Air Swirl and Flow Field Study of Diesel Engine Inlet Manifolds and Intake Valves

1. Aadesh J Srivastava, Tolani Maritime Institute, Pune, aadeshsrivastava.vasu@gmail.com
2. Akshat Gupta, Tolani Maritime Institute, Pune, akshatsml@gmail.com

Abstract: For any diesel engine driven vehicle, it is important to check the performance of the engine from time to time so as to ascertain working condition and fault finding.
Inlet air through the inlet manifolds has a great impact on the diesel engine performance. The objective of the present analysis is to optimize the air flow through intake manifold and masked inlet valves of a diesel engine. The present study also predicts the effect of induction swirl and turbulence over different valves design.
Modeling of cylinder inlet manifolds and inlet valves with masking is created and meshed using commercially available C.F.D. (fluent) software. Pressure boundary conditions were used to define the fluid pressure at the inlet of air intake system.
The mean air inlet velocity vectors and pressure contours were plotted to give informative pictures of flow field which helps to understand the effect of induction swirl inside the cylinder. The predicted flow structure, swirl velocity and variation of turbulence inside the cylinder with different valves and manifolds were plotted using fluent.
C.F.D. contributes to the understanding of flow and also the detailed information where measurements are rather difficult to made. This in effect will reduce the number of experiments to be carried out for arriving at final optimized system for efficient diesel engine.
The results indicate that the CFD model can be used as a tool to understand the effect of various parts of air intake system for optimization.

Keywords — Engine, intake manifold, C.F.D., Swirl, Turbulence.

1. INTRODUCTION

Major objectives of engine designers today are to achieve the twin goals of best performance and lowest possible emission levels. To maximize the mass of air inducted into the cylinder during the suction stroke, the intake manifold design, which plays an important role, has to be optimized. The design becomes more complex in case of a multicylinder engine as air has to be distributed equally in all the cylinders. Hence, configuration of **manifold geometry** becomes an important criterion for the engine design. Achieving this by means of experimental methods would cost time and money. There is a need for **computational fluid dynamics (CFD)** method (numerical method), which could estimate the volumetric efficiency of the engine during the design stage itself, without undergoing any time consuming experiments. Also mapping the total pressure distribution at the manifold, port and valve is an effective method for analyzing computational prediction of the flow separation process in the region upstream of the valve stem and in the vicinity of the valve seat, because the total pressure is influenced by the mean.

Nowadays, with the availability of powerful computers, the CFD prediction methods for **in-cylinder flow** of IC engines have become popular. They can give very useful information regarding the **flow pattern** and has the potential to reduce the total development time of the intake system of an IC engine. Engine manufactures require precise engine design to bring the end product to the market in a short time period and hence CFD codes play an important role in IC engine design.

1.1.Intake System

The whole combination of devices through which air or air mixture flows and gets into the engine cylinder is called the **Engine Intake System**. The intake system can have several functions:-

I. Filter the air entering the engine.
II. Monitor and control the temperature of air entering the engine.
III. Monitor the amount of air fuel mixture flowing into the engine.
IV. To ensure the charge received by each cylinder unit is well mixed and has the same physical and chemical characteristics.
V. To attain a high volumetric efficiency at full throttle.

In competition automobile industry, Performance demand is critical as small improvements may give great advantages over competitors. The following are the major concerns considered by the competitors in automobile industry.

- Fuel crisis
- Pollution control technologies.
- Efficiency, compact size and cost of an engine

Diesel engine has become the most popular due to cost over the fuel. Currently the diesel engine is facing some of its greatest technological challenges concerns over public health and the environment has resulted in strict legislation covering the gaseous and particulate emission from engines. This legislation has forced diesel engine manufacturers to find ever more complex and creative ways to solve the emissions problem. “Exhaust gas recirculation” (EGR) is one of the major technologies that have been developed to reduce emission of oxides of nitrogen (NOx). Therefore potential to study the intake exhaust flow in a multicylinder four-stroke engine especially using CFD techniques for getting more insight about the flow field, thereby improve the intake manifold configuration, to obtain better performance from the engine.

1.2.Intake Manifold

The intake manifold is attached to the cylinder head. It is a metal casing that bolts over and covers the intake ports in the cylinder heads. The functions of intake manifold are:-

I. Evenly distribute air to each engine cylinder.
II. Carry air into the cylinder ports.

III. Assist in filling the engine combustion chamber with a fuel air mixture to the maximum extent possible.
IV. Assist in creating turbulence and air swirl inside the combustion chamber.

The intake and compression stroke is one 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. As a result of the high velocity inside the internal combustion engine (ICE) , in cylinder flows are typically turbulent. The exception to this is the flows in the corners and small crevices of the combustion chamber where the close distance of the walls diminished out turbulence.

In today's world, major objectives of engine designers are to achieve the twin goals of best performance and lowest possible emission levels. To maximise the mass of air inducted into the cylinder during the suction stroke, the intake manifold design, which plays an important role, has to be optimised. The design becomes more complex in case of a multicylinder engine as air has to be distributed equally in all the cylinders. Hence, configuration of manifold geometry becomes an important criterion for the engine design.

A manifold with circular cross section is desirable. This cross section offers least resistance to flow. Such a cross section will be most desirable if only air or gas is to be distributed among the cylinders.

1.3.Masked Inlet Valves

The purpose of masked inlet valves is to create a more atomised mist of air fuel mixture entering into the combustion chamber giving a more and complete burn of fuel.

A poppet type valve having the addition of a shroud or mask for part of circumference under the valve head.

The objective of masked valve used in this paper is to direct the incoming gases into a rotation of the bore and so increase turbulence to enhance through combustion of the air/gas mixture.

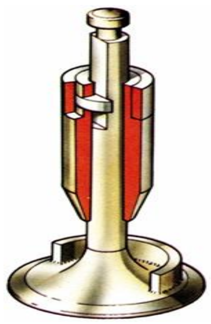

Figure 1. Intake Valve

2. <u>Simulation and Data</u>

2.1. Design Parameters for Intake Manifold

To design an optimal intake manifold, following parameters should be taken into consideration:

- Uniform distribution of air to all cylinders.
- Minimum possible resistance in IM runners.

Proper designs of IM profile helps to reduce the sudden raise in pressure waves which improve induction process and also eliminate the unnecessary turbulence and eddies inside the intake manifold.

2.2.Turbulence

Turbulence is generated whenever air flows quickly past a stationary surface, but rapidly decays away through viscosity once the bulk air speed reduces. So modern thinking is to use careful design of the engine's inlet ports. By aiming the intake flow correctly, rapid air motion is set up during the induction stroke.

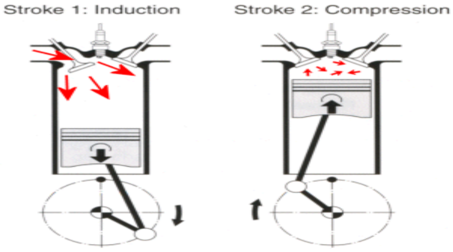

Figure 2

It is clear from Figure rapid motion breaks down into turbulence as the piston rises on the compression stroke, and if the engine is correctly designed, hits just the right level at the point of ignition. This "swirl" technology has been standard on engines for decades and is well understood.

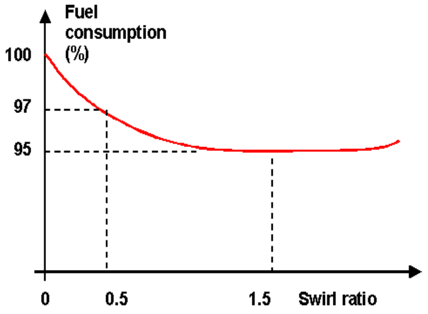

Figure 3. Fuel consumption Vs Swirl ratio

The amount of turbulence an engine should have is a compromise. The number swirl ratio is used to characterise the level of swirl, where 0.3 would represent quite low swirl and 1.5 pretty high, for a petrol engine.
Since generating more swirl requires more restrictive inlet ports, values around 0.5 to 1.0 are usually found in production engines. Adding more swirl speeds up the burn, less swirl slows it down.
Most "fuel saving devices" that claim to speed up the burn says that this improves fuel economy. To some extent that's right, but only at levels lower than found in most production engines. A very slow burn gives bad economy because the fuel is still burning when the exhaust valve opens! Theoretically the best efficiency would be obtained by an instantaneous burn, but this would produce an extremely high in-cylinder temperature and so the heat loss to the cylinder walls would be much higher.

The overall effect is something like this: The optimum is typically around 1.5 to 2.0, but the fuel economy loss from dropping to around 0.5 is very small and gives much better "breathing", so improving power. This is the kind of swirl level that most modern engines operate with. I spent many years working on engines with various swirl ratios, and indeed where the swirl ratio could be altered while the engine was running (high at part load, low at full load), but in the end the higher turbulence levels simply did not give enough benefit to justify the cost and/or reduced performance. Some engines do employ variable-swirl technology, such as Vauxhall (Opel)'s new Twinport engine, and some Fords. Partly this is because there is a slight economy benefit, but mainly because it allows use of high levels of valve overlap or exhaust gas recirculation while still giving a stable burn. Normally high overlap or EGR leads to rough engine running; adding turbulence increases the engine's tolerance to overlap or EGR, which bring their own benefits.

2.3.Methodology

The methodology adopted for the present work is as follows.
Flow through the intake manifold is simulated to study the in cylinder flow field during non-reacting conditions, which includes the following steps:

- Solid modeling of the intake manifold and cylinder geometry with valves.
- Mesh generation. Solution of the governing equations with appropriate boundary conditions.

2.4.Geometric Model

The configuration of the inlet port, valve, and cylinder is shown in Figure,
The port axis is offset set from the cylinder axis by 4.00 mm in the x direction and 21.87 mm in the y direction and it is elevated from the horizontal plane at an angle of 40 degrees.
The cylinder diameter is 93.65 mm, the inlet port diameter is 46.00 mm, and the valve diameter is 43.00 mm. Fig 3.2 shows the geometry of the intake port.
Fluid Properties Properties of the fluid (mixture of oil of turpentine and tetraline) are given in Table 1. These values are constant.

Table 1:

DENSITY	**894 kg/m3**
VISCOSITY	**0.00152875 kg/m-s**

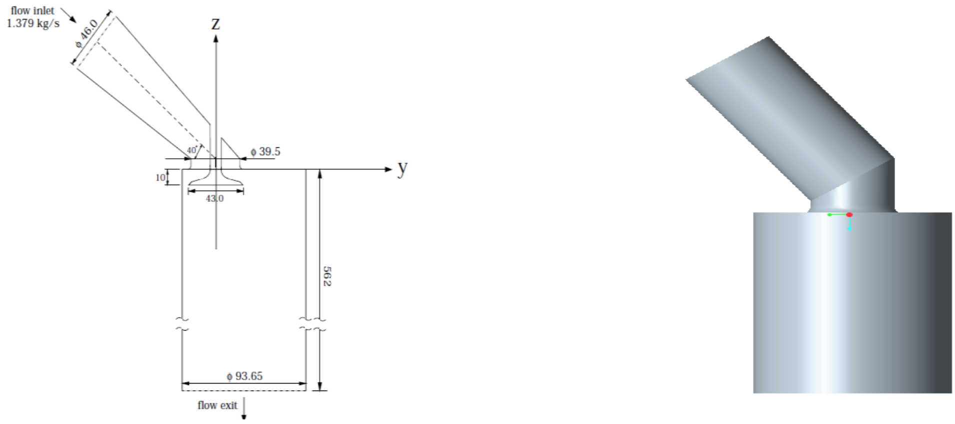

Figure 4.Geometric model of Engine

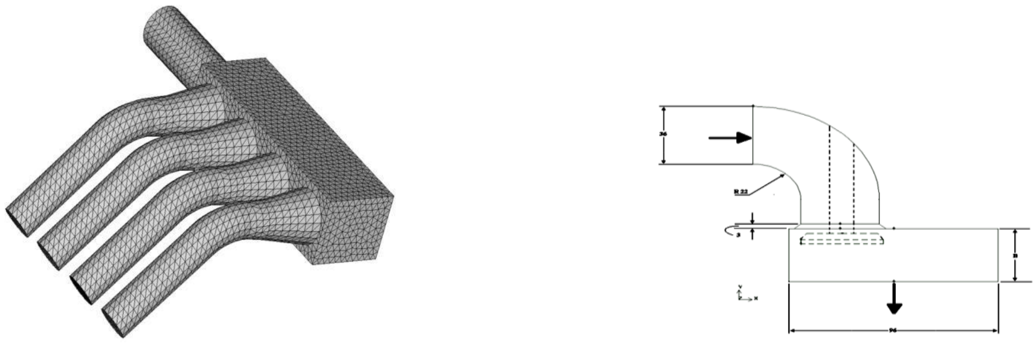

Figure 5.CFD Mesh model of intake manifold

2.5.Modified intake valve for computational investigation

For the study three new types intake valve has been modeled by making simple modification in the geometry of the Poppet intake valve in order to analysis the effect produce by the change in geometry on the level of intake swirl generation within the engine.

2.5.1. Poppet intake valve

A Poppet intake valve is used in the SI engine in which the computational analysis is performed. The dimensions of the Poppet intake valve are shown in the **Fig.1**

Figure 6.Intake valve Model-1

Intake valve Model-1 is the first modification of Poppet intake valve for the CFD analysis. The dimensions of Intake valve Model-1 are shown in the **Fig** above.

2.6.BOUNDARY CONDITIONS

The numerical formulation of the problem is incomplete without prescribing boundary conditions, which correspond to the specific physical model. The specification of mathematically correct boundary conditions that ensure the uniqueness of the solution, while being compatible with the physics at the boundaries, is not always straightforward. Before arriving at the boundary conditions at various boundaries, we have to first identify the solution/computational domain of the problem. The physical domain and computational domain usually differ. However, the computational domain largely depends on the geometry of physical domain. The computational domain boundary (truncated from the real boundary) along with appropriate boundary conditions should be chosen in such a way that there is negligible change in the results with further increase in its size.
Boundary conditions applied to the meshed model of intake manifold by using the post processor software FLUENT 6.3 which is shown in figure 3.5. The Reynolds number, Re = 24,970, is based on the diameter of the inlet port and the velocity at the inlet which is calculated from a mass flow rate of 1.379 kg/s [1]. The turbulence length scale is taken to be the diameter of the port (46 mm), and, in the absence of experimental measurements, the turbulence intensity is kept at the default value of 10%. The normal velocity at the inlet guided by the given mass flow rate is 0.928156 m/s. The default boundary conditions are used at the outlet (zero gauge pressure). The wall boundaries have a no-slip condition. The RNG k-" epsilon turbulence model with standard wall functions is employed in the calculation using the inlet boundary conditions.

Validation analysis of intake port

The flow field is initialized to the inflow conditions. The calculation in each of the three runs is performed as follows:

- The original grid is converged using first-order discretization.
- The second-order discretization scheme is enabled and convergence achieved.

The grid is refined by doing adaption in the region of high gradients of velocity, total pressure, and turbulent kinetic energy. Convergence is reached again with this refined grid.

3. RESULTS AND DISCUSSIONS

3.1. INLET MANIFOLD

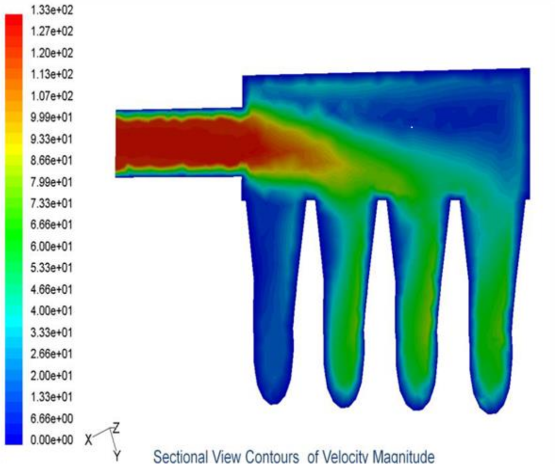

Figure 7.Velocity Magnitude contours

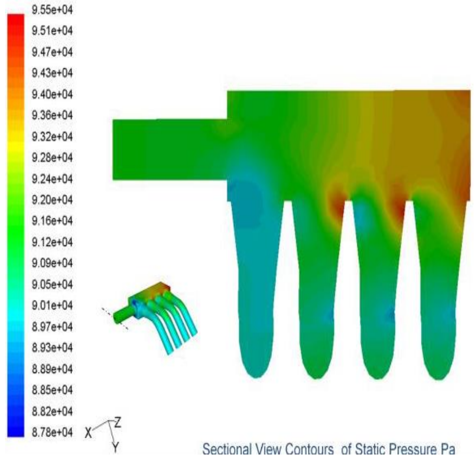

Figure 8.Static pressure contours

Figure 1 illustrates the velocity magnitude at one slice of the intake manifold. It was observed that the different colored regions indicate variation in the velocity magnitude of the system. The red colored region at the middle of the runners and the plenum wall portion indicates the high velocity. Due to the recirculation inside the plenum chamber at the slice of intake manifold the velocity magnitude was low as compared to other region.

Table 2:

Mass Flow Rate	**Kg/s**
Inlet	**0.430867**
Runner1	**-0.033505**
Runner2	**-0.124936**
Runner3	**-0.137032**
Runner4	**-0.135487**

Figure 2 illustrates the static pressure contour inside the intake manifold and at one slice along y - axis shows the pressure characteristics inside the manifold. The pressure was observed in plenum and runners. The different shades or red colored region on the walls, fresh air inlet, which determines the amount of flow inducted in the system. The green colored region at the plenum the pressure strength is less and the blue region at the runners, the pressure strength is very low in the system. The pressure is very high while entering into the manifold and at the runners the pressure is very low. Similarly total pressure contours is shown in Fig.

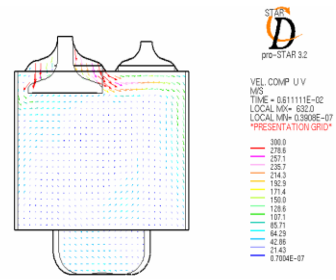
Figure 9.Helical

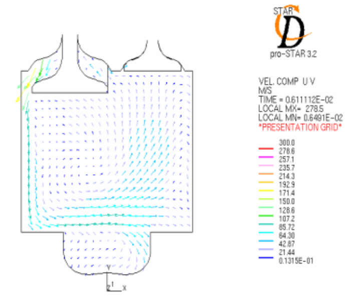
Figure 10.Spiral

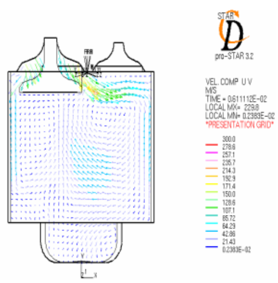
Figure 11.Helical-Spiral

3.2.INTAKE VALVE

Figures below represent the comparison between the experimental and predicted flow fields. Qualitatively, the predicted flow field is in good agreement with the measurements. All the characteristics of the flow described in the angle of the inlet jet, the vortices at the far right and the far left side of the cylinder, and the little vortex to the left of the valve) are correctly predicted by FLUENT. Velocity magnitude is shown in Fig 4.9, it shows velocity accelerates in inlet port valve section. Fig 3.10 shows the path lines colored by velocity magnitude.

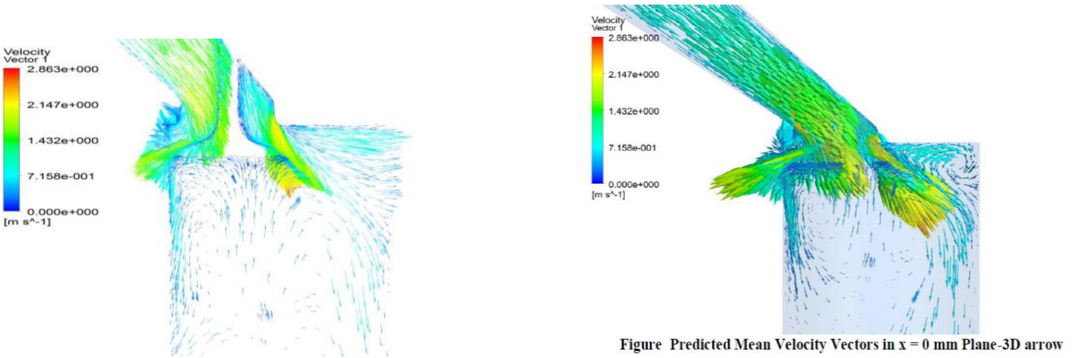

Figure 12.Predicted Mean Velocity Vectors in x = 0 mm Plane

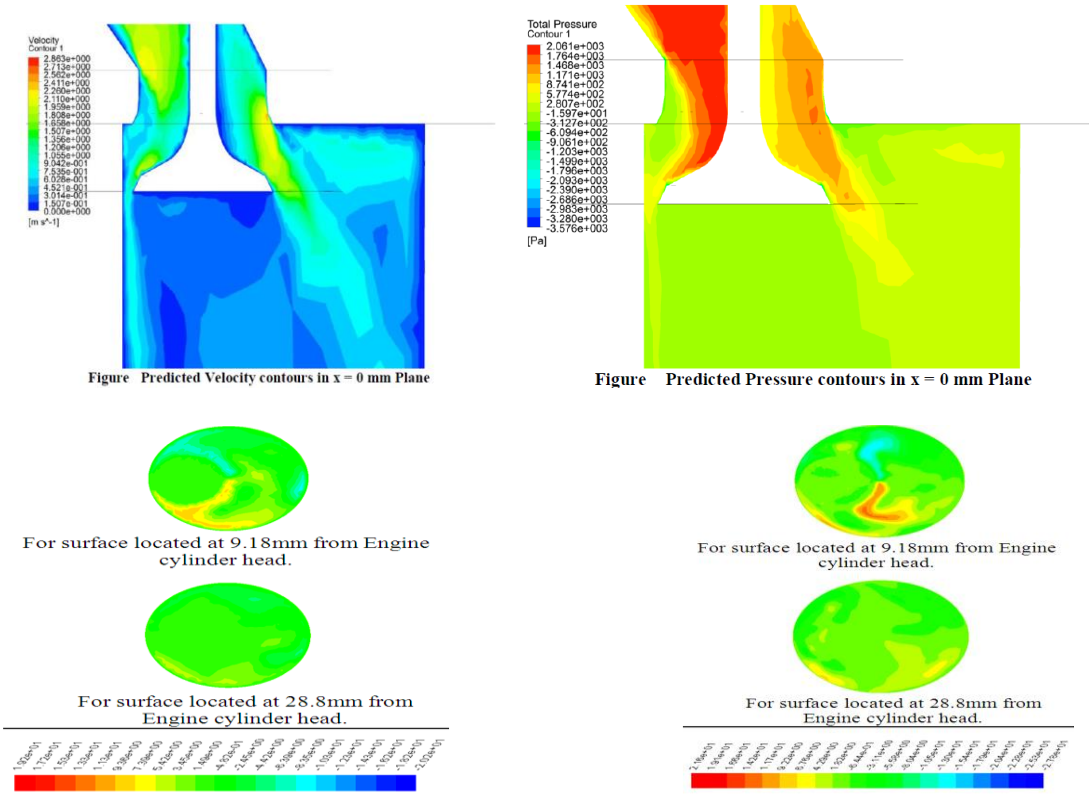

Figure 13.Tangential velocity contour

Poppet Intake Valve High Swirl Induction Valve

4. Conclusion

The overall flow field inside the intake manifold were examined and the summary of the result is as follows-

- Helical-spiral combined manifold creates higher swirl inside the cylinder than spiral manifold.
- Spiral and helical-spiral manifold creates higher swirl than Normal inlet manifold. In particular the helical-spiral combined manifold created higher swirl inside the cylinder as it is a combination of both spiral and helical & so the twisting action will be more.

Predicted results inlet port compares favorably with experiments for velocity magnitude.
From this study the following point can be concluded Poppet intake valve is the better design in term of intake swirl generation within the engine out of new types of intake valve has been modeled by making simple modification in the geometry of the Poppet intake valve. By incorporated two curve blades are on its neck the poppet intake valve the intensity intake swirl generation within the engine increased significantly.
From the computational analysis it is seen that in all the cases the surface which is closer to the valve shows higher tangential velocity at various location compared to the surface at which is at higher distance from the intake valve i.e. the intensity of swirl decreases along the stroke length of the engine cylinder.

Acknowledgement

We express our sincere gratitude and admiration for **Mr. Shrikant Madiwale** for his extraordinary contribution towards this project and for guiding us throughout our work with finite element modeling (FEM).
We admire and respect **Mr. ARUN MAHAJAN** for encouraging us to work on this topic.
Also We would like to thank **V.GANESAN** for the key inputs that we get from his technical paper which have been instrumental for our project.

References

- Benny Paul1, V. Ganesan, Flow field development in a direct injection diesel engine with different manifolds, International Journal of Engineering, Science and Technology V2, 2010.

- W.H. Kurniawan, S. Abdullah., Turbulence and Heat Transfer Analysis of Intake and Compression Stroke in Automotive 4-stroke Direct Injection Engine, Algerian Journal of Applied Fluid Mechanics, V1, 2007.

- David Rathnaraj and Prof. T. Michael. N. Kumar, "Studies on variable swirl intake system for DI diesel engine using CFD", Thermal Science: V12 pp. 25-32 2008.

- S.C. Kale and Prof. V. Ganesan, "A study of steady flow through a SI engine intake system using CFD”, Journal of the Institution of Engineers (India). V86, pp. 61-65.1996.
- Internal Combustion Engine, Ramalingam