stokes (RANS) approach, and among those the most widely validated belong to the k-ε family (Versteeg and Malalasekera, 2007).

As we know that the engine cycle of typical internal combustion engines consists of four consecutive processes i.e. intake process, compression process, expansion process and exhaust process. Of these four processes, the intake and compression stroke process are two of the most important processes which influences the pattern of air flow structure coming inside cylinder during intake stroke and generates the condition needed for the fuel injection during the compression stroke. Especially for intake manifold, a deep knowledge of the intake processes and compression stroke process are basic science to design and optimize modern internal combustion engines efficiently.

The objective of this research is analysis of flow field model especially in cylinder engine by numerical simulations that performed in a four stroke of gasoline engine of motor cycle under cold flow conditions at three different speeds (1500 rpm, 2000 rpm and 2500 rpm) to investigate the flow characteristics like swirl, tumble, turbulence during cold start condition.

2. Methodology

2.1 Simulation Setup

The internal combustion engine is a heat engine that converts chemical energy in a fuel into mechanical energy, usually made available on a rotating output shaft. The wide range of internal combustion engines is classified and they have its own advantages and disadvantages. According to the type of the fuel used the engine is classified as follows: gasoline engine, diesel engine and gas engine. The gasoline engine used in this research. The detailed specification of the gasoline engine selected for the simulation is given in Table 1. By the CFD Numerical simulation, the gasoline engine 249 cc used in this study. In the beginning research, the important and the main component of this research is engine.

In this study, the CFD analysis is carried out using Converge, a commercial CFD package. The Converge uses a fully automatic structured grid generation technique for meshing a geometry during run time (Converge, 2017). The grid independence study is carried out for the engine under consideration (Krishna and Mallikarjuna, 2015). The flow turbulence is analyzed using the renormalized group (RNG) k-e model (Krishna, et al., 2013; Yakhot et al., 1994). The finite volume-based implicit discretization procedure is used to solve the discretized Navier–Stokes' equations on a Cartesian grid. The in-cylinder flow is modeled by solving mass, momentum and energy conservation equations along with the equations for species, turbulent kinetic energy (TKE) and turbulent dissipation rate at each cell and at each time step. The ''pressure implicit for splitting of operator" (PISO) algorithm is used to solve the pressure-velocity coupling.

Table 1: Specification of gasoline engine

Parameter	Value
Type air / Urethane	DOHC 8 valve 75 °V type
Number of cylinder	2
Displacement	249cc
Bore	57 mm
Stroke	48.8 mm
Compression ratio	10.2: 1
Maximum output	25.5 ps / 9000 rpm
Maximum torque	2.21kg-m / 7000 rpm
Fuel injection	Electric Port Fuel Injection

2.2 The CAD Model

The engine model was scanned by three-coordinate measuring machine to gain its clouding points figure. Then the figure was putted into the Solid Work or other CAD software to be modified. Finally, three-dimensional CAD model of the engine was obtained. It is showed in figure 1. Then this CAD model was modified to become a simulation calculation model. The fluid volume is extracted in one step and the inlet and outlet boundaries are created by renaming the surfaces. Converge uses a fully automatic structured grid generation technique for meshing the geometry during run time. However, the grid size and total number of cells can be controlled by the user through the base grid size, adaptive mesh refinement and grid embedding controls.

Fig. 1: The CAD model of gasoline engine

2.3 Boundary Condition

In this study, the piston and valves are defined as moving boundaries. The domain is divided into three regions i.e. cylinder, intake and exhaust. All the three regions are connected based on events described by the valve closing and opening. The inlet and outlet pressure, and temperature are defined at 1 bar and 300 K respectively. An initial turbulence level of 10% is specified at the inlet (Sagayaraj et al., 2013). The summary of boundary conditions is given in Table 2.

Table 2: Boundary conditions

Parameter	Value
Inlet	Pressure inlet at 1 bar 300K
Outlet	Pressure outlet at 1bar 300K
Piston and valves	Moving walls
Cylinder, intake and exhaust ports	Stationary walls

In this study, the global transport number are prandtl number with 0.9 and Schmidt number with 0.78. A length scale of 10% of the port diameter at the inlet plane were presented to estimate the inlet boundary condition on turbulent kinetic energy (Tke) and its dissipation rate. The choice of differencing scheme will affect the convergence rate and accuracy of the final computational solution. Lower order schemes tend to be more stable but introduce numerical viscosity into the solution, while higher order schemes are more accurate but require more computer time to solve and are less stable. Present study makes use of first order scheme, which is stated to be far more robust and stable. Currently the most popular turbulence model, which is used in a practical setting, is the two-equation k-ε model. This model employs two additional transport equations one for turbulence kinetic energy and another one for the turbulent dissipation rate (Tdr). Near wall treatment is handled through generalized wall functions. In this study, the flow turbulence is renormalized group (RNG) k-e, the in-cylinder flow is modeled by solving mass, momentum and energy conservation, turbulent kinetic energy (TKE) and turbulent dissipation rate at each cell and at each time step and the pressure implicit for splitting of operator (PISO) algorithm is used to solve the pressure-velocity coupling. This model is well established and the most widely validated turbulence model.

3. Result and Discussions

3.1. In-cylinder Tumble ratio

The gasoline engines are normally designed to be tumble oriented. The pent roof combustion chamber and the orientation of the intake port geometry helps in the generation of tumbling motion inside the engine cylinder. Tumble flow has been studied by using the numerical simulation method. The change in tumble flow of in-cylinder according to the crank angle (around both X-axis and Y-axis) is shown in Fig. 2. Fig. 2 shows the temporal variation of tumble flow for the three speeds 1500, 2000 and 2500 rpm. The rotation of vortices about the axis Y (T_{ry}) is called as normal tumble about y coordinate and the tumbling vortices about the axis X is denoted as cross tumble (T_{rx}). This tumble ratio is the average from T_{rx} and T_{ry}. Tumble motion inside the engine cylinder can be divided into three phases as generation, stabilization and destruction. Generation phase occurs usually during the intake stroke which proceeds from 360 up to 420° CA. The stabilization and spin up phase occur due to the upward moving piston. Because of the moving piston the spin up phase enhances the tumble motion again up to 300 degrees. The tumble destruction phase results in increased turbulence. This can be confirmed with reference to Fig 2 where there is increase in the turbulence level starting from 200°CA and lasts up to 330°CA. Regarding the effect of speed, the tumble is decreased two folds between the speeds 1500 and 2000 rpm, whereas down one time in tumble between 2000 and 2500 rpm. This may be attributed to the poor strength of the intake generated tumbling vortices which are not sustained during the stabilization and spin up phase.

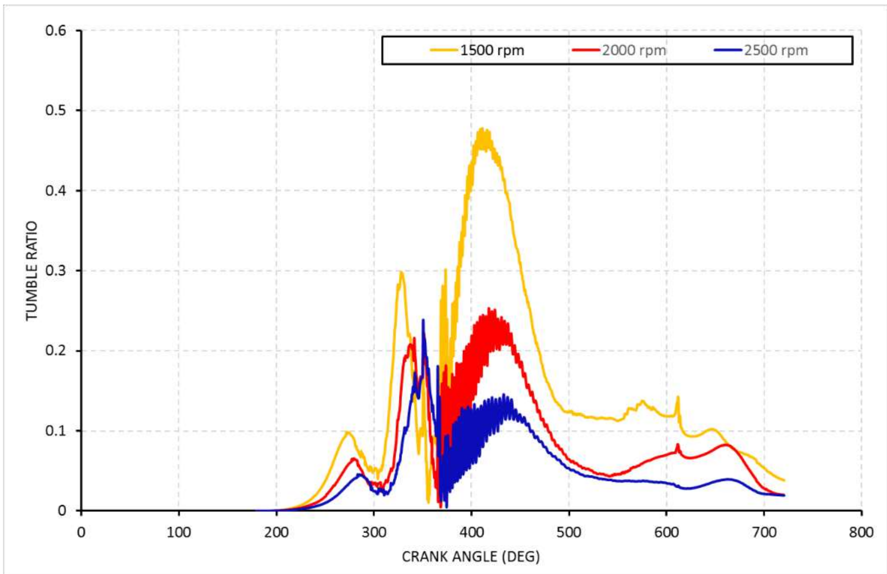

Fig 2: In-cylinder tumble ratio in variation speed

3.2. In-cylinder Swirl ratio

Swirl is created by bringing the intake flow into the cylinder with an initial angular momentum and the interaction of the intake jet/cylinder wall/piston face. While many other flow structures generated during the intake process decay very quickly, swirl can survive throughout the compression process, and even into combustion and expansion processes. The swirl flow has been studied by using the numerical simulation method. The change in swirl flow of in-cylinder according to the crank angle is shown in Fig. 3. Swirl flow is generated even with the standard condition of in-cylinder flow at full throttle. Fig. 3 shows the temporal variation of swirl flow for the three speeds 1500, 2000 and 2500 rpm. Fig. 3 shows the predicted swirl ratio versus crank angle during the exhaust, intake and compression strokes. It isn't appeared in the exhaust process and begin appeared and formed during the induction process. The swirl ratio is greatest at the middle of the intake stroke (approximately 350° CA) and has a slight drop after that point as angular momentum is dissipated owing to friction at the walls and turbulent dissipation within the fluid. However, the swirl is fairly stable and the swirl ratio remains almost constant during the early phase of the compression stroke. The swirl ratio increases during the late compression stroke. This phenomenon is expected as the tangential velocity of the swirling air flow is increased owing to the compact bowl-in-piston/combustion chamber interaction. Regarding the effect of speed, the swirl in negative axis is increased two folds between the speeds 1500 and 2000 rpm, and increased a half time between 2000 and 2500 rpm. This may be attributed to the poor strength of the intake generated swirling vortices which are not sustained during the stabilization and spin up phase.

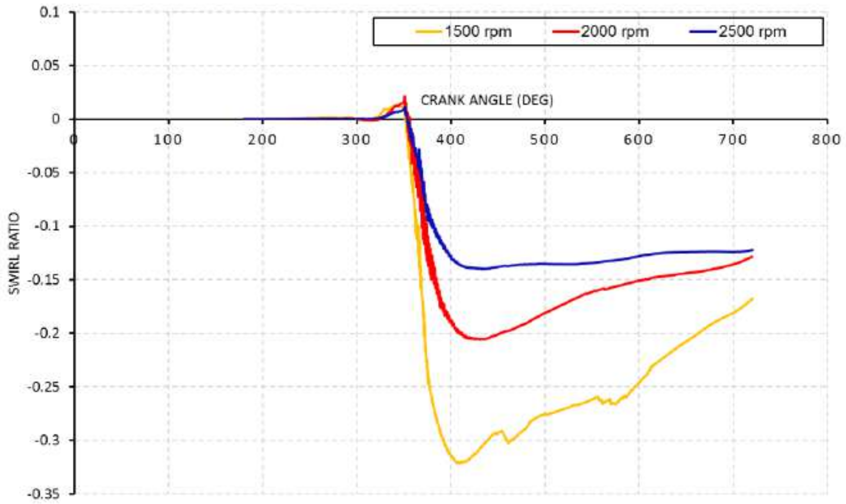

Fig 3: In-cylinder Swirl ratio in variation speed

3.3. In-cylinder Turbulent Kinetic Energy (TKE)

Turbulent Kinetic Energy is the mean kinetic energy per unit mass associated with eddies in turbulent flow. Physically, the turbulence kinetic energy is characterized by measured root-mean-square (RMS) velocity fluctuations. The turbulent kinetic energy (Tke) is an important parameter to determine the turbulent viscosity. Figure 4 shows the Turbulent Kinetic Energy (Tke) versus crank angle. There are two peaks in turbulent kinetic energy seen in Fig. 4. The first peak appears during the intake stroke despite a slight decline and a rebound, and is related to turbulence generated as the air flows through the intake valve curtain area. It is observed that this configuration affects the turbulence of the fluid inside the cylinder. The second peak appears lower than the first peak and occur in compression stroke process. It reaches the peak value during the maximum valve open condition (400°CA) and converge from 700 °CA. The effect of the speed is the higher the speed the lower the value tke. The variation of Tke is probably due to different level of air induced through the inlet manifold.

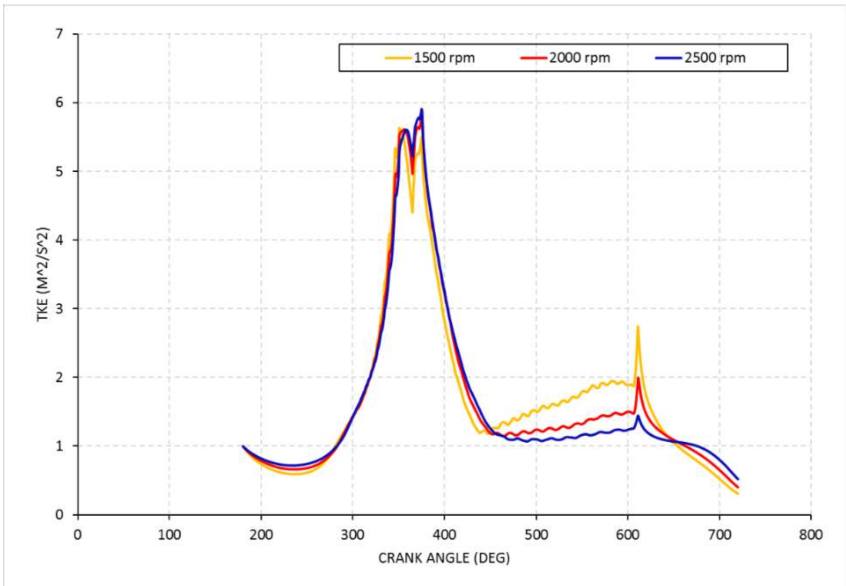

Fig 4: In-cylinder TKE in variation speed

3.3. In-cylinder Turbulent Dissipation rate (TDR)

Figure 4 shows the Turbulent Dissipation rate (TDR) versus crank angle. This figure similar with the turbulent kinetic energy. There are two peaks in turbulent dissipation rate seen in Fig. 4. The first peak appears during the intake stroke despite a slight decline and a rebound, and is related to turbulence generated as the air flows through the intake valve curtain area. It is observed that this configuration affects the turbulence of the fluid inside the cylinder. The second peak appears lower than the first peak and occur in compression stroke process. The effect of the speed is the higher the speed the lower the value tke. The variation of Tke is probably due to different level of air induced through the inlet manifold.

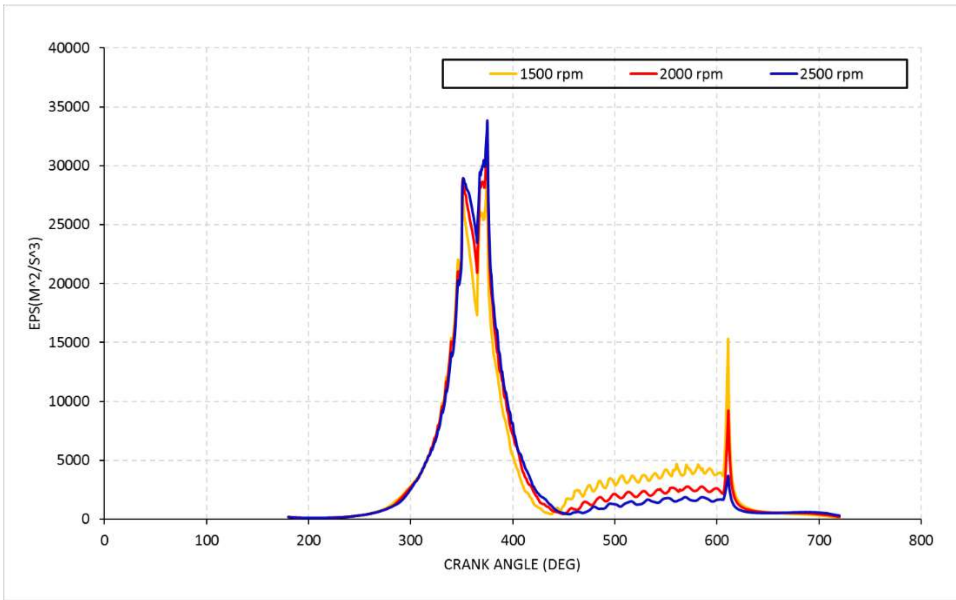

Fig 4: In-cylinder TDR in variation speed

4. Conclusion

A three-dimensional unsteady turbulent compressible Navier–Stokes solver, Converge, was utilized in the present study to investigate the in-cylinder flow field of a four-valve gasoline direct injection engine of motor cycle. The following conclusions were found.

1. The tumble flow is formed during the late intake stroke and then compressed during the compression stroke. It is broken down at the end of the compression stroke, which results in enhanced turbulence kinetic energy. Such flow phenomena will be important to improve atomization distribution of the fuel spray in the engine operation.
2. Swirl flow can survive through the compression stroke. The swirl ratio even increases with crank angle at the end of the compression stroke.
3. The results indicate that the CFD model can be used as a tool to investigate deeply the effect of various parts of port for optimization like manifold, spray, film, mixture formation and combustion. Finally, this study gives an effect to reduce the number of experiments to be carried out for arriving at the final optimized system.

5. Nomenclature

CFD: Computational Fluid Mechanic

PFI: Port Fuel Injection

TKE: Turbulent Kinetic Energy

TDR: Turbulent Dissipation Rate

RMS: root-mean-square

6. Acknowledgments

This research was financially supported by CEFV (Centre for Environmentally Friendly Vehicle) as Global Top Project of KMOE (2016002070009, Development of Engine System and Adapting Vehicle for Model 110 cc and 300 cc Correspond to EURO-5 Emission). This research was supported by The Leading Human Resource Training Program of Regional Neo industry through the National Research Foundation of Korea (NRF) funded by The Ministry of Science, ICT and Future Planning (2016H1D5A1908826). This research was also supported by the Industrial Strategic technology development program (10053151, Development of the 800 kPa Fuel System of a High Pressure Precision Control for NGV) funded by the Ministry of Trade, Industry & Energy (MI, Korea).

7. References

Li, Y., Zhao, H., Peng, Z., and Ladommatos, N., "Analysis of Tumble and Swirl Motions in a Four-Valve SI Engine," SAE Technical Paper 2001-01-3555, 2001, doi: 10.4271/2001-01- 3555.

Nonaka, Y., Horikawa, A., Nonaka, Y., Hirokawa, M. et al., "Gas Flow Simulation and Visualization in Cylinder of Motor-Cycle Engine," SAE Technical Paper 2004-32-0004, 2004, https://doi.org/10.4271/2004-32-0004.

Lumley, L. J., "Engines, an Introduction," Cambridge University Press, Cambridge, 1999.

Hill, P. G.; Zhang, D., "The effect of swirl and tumble on combustion in spark ignition engines", Progress in Energy and Combustion Science, 20(5):373-429, 1994, doi: 10.1016/0360-1285(94)90010-8.

Aita, S., Tabbal, A., Munck, G., Montmayeur, N. et al.,"Numerical Simulation of Swirling Port-Valve-Cylinder Flow in Diesel Engines," SAE Technical Paper 910263, 1991, doi:10.4271/910263.

Wheeler, J., Polovina, D., Ramanathan, S., Roth, K. et al., "Increasing EGR Tolerance using High Tumble in a Modern GTDI Engine for Improved Low-Speed Performance," SAE Technical Paper 2013-01-1123, 2013, doi: 10.4271/2013-01-1123.

Le Coz, J., Henriot, S., and Pinchon, P., "An Experimental and Computational Analysis of the Flow Field in a Four-Valve Spark Ignition Engine-Focus on Cycle-Resolved Turbulence," SAE Technical Paper 900056, 1990, doi: 10.4271/900056.

Fansler, T. and French, D., "Cycle-Resolved Laser-Velocimetry Measurements in a Reentrant-Bowl-in-Piston Engine," SAE Technical Paper 880377, 1988, doi: 10.4271/880377.

Rouland, E., Trinité, M., Dionnet, F., Floch, A. et al., "Particle Image Velocimetry Measurements in a High Tumble Engine for In-Cylinder Flow Structure Analysis," SAE Technical Paper 972831, 1997, doi: 10.4271/972831.

Choi, K., Park, J., Lee, N., Yu, C. et al., "A Research on Fuel Spray and Air Flow Fields for Spark-Ignited Direct Injection using Laser Measurement Technology," SAE Technical Paper 1999-01-0503, 1999, doi: 10.4271/1999-01-0503.

Reuss, D., "Cyclic Variability of Large-Scale Turbulent Structures in Directed and Undirected IC Engine Flows," SAE Technical Paper 2000-01-0246, 2000, doi: 10.4271/2000-01-0246.

Kurniawan, W. H.; Abdullah, S.; Shamsudeen, A., "A computational fluid dynamics study of cold-flow analysis for mixture preparation in a motored four-stroke direct injection engine". Journal of applied sciences 7(19):2007-2724, 2007.

Versteeg, H.; Malalasekera, W. "An Introduction to Computational Fluid Dynamics: The Finite Volume Method". 2a edition. Prentice Hall, 2007. 520p.

CONVERGE v2.3.21, Theory Manual, Convergent Science Inc, 2017.

A.S. Krishna, J.M. Mallikarjuna, Effect of fuel injector location on the equivalence ratio near the spark plug in a GDI engine – a CFD analysis, in: 24th National Conference on Internal Combustion Engines and Combustion, Oct 30th –Nov 1st, Dehradun India, 2015.

A.S. Krishna, J.M. Mallikarjuna, K. Davinder, Y.R. Babu, Incylinder flow analysis in a two-stroke engine – a comparison of different turbulence models using CFD, SAE paper no. 2013-01-1085, 2013.

V. Yakhot, S.A. Orszag, S. Thangam, T.B. Gatski, C.G. Speziale, Development of turbulence models for shear flows by a double expansion technique, Phys. Fluids A4 (7) (1994).

Sagayaraj, A. G., J. M. Mallikarjuna, V. Ganesan. Energy efficient piston configuration for effective air motion – A CFD study, Applied Energy, Volume 102, February 2013, Pages 347-354.