Numerical Simulation of Steady Flow through Engine Intake System Using CFD

Rajesh Holkar[1], Yogesh N. Sule-Patil[2], Somnath M.Pise[3], Yogesh A. Godase[4], Vishal Satish Jagadale[5]

(Assistant professors, department of mechanical engineering,smsmpitr"s college of engineering, akluj,india)

Abstract: *The objective of present study is to predict and analyze the flow through intake manifold and inlet port system using Computational Fluid Dynamics (CFD) and to validate the prediction by experimental data. Three-dimensional model of air intake system was analyzed by using the commercially available FLUENT software. The mesh was generated using the tetrahedral hybrid scheme which includes primarily of tetrahedral mesh elements but may include hexahedral, pyramidal and wedge elements using HyperMesh. The pressure boundary conditions were used to define the fluid pressure at the inlet and outlet of Air Intake System. In the present study, the CFD code was validated by the experimental work using intake port analysis. The CFD plots give informative pictures of the flow field, which will help the designer to understand the effect of various components of Air Intake System. The predicted airflow rate shows good agreement with the experimental results. 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. This in effect will reduce the number of experiments to be carried out for arriving at final optimized system.*

Keywords: *Engine, intake manifold, vanes, intake port model*

I.   Introduction

To Intake Manifold

The engine cycle of typical internal combustion engines consist of four consecutive processes as intake, compression, expansion (including combustion) and exhaust. Of these four processes, 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. Heat transfer, evaporation, mixing and combustion rates all increase as engine speed increases. This increases the time rate of fuel evaporation, the mixing of the fuel vapor and air as well as combustion process. 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. Achieving this by means of experimental methods would cost time and money. There is a need for 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. An outlook and internal details of typical diesel engine with intake manifold are shown in Fig 1.1 and 1.2 respectively.

Figure 1.1 Outlook of a diesel engine
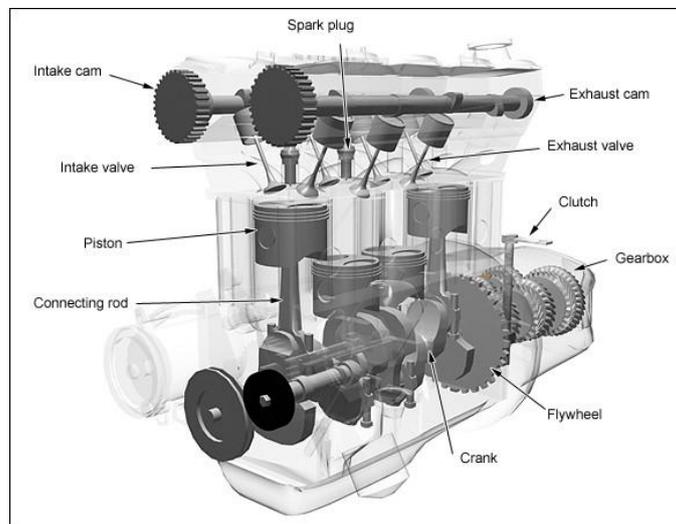
Figure 1.2 Sectional view of Diesel engine with intake manifold

1.2 Engine Intake System

The basic function of engine intake systems is to provide the engine with a fresh air-fuel mixture every cycle for combustion to take place. Different engine operation requirements demand individualised intake systems such as the racing engines and passenger cars (Everyday-use vehicles). For racing cars require maximum volumetric efficiency for increased power and torque, but this is not desirable as far as economy is concerned and for passenger cars hardly requires top end power, thus economy and driveability at lower speeds are more important in this instance. It is therefore important that intake systems are designed to suit the purpose for which the engine is intended.

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 (NO_x). 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.

II.   Simulation Set Up And Data   Input

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: Petrol Engine, Diesel Engine and Gas engine.

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.

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.
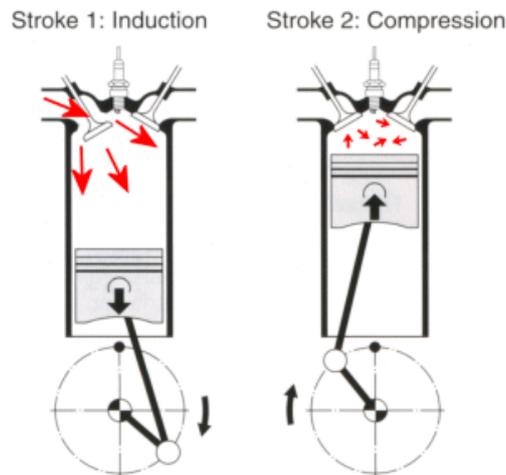
Figure : Low swirl engine: Inlet port designed for high flow, no strong direction to air jet. Very little air motion in the cylinder at the point of ignition.

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.
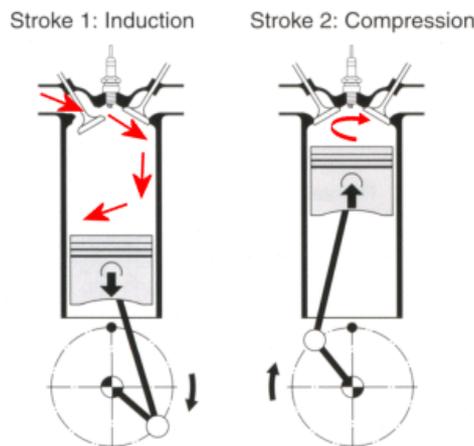
Figure High swirl engine: Inlet port designed to produce very directional air jet. Strong swirling motion set up in the cylinder From the figure stable, circulating flow pattern in a diesel engine designated as swirl motion, with the cylinder axis as the axis of rotation. The flow enters tangentially through the intake ports.
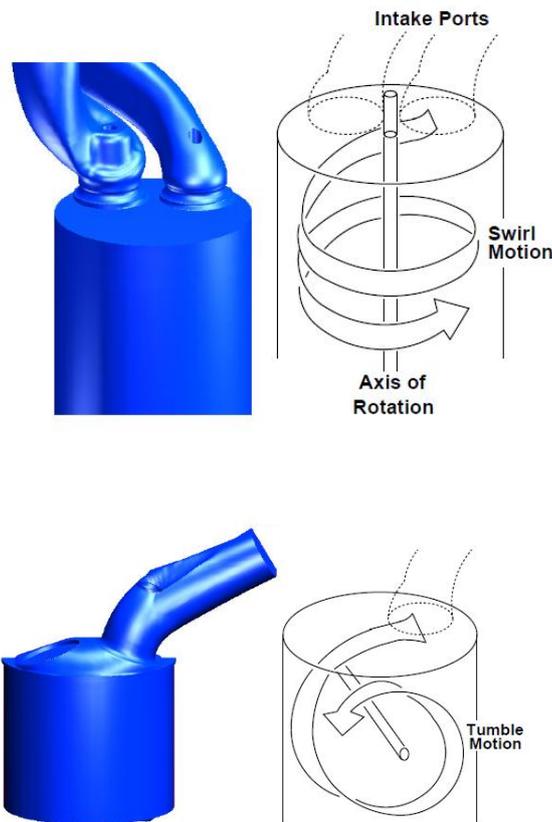
Figure Swirl and tumble motion of an engine

      The swirl strength and mass flow rate are main factors to improve the combustion efficiency. The swirl has a role to improve the mixing effect of fuel and air in the cylinder chamber. Tumble motion was measure on the vertical symmetric plane of the combustion chamber. Swirl motion was measured on a plane parallel to the piston crown with one of the intake ports blocked. Tumble vortex is produced in the early stage of the compression stroke and distorted in the late stage of the stroke.

The axis of motion moves as the cylinder expands and stays halfway between the top cylinder wall and the piston head at the bottom.
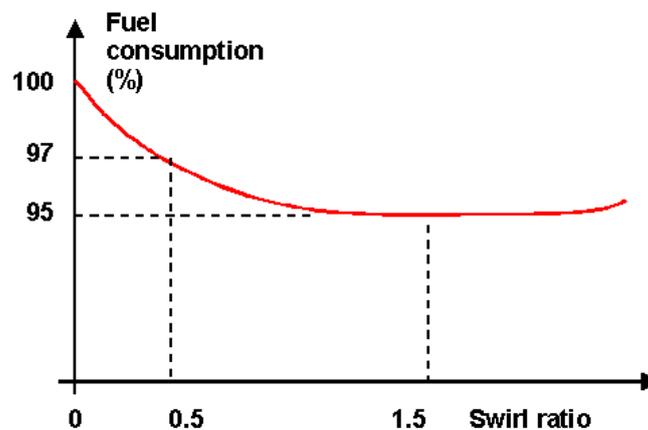
Fuel consumption Vs Swirl ratio

      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

III.   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 3.1. These values are constant.

Table 3.1 Fluid Properties

Density,	894 kg/m^3
Viscosity	0.00152875 kg/m-s
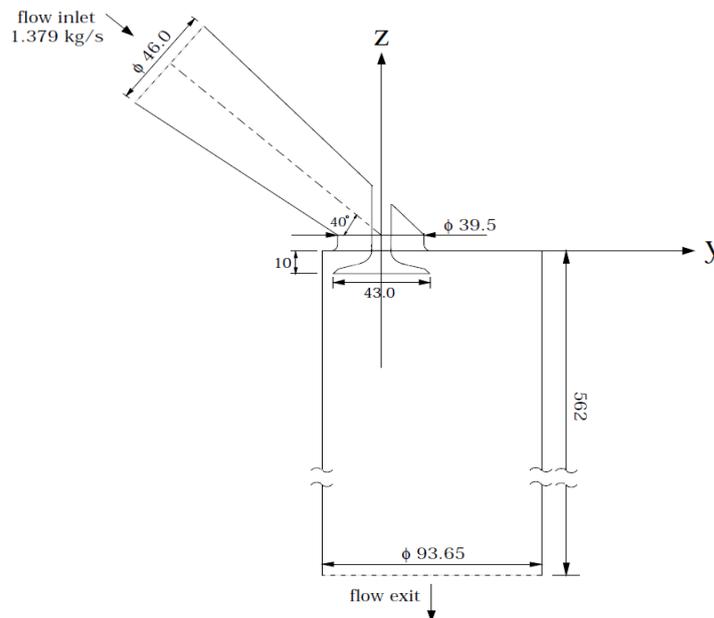
Figure Problem Description (all dimensions in millimetres)
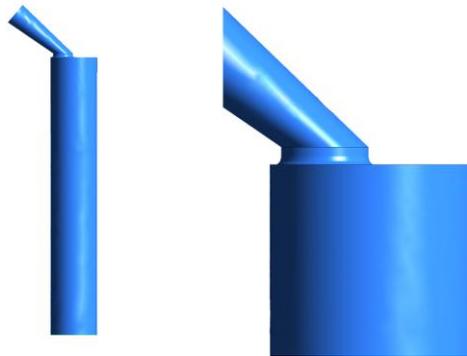
Figure   Geometric model of Intake port

IV.   Hypermesh

In this project Hyper Mesh is used for pre-processing of the intake port and manifold. HyperMesh 8.0-SR1 is a high-performance pre-processor for computational fluid dynamics (CFD) and injection molding simulations. Hyper Mesh enables users to generate quality grids and meshes in a highly interactive and visual environment. HypeMesh supports most CAD geometry formats and exports CFD meshes in native formats, including FLUENT,STAR-CD, CFD++ and MoldFlow. Hyper Mesh's state-of-the-art meshing technology allows users to develop high-quality meshes to guarantee fast convergence rates. Automated mixed-type mesh generation minimizes meshing time, while batch meshing enables large-scale meshing jobs with no model clean-up and minimal user input. Hyper Morph, a module of HyperMesh, enables users to define geometrical changes or morphing shapes to study the sensitivity of CFD models to changes in the flow domain, as well as to perform manual or automatic CFD optimizations. HyperMesh 8.0-SR1 incorporates a variety of tools for seamless integration into any existing engineering process, with a streamlined meshing process that leads to shorter turnaround times.

V.   Discretisation Of Intake Manifold Geometric Model
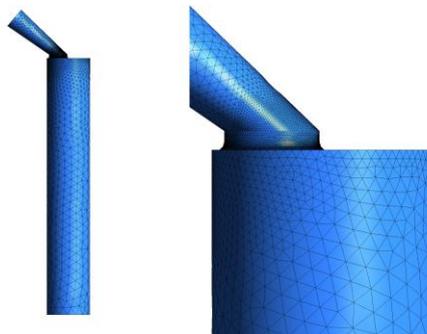
Figure   Tetrahedral meshed of intake port

The most commonly used approach includes importing CAD data from a variety of CAD formats; performing a combination of manual and automatic geometry clean-up with automatic or user-defined criteria; and surface meshing with default or user-defined element types, sizes, biasing, quality-index, etc.

Another approach includes fully automatic, 3D CFD meshing with boundary-layer meshes consisting of hexahedral and/or wedge elements, an arbitrary number of layers, thicknesses or growth rates, and a tetrahedral core-volume meshing algorithm. This tetrahedral algorithm has several meshing options, including size, growth rate and element-quality criteria. Hyper Mesh's CFD meshing capabilities span a very wide range of target applications, including internal and external flows, conjugate heat transfer, species and chemical reactions, combustion, etc. A very large class of CFD applications can benefit from the high-quality simulation meshes generated by Altair HyperMesh.

The mesh generated for the base model used for the analysis. The tetrahedral fine and coarser mesh was generated by using HYPERMESH V9.0 which is shown in figure 3.4. The accuracy of the results depends on the size of the mesh. The results will be accurate if the mesh is finer but the time required for computation is very high; if the coarser mesh is used then the results will be inaccurate so to balance between accurate results

and computational time which are critical regions such as the inlet port are meshed finer and remaining regions were meshed coarser.

The inlet port region is important because of the happening very rapidly hence this region was provided with much finer mesh. The no of grids used in this simulation is 315410 tetrahedral cells. The same grid model was used in all the simulation. 176,461 cells, shown in Figure 4.4 are used. The concentration of cells in the vicinity of the valve is due to the complexity of the flow in the region.

VI.   Boundary Conditions For Manifold
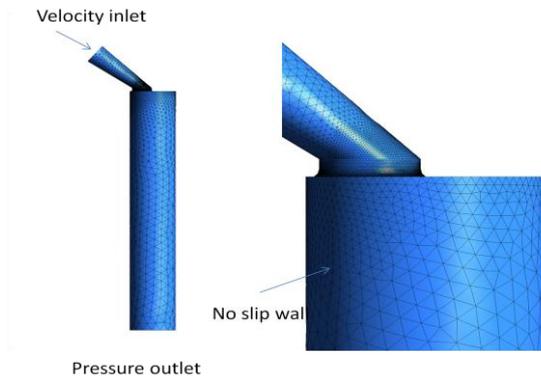
Figure   Meshed model with boundary conditions

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.

VII.   Results And Discussions

Figures 3.6 to 3.8 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.
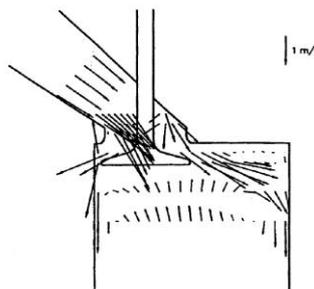
Figure Measured Mean Velocity Vectors in x = 0 mm Plane
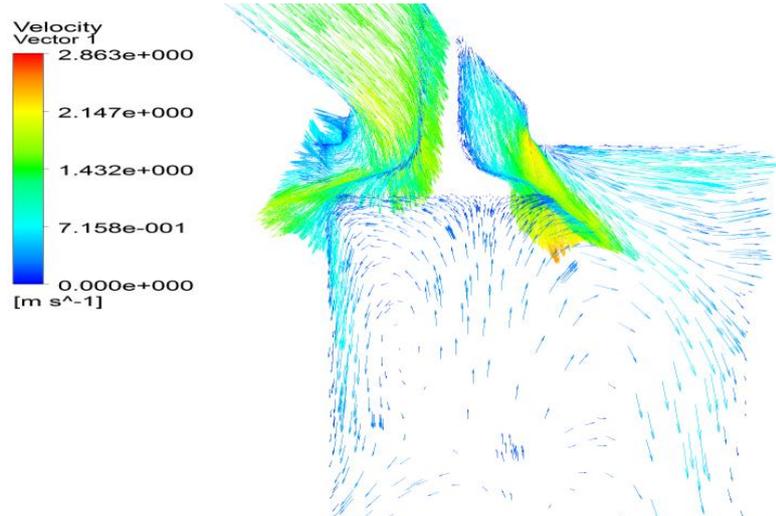
Figure Predicted Mean Velocity Vectors in x = 0 mm Plane
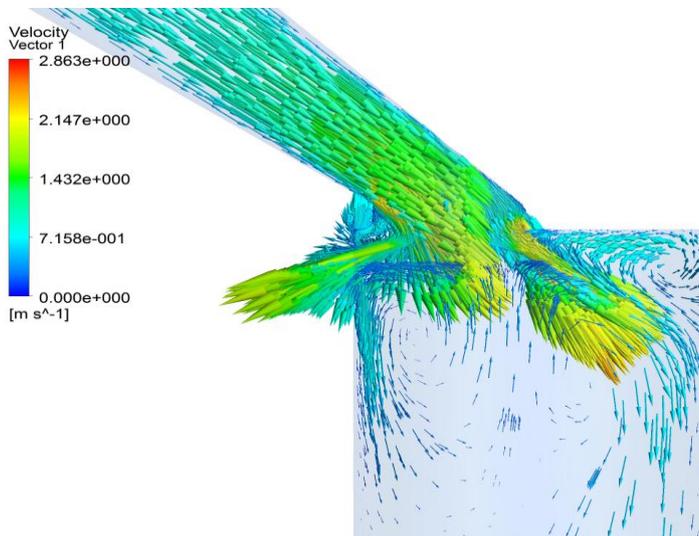
Figure Predicted Mean Velocity Vectors in x = 0 mm Plane-3D arrow
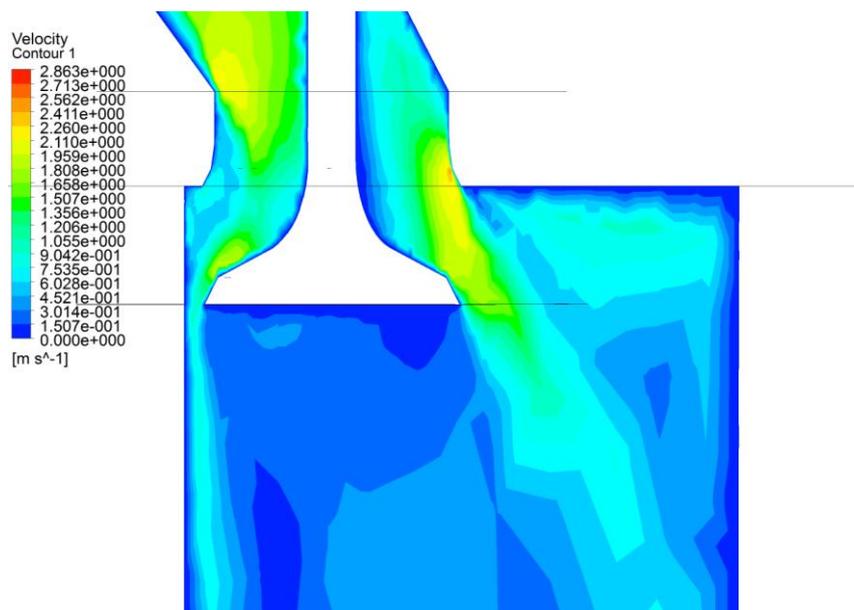
Figure   Predicted Velocity contours in x = 0 mm Plane
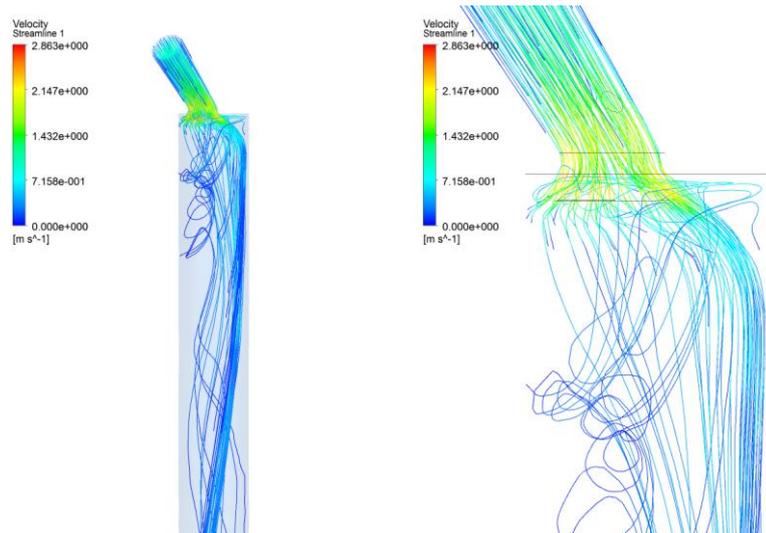
Figure Path lines colored by velocity magnitude in x = 0 mm Plane
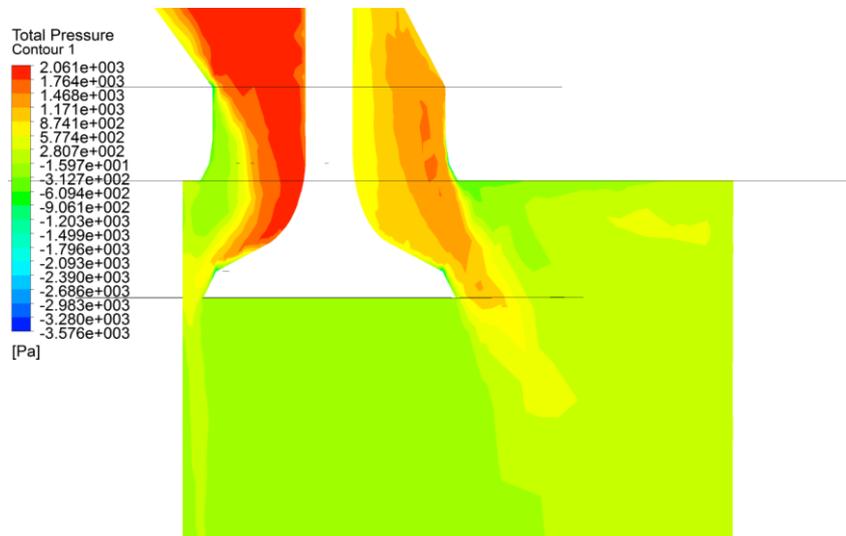
Figure   Predicted Pressure contours in x = 0 mm Plane
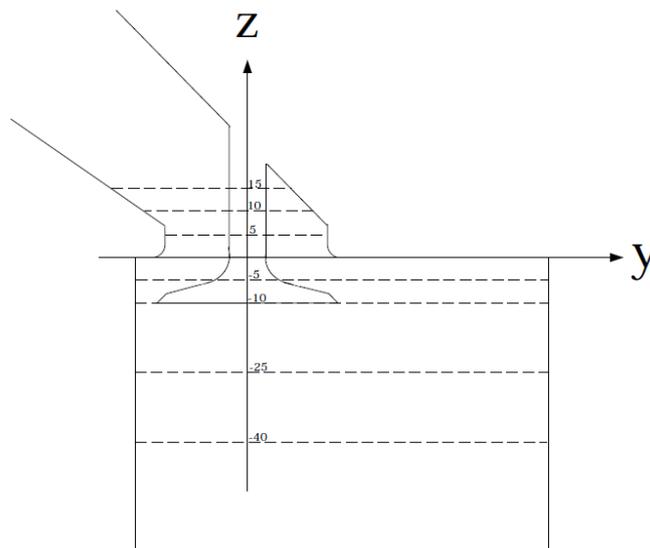
Figure Lines on which z-Velocity Measurements were taken
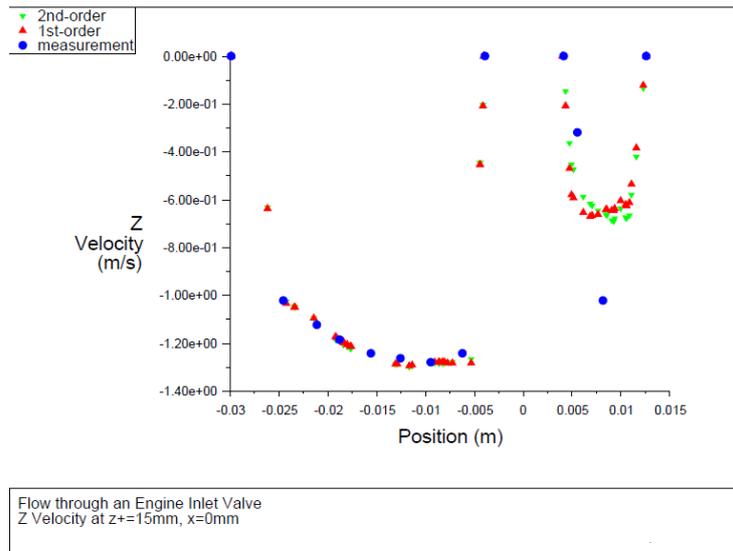
Figure Z-Velocity Components at z = +15 mm, x = 0 mm
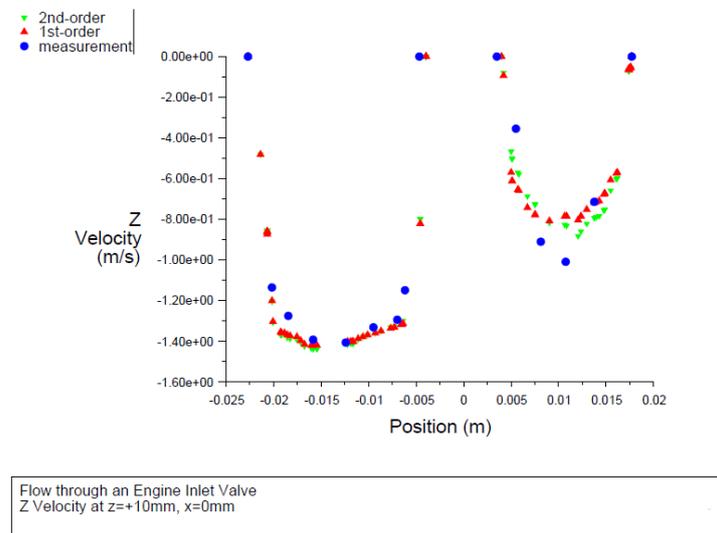
Figure   Z-Velocity Components at z = +10 mm, x = 0 mm
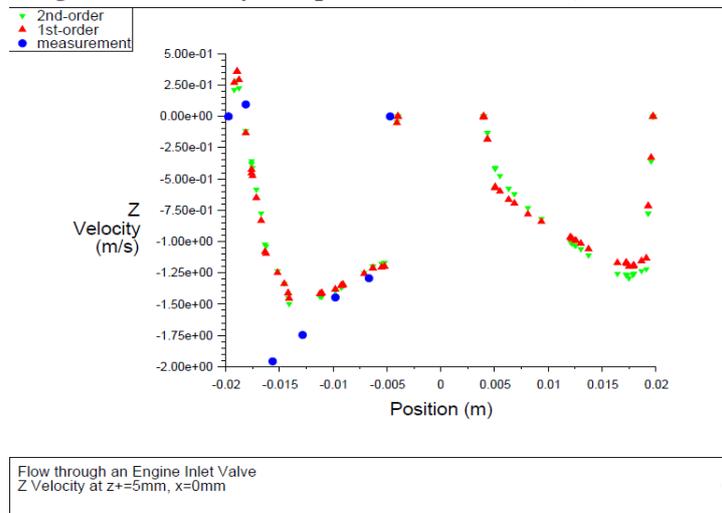
Figure Z-Velocity Components at z = +5 mm, x = 0 mm
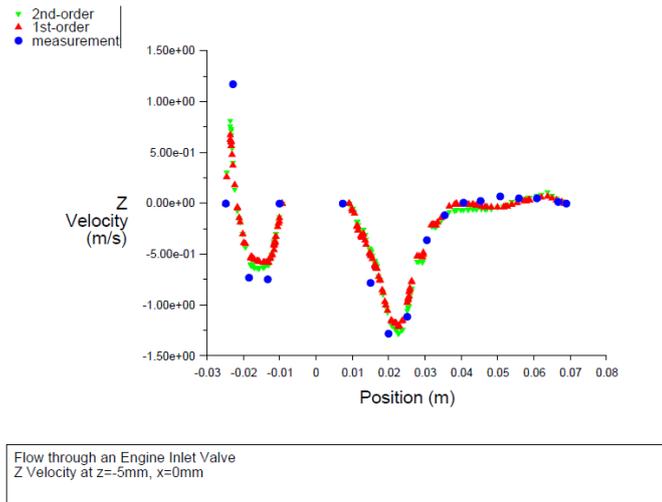
Figure Z-Velocity Components at z = -5 mm, x = 0 mm
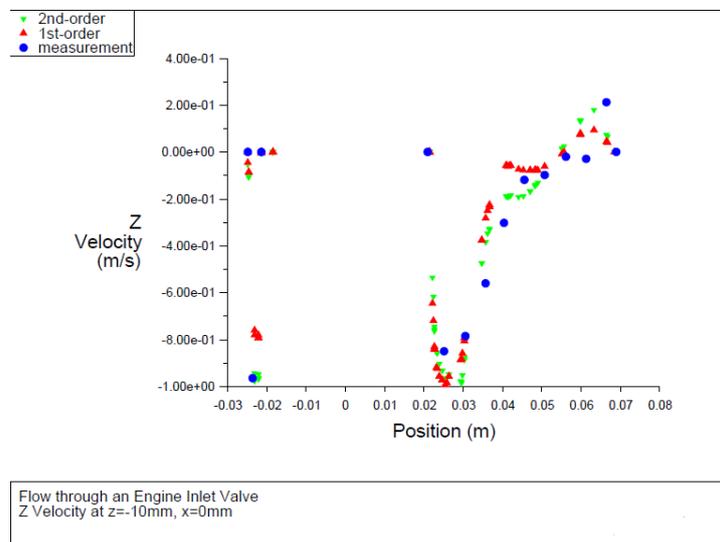
Figure   Z-Velocity Components at z = -10 mm, x = 0 mm
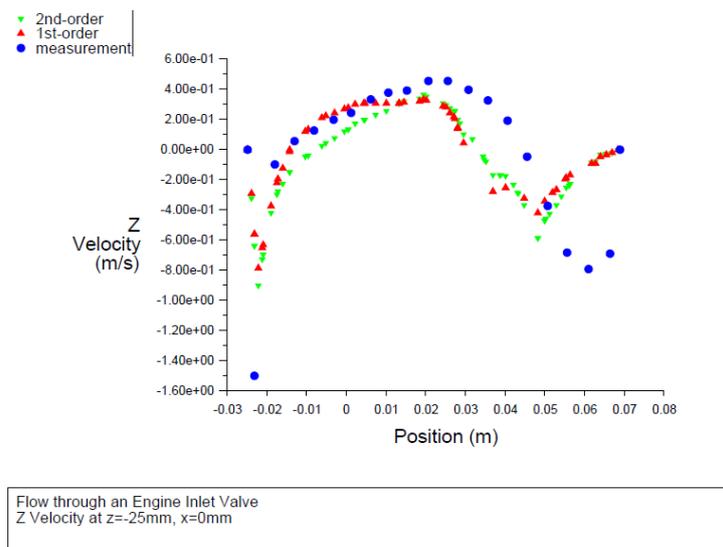
Figure Z-Velocity Components at z = -25 mm, x = 0 mm
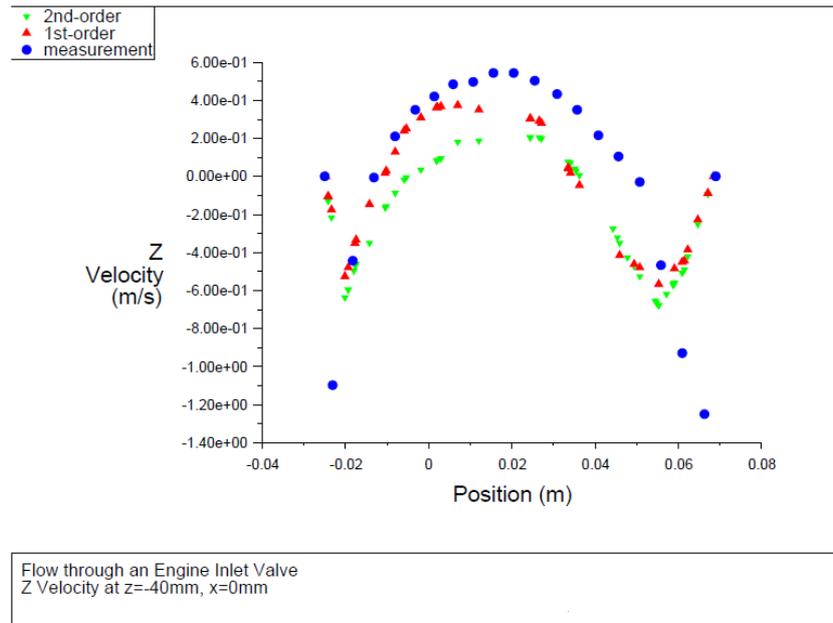
Figure   Z-Velocity Components at z = -40 mm, x = 0 mm

VIII.   Modelling of Computational Domain

The geometry under consideration for the current study is shown in Figure 4.1. It consists of the inlet duct, plenum and various runners. This domain represents the intake regions of four cylinders. The upstream effects on the flow entering the ports are to be reasonably captured leading to realistic in-port flow structure prediction.The flow pattern in the intake region is insensitive to flow unsteadiness and valve operation and thus could be predicted through steady flow test and computational simulation with reasonable accuracy.
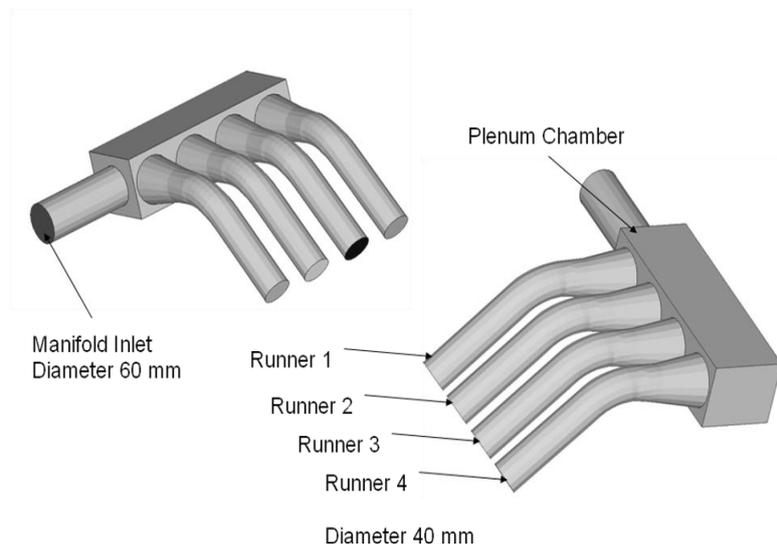
Figure Geometric Model of Intake Manifold

The geometry has been generated in CATIA software and imported to HyperMesh for the surface mesh generation and then it has been transferred to Fluent for CFD analysis generation. A Tetrahedral mesh has been generated as shown in Fig. The original mesh size was between 26000 and 137000 cells. Cells were refined in the region of critical valve and port region in anticipation of the high velocity and pressure gradient. Grid independent tests were conducted to ascertain the adequacy of the grid and it was found that the grid distribution used was quite adequate.
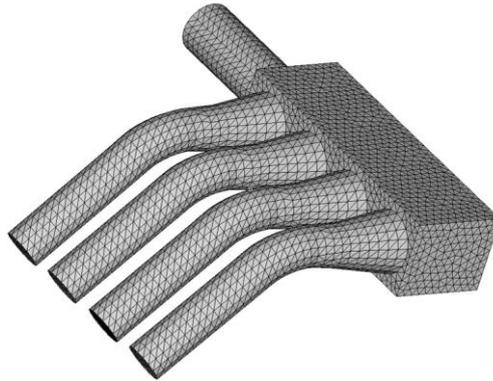
Figure CFD Mesh model of intake manifold

Based on Experimental measurements in providing the necessary boundary conditions to the prediction. At inlet of the plenum chamber a manifold pressure and at the exit of valve cylinder pressure were given corresponding to the valve lift as boundary conditions. A turbulence intensity of 5%, and 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 k 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 (k) and another one for the dissipation rate (ε). Near wall treatment is handled through generalized wall functions. In this study high Reynolds number k-ε model has been used. This model is well established and the most widely validated turbulence model.
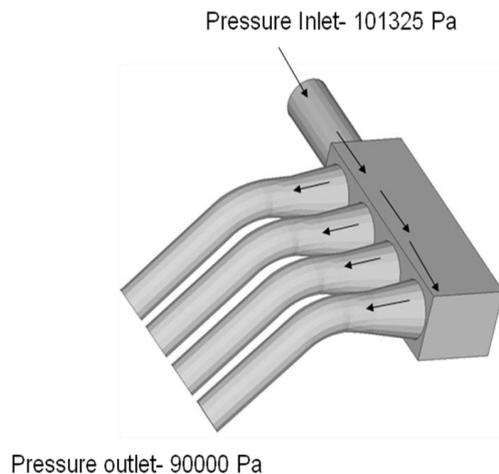
Figure Boundary conditions of intake manifold

IX. Results And Discussions

Figure 4.4 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

Sectional View Contours of Velocity Magnitude
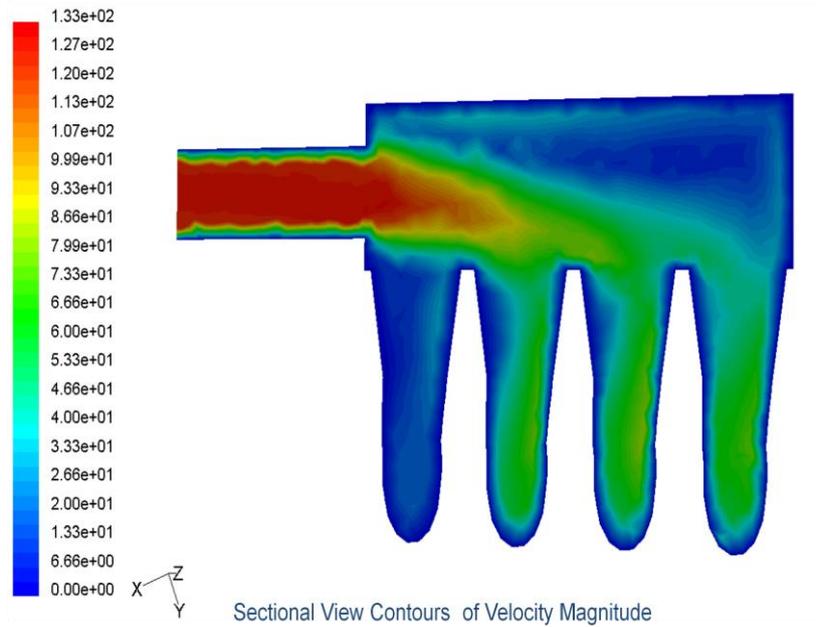
Figure contours of velocity Magnitude
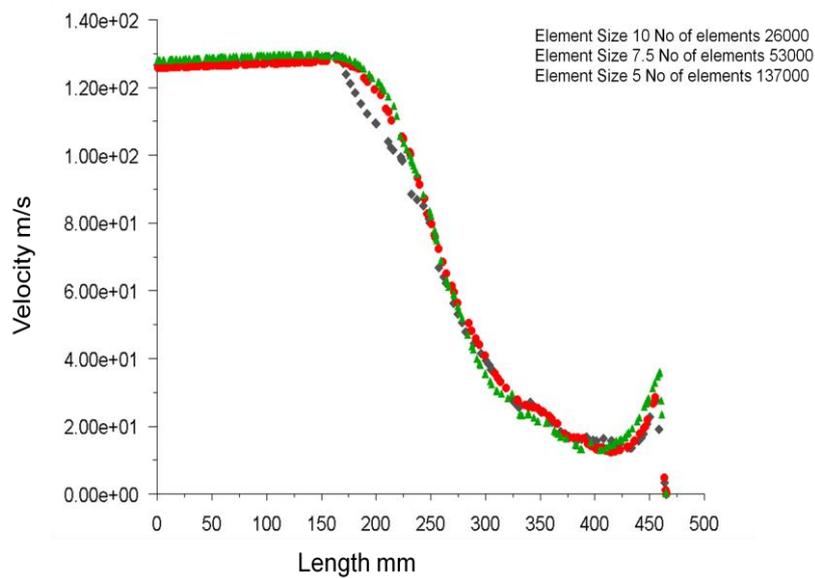
Figure Grid convergence test velocity

Grid independence study was carried out for the intake manifold. Grid independent tests were conducted to ascertain the adequacy of the grid and it was found that the grid distribution used was quite adequate. Tetrahedral grid type was used for the further simulation, since the results are more accurate. Fig 4.5 illustrates the grid independence study effects the velocity magnitude in the centre of the plenum compared with 3 different grids.

Mass flow Rate	kg/s
Inlet	0.430867
Runner4	-0.033505
Runner4	-0.124936
Runner4	-0.137032
Runner4	-0.135487

Figure 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.
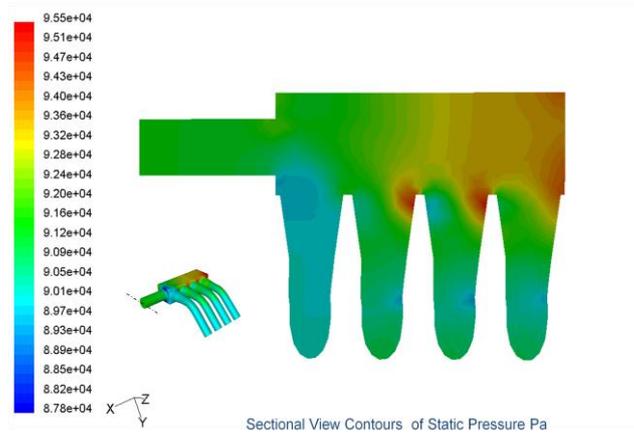
Figure   Static pressure contours
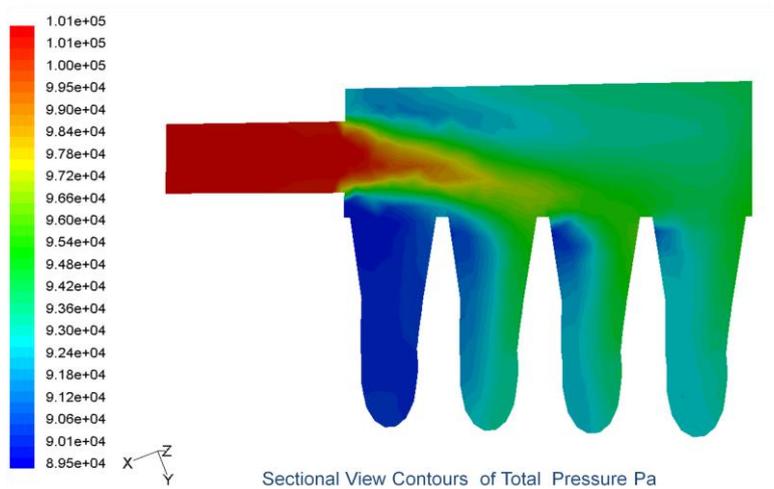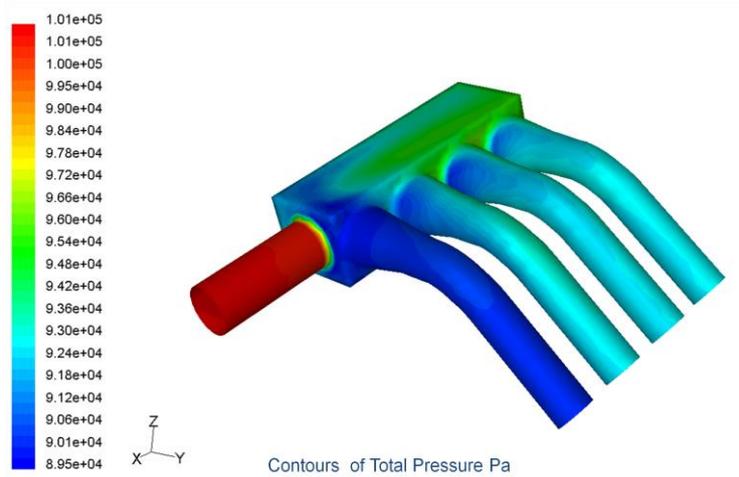
Figure Total pressure contours

X.   Conclusion

CFD contributes to the understanding of three-dimensional flow and also provides detailed information, where measurements are rather difficult to make.

- Multi-dimensional modelling of the air intake region including the inlet manifold, plenum, port and valve is the key for predicting pressure loss influenced by the physical configuration.
- The flow is highly three-dimensional. It is strongly dependent on the valve lift except upstream of the port bend.
- Losses in the valve clearance are higher. At higher valve lift flow separation is critical.

Predicted results inlet port compares favourably with experiments for velocity magnitude

References

[1].   Chen, A., Lee, K.C., Yianneskis, M., and Ganti, G., Velocity Charac- teristics of Steady Flow Through a Straight Generic Inlet Port, Interna- tional Journal for Numerical Methods in Fluids, 21:571{590, 199

[2].   J. David Rathnaraj and Prof. T. Michael. N. Kumar, "Studies on variable swirl intake system for DI diesel engine using CFD", ISSN 0973-4562 volume 2, number 3 (2007)

[3].   F. Millo, C.V. Ferraro and M. Gianoglio Bernardi, "Experimental and Computational Analysis of Different EGR Systems for a common Rail Passenger Car diesel Engine", SAE, 2009-01-0672

[4].   John B Heywood, "Internal Combustion Engine Fundamentals" McGraw-Hill Internal Editions, 1998

[5].   B. Murali Krishna and Mallikarjuna J. M, "Characteristics of flow through the intake valve of a single cylinder engine using practical image velocimetry", Indian institute of technology madras

[6].   S.C. Kale and Prof. V. Ganesan, "A study of steady flow through a SI engine intake system using CFD", ISSN 1546-9239

[7].   Soonseong Hong, Byungkeunoh and Yongtae Kim, "Optimization of intake manifold design using design for six sigma, SAE, 2007-01-3534

[8].   Michael J. Rath, "Resolving EGR distribution and mixing", SAE, 2002-01-2882

[9].   Colin P. Garner, "Development of a Validated CFD Process for the Analysis of Inlet Manifold Flows with EGR" , SAE 2002-01-0071

[10].   Mike Dong, Grant Chen, Min Xu and Chao Daniels, "A Preliminary CFD Investigation of In-Cylinder Stratified EGR for Spark Ignition Engines", SAE-2002-01-1734

[11].   T.R. Fuchs and C.J. Rutland, "Intake flow effects on combustion and emissions in a diesel engine", 980508

[12].   Semin, Rosli Abu Bakar and Abdul Rahim Ismail, "Computation visualization and simulation of diesel engines valve lift performance using CFD", American Journal of applied science 5 (5) : 532-539, 2008 ISSN 1546-9239

[13].   Chen. A, Lee, K.C. Yianneskis. M and Ganti.G, "Velocity characteristics of steady flow through a straight generic inlet port", International Journal for numerical method in fluids, 21:571{590, 1995}

[14].   W.H. Kurniawan, S. Abdullah and A. Shamuses, "Turbulence and heat transfer analysis of intake and compression stroke in Automotive four stroke direct injection engine", Algerian Journal of Applied fluid mechanics, Volume 1, 2007

[15].   Rolf Egnell, "The influence of EGR on heat release rate and no formation in a DI diesel engine", SAE, 2000-01-1807