International Journal For Technological Research In Engineering (IJTRE)
Volume 2, Issue 7, March-2015
ISSN: 2347-4718

Volumetric Efficiency Appraisal and Cold Flow Study of Formulae Student Vehicle

Kartik Panchal[1*], Sumit Kanchan[2]
[1]Project Engineer, Mechanical Engineering department, Indian Institute of Technology, Gandhinagar, INDIA.
[2]Assistant Professor, Mechanical Engineering department, Lovely Professional University, Jalandhar, INDIA.

Abstract- **This study presents volumetric efficiency (η_v) as a measure of breathing capacity of internal combustion engine (ICE). The combustion characteristic is highly dependent on the η_v of ICE. This study supports redesigning and enhancing the power output in the presence of restrictor fitted just downstream of throttle for the student designed Formula type vehicle. Considering the constraints laid by the Formula SAE, the design has been implemented in student designed formula vehicle. The study has been performed on single cylinder 499cc Spark Ignited (SI) engine. In this work, the new intake geometry has been examined in order to estimate the η_v. It also shows the effect of plenum volume on the in-cylinder flow field using computational fluid dynamic (CFD) solver. The computation has been carried out with the compressible Navier-stokes and Energy equations in combination with the standard k-ε model. The result discusses the in-cylinder flow field variation with different engine speeds. Also the plenum effect on the breathing capacity of engine during initial 80° crank rotation in its 240° suction path, at different engine speeds has been carried out. It was observed that the volumetric efficiency calculated with the Reynolds Average Navier Stokes (RANS) approach of new intake design comes out in range of 85 to 90%.**

Keywords: Internal Combustion Engine, Plenum, Computational Fluid Dynamics (CFD), Reynolds Average Navier Stokes (RANS), Volumetric Efficiency.

1. INTRODUCTION

Intake manifold is an engine component which supplies the air charge to the combustion chamber and is also called the breathing system of the engine. Thus, the performance of the internal combustion engine depends on the breathing system of the engine. Higher the mass flow through intake more the efficiency results. The prime function of the intake manifold is to distribute the air charge evenly in each cycle and to each cylinder. The major parts of the intake manifold are: intake runner, plenum, throttle valve and air filter. The present design of intake manifold is according the standards of Supra SAE. [SAE rulebook, 2014]. Primary factor in optimizing is the power which mainly depends on volumetric efficiency of the engine. This study is supported out to analyse the effect of newly design intake manifold for formula student vehicle. The present study utilizes the length of the intake runner calculated as 14.32 inches according to Eqn. 1 along with the plenum volume is taken as ≈850cc. Also, the restrictor has been designed with the constraint of Supra SAE rules.

$$L = \left(\frac{ECD \times 0.25 \times V \times 2}{N \times RV} \right) - \frac{D}{2} \tag{1}$$

$$ECD = 720 - (\text{adv.duration} - 30) \tag{2}$$

Where, ECD is Effective Cam Duration, D is runner diameter and V is speed of sound, RV is reflective value which has been taken as 4 in calculations.

2. LITERATURE REVIEW

Literature reveals that to reduce the tendency of reverse flow through intake valve, studies [McFarland et al, 1979] showed that steps or sudden enlargement of runner can disperse the vigour of a pressure pulse travelling up a runner as well as guiding it back to the inlet port. Studies concerned [Takii et al, 1989 and 1991] outlines the individual intake passage for each pair of cylinders in V type configuration gives higher efficiency. Active runner mechanism varies effective length of runner maximises tuning of the engine over the wide range of engine speed [Narayanaswamy et al, 1991]. Variable plenum upstream of the primary runners are beneficial for the higher range of engine speed, which can operate as a single plenum or multiple plenums depending upon the engine operating conditions. [Verkleeren et al, 1998].

3. ADOPTED METHODOLOGY

3.1 Computational modelling

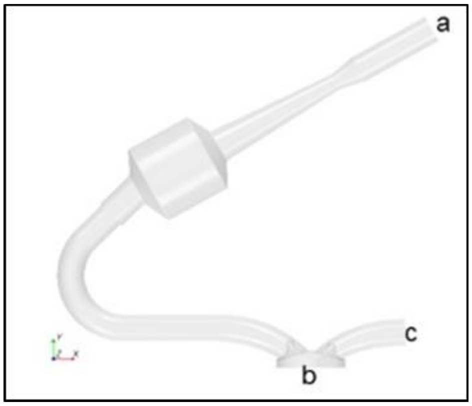
The mathematical models used for the computational study of the intake manifold using CFD (Computational Fluid Dynamics) are primarily the bunch of conservation laws for mass, momentum and energy for compressible flows. The flow is assumed to be turbulent and ruled by the Reynolds Averaged Navier-Stokes (RANS) equations which are as follows:

$$\rho \iint_A (\vec{V} - \vec{V}_b) \cdot d\vec{A} = 0 \tag{3}$$

$$\rho \frac{\partial}{\partial t} \iiint_v \vec{V}.dv + \iint_A \vec{V}(\vec{V} - \vec{V}_b).d\vec{A} = -\iint_A p \cdot d\vec{A} + \iint_A \vec{\vec{\tau}} \cdot d\vec{A} + \rho \iiint_V \vec{f} dV \tag{4}$$

$$\frac{d}{dt} \int_V \rho E dV + \oint_A [\rho H(\vec{V} - \vec{V}_b) + V_b p].da = -\oint_A \dot{q}''.da + \oint_A T.Vda + \int f.VdV + \int s_u dV \tag{5}$$

Where V_b term is the grid velocity for moving grids, and is non-zero for the present study. $\vec{\vec{\tau}}$ is the stress tensor and for the RANS model it consist of both laminar shear stresses and modelled turbulent shear stresses based on appropriate turbulence models. The convective flux in the momentum Eqn. 4 is discretized using a second order scheme and the time integration is carried out using global time stepping all within the framework of the STAR-CCM+ CFD software [CFD, user manual, 2013]. In the energy equation $E = e + \frac{u^2}{2}$. Where "e" denotes the internal energy. For an ideal gas at constant specific heat ratio the pressure is known by the perfect gas equation as: $p = \rho RT$. Turbulence is modelled by the standard k-ε [Launder et al, 1974] turbulence model coupled with compressible flow. Sutherland's law describes the viscosity change with the function of temperature for ideal gas. This governs by $\frac{\mu}{\mu_0} = \left(\frac{T}{T_0}\right)^{\frac{3}{2}} \left(\frac{T_0 + S}{T + S}\right)$, Where, T_0 and μ_0 are the reference temperature and viscosity, respectively, and S is the Sutherland constant. The fluid domain consider for this study is cavity of cylinder block of Royal Enfield 500cc engine and the internal cavity of newly designed intake system shown in figure 1 . The inflow boundary is given as the pressure outlet open to atmosphere pressure, because the flow velocity is affected by the suction pressure. The reference pressure is set to zero gauge pressure. On the exhaust port the same condition has been given as inflow. The moving boundary considered as the piston head is displaced by, $\vec{V}_{piston} = -r\omega\left(\sin\theta + \frac{\sin 2\theta}{2n}\right)$. where, r is crank radios, θ is crank angle and n is l/r ratio. The intake and exhaust valve also considered as the moving boundary according to, $\vec{V}_{Valve} = \frac{dS}{t}$. The rest of the boundary is considered as the adiabatic wall as there is no transfer of heat to outside.



a = Open to Atmosphere

b = Moving boundary

c = Open to Atmosphere

Valves = Moving boundary

Figure 1: Computational domain and boundary conditions for intake manifold

Grid was generated in the domain using Trimmer unstructured meshes with the finite volume method. The prism layers have been generated close the wall in order to imprison the boundary layer accurately. The mesh formation can be seen in the figure 2. For the piston displacement motion mesh morphing technique has been approached. The morphing motion reallocates mesh apexes in reply to the drive of control points. The mesh morphing uses control points and their associated displacements to produce an interpolation field all through the region which can then be used to dislocate the actual apexes of the mesh. Individual control point has a linked distance vector which stipulates the movement of the point within a single time-step.

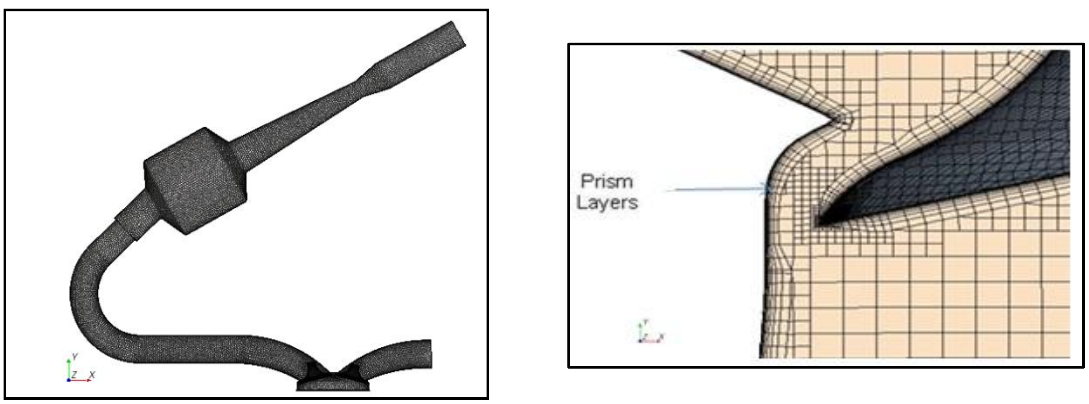


Figure 2: Trimmer mesh and hybrid mesh is generated for the wall treatment

4. RESULTS AND DISCUSSION

Flow structure has been studied through the valve and new designed intake manifold for the suction stroke. Volumetric efficiency, velocity, pressure, temperature and characteristics of turbulence kinetic energy and dissipation rate have been strong-minded. For the estimation of volumetric efficiency fluctuation at different engine speeds with respect to piston velocity has been plotted as shown in figure 3; the valve displacement is therefore correlated with the engine speed accordingly. The computed flow fields of different fluid mechanics entity deliberated in this study is at 5000 RPM.

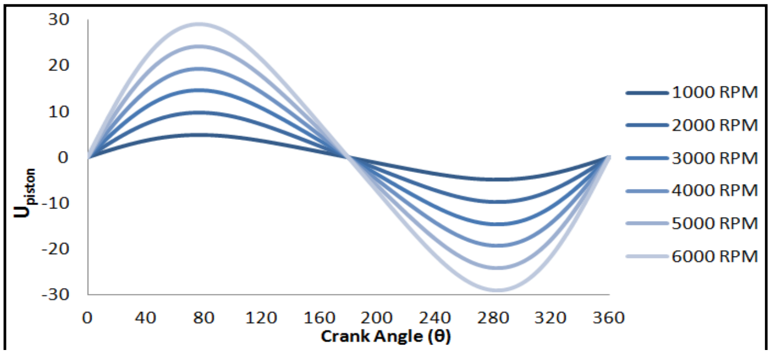


Figure 3: Piston velocity vs. Crank angle plot at different engine speed

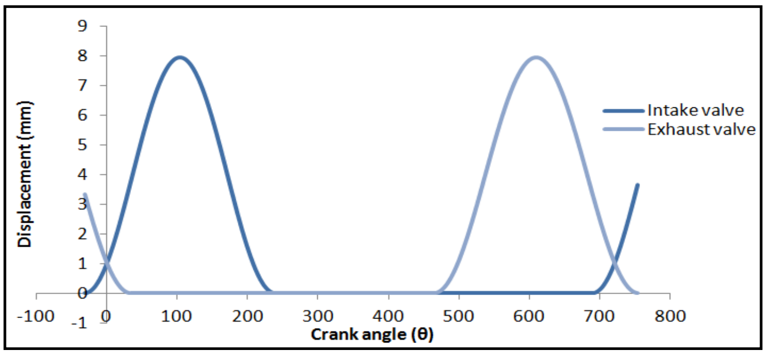
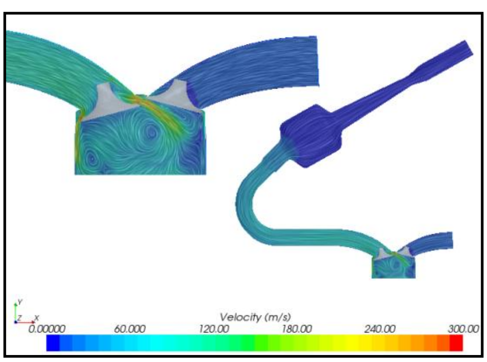


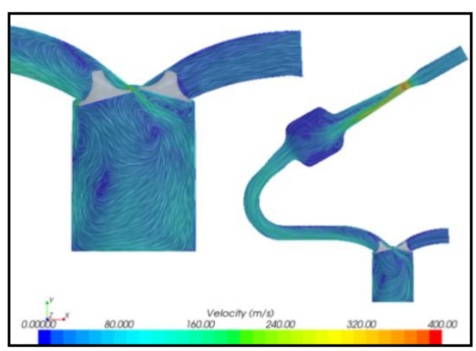
Figure 4: Inlet and Exhaust valve displacement with relative Crank angle.

4.1 Cold flow Analysis

Velocity Line Integral Convolutions (LIC's) during the intake stroke at θ = 60˚ and θ = 180˚ crank angle in the intake manifold and combustion chamber is shown in figure 5. The motion of air in the combustion chamber controls the air/fuel mixture and improves the flame velocity. Presence of swirl and tumble in the cylinder is highly preferable to have homogenized combustible mixture. LIC represents the air motion in the cylinder. It has been observed that at 60˚ crank angle downward motion of piston sucks the air charge, the valve moves downward restricting the flow of air. The restriction and geometry of valve face and valve seat makes the air accelerated form tumbles near the valve tip. This accelerated air with tumble makes the cylinder volume fills quite good. At 180˚ crank angle the velocity near the intake valve gets decreased and the velocity of air increases rapidly at the throat of restrictor, at this stage the piston velocity is zero as the piston has reached to the bottom dead centre. In the plenum turbulence has been created due to the sudden enlargement of the plenum chamber.



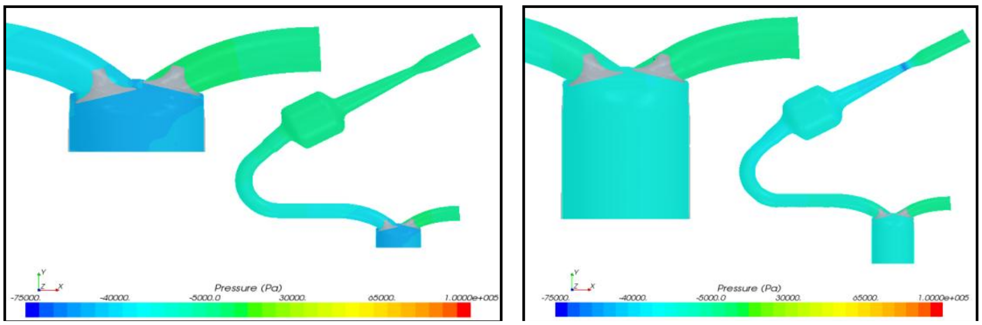
θ = 60



θ = 180

Figure 5: Velocity LICs distribution

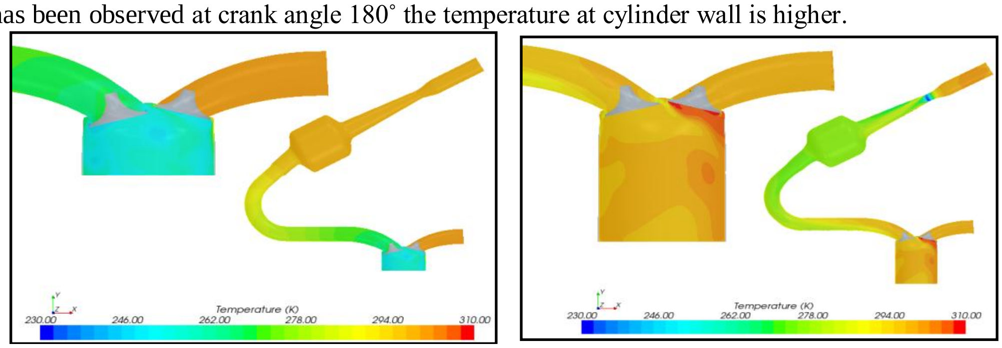
The pressure field spreading, shows the confrontation behaviors to the flow during the intake stroke. Figure 6 shows the pressure distribution on the sectioned plane. As the reference pressure has been taken by the zero gauge pressure the negative pressure field represents the suction in the cylinder. It is observed that the pressure at 60˚ crank angle is lesser than the 180˚ which is triggered due to air entrapped in the chamber during the downward motion of the piston. This indicates that the mass flow rate to the combustion chamber has been decreased. It is observed that the pressure is decreased by ~-40000Pa at 60˚ crank angle where as the pressure at BDC has been increased to ~-20000Pa.



θ = 60 θ = 180

Figure 6: Pressure distribution

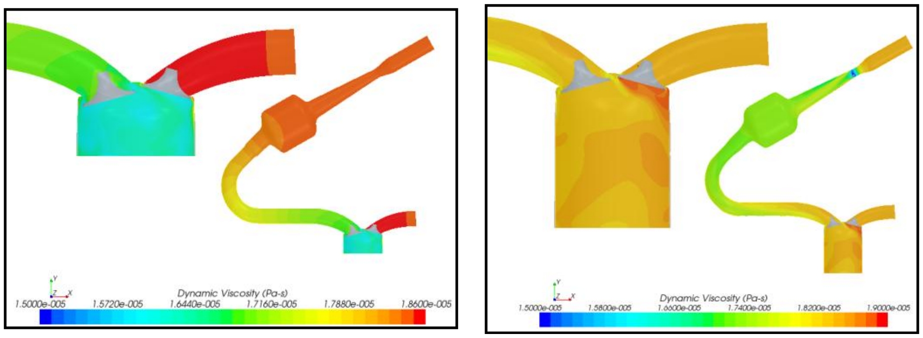
Temperature contours inside the combustion area presented in figure 7 is result of air motion with an assumption made, the atmospheric temperature and cylinder wall temperature are 300 K. Combustion of air charge is highly depends on the suction temperature. Also, the temperature causes the engine malfunctioning as the engine involves such material that deforms and hence, the sizes of the different parts which form the engine changes. Initially at the inflow the temperature is of atmospheric temperature, as the mass of air is only taken from the plenum chamber. But during the added travel of piston to BDC, flow gets accelerated at restrictor throat due to the suction the temperature starts decreasing, the noticeable change in temperature has been observed from the valve as due to greater velocity of flow. It has been observed at crank angle 180˚ the temperature at cylinder wall is higher.



θ = 60 θ = 180

Figure 7: Temperature flow field

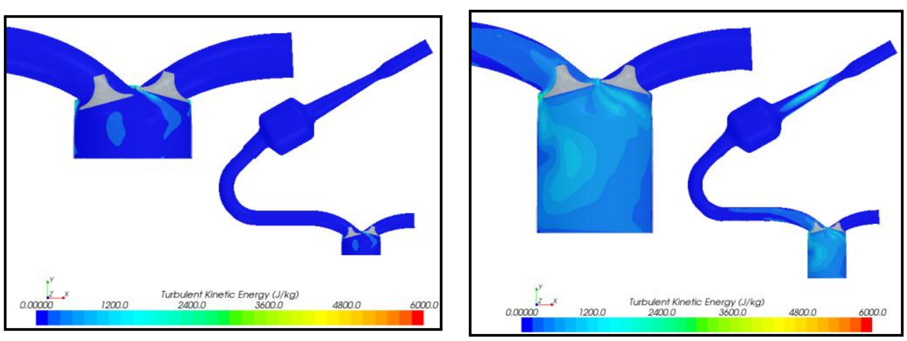
The dynamic viscosity is proportional to the temperature and density which is monitored by the Sutherland's law. It is observed in the figure 8 that the flow field of dynamic viscosity and temperature is quite similar.



θ = 60 θ = 180

Figure 8: Dynamic viscosity distribution

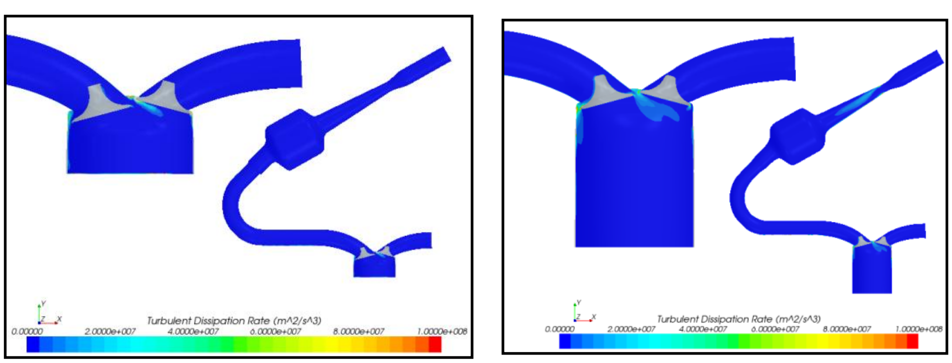
To characterize the turbulent flow two parameters are most important, one turbulent kinetic energy, *k* and other turbulent dissipation rate, *ε*. Velocity flow characteristic kinetic energy is related to turbulence intensity. Turbulence generated in the internal combustion engine during the suction stroke is always beneficial to optimize the chemically correct homogenized fuel/air ratio. Velocity fluctuations in the flow field has been detected with the turbulence dissipation rate. In figure 9, turbulent kinetic energy has been presented which illustrates the kinetic energy during the downward motion of piston. It is detected that at the valve tip region the value of kinetic energy is low in the chamber, but is bit high at the bypass section of intake valve. This increase in turbulent energy is owing to the strong gradient velocity of gases in the region. Massive energy has been produced in the chamber when the piston is on BDC, leading to form the homogeneous fuel/air mixture. Due to the narrow section of intake manifold restrictor the kinetic energy is seen to rise whiles the piston is at BDC.



θ = 60 θ = 180

Figure 9: Turbulent kinetic energy distribution

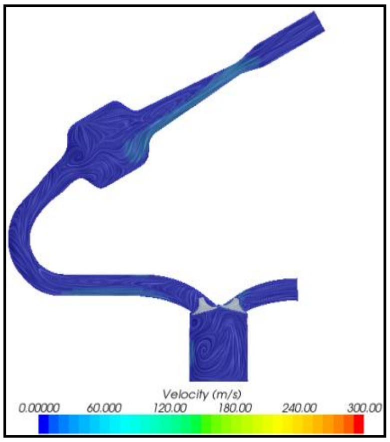
Figure 10 shows the dissipation rate in the combustion chamber. Flow field distribution is quite similar as of the kinetic energy. It is observed that, in the region where high kinetic energy was found the dissipation occurred at those spaces. This has been noticed near the valve tips of inlet valve. During the crank angle 60˚ the value of dissipation rate is ~ε=10^5, later during the crank angle 180˚ the intensity of ε decreases.



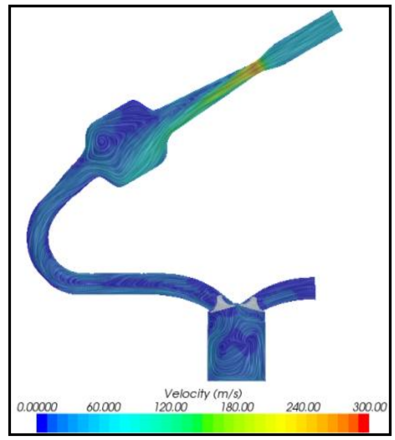
θ = 60 θ = 180

Figure 10: Turbulent dissipation rate

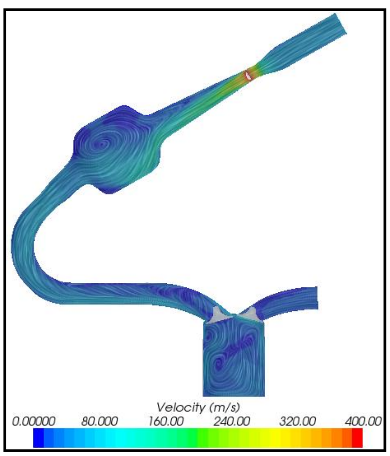
Figure 11 shows the velocity LICs distribution on the mid section plane in the intake manifold and the cylinder at the crank angle of 180˚ at different engine speeds. It has been investigated from the figures that, air swirl and tumble increases in the cylinder with increase in engine speed. There has been no air motion generated in the plenum at 1000 RPM but as the engine speed drives high the turbulence in the plenum has been noticed. At the throat the velocity increases as the engine speed increases, that is due to increase in number of suction strokes.



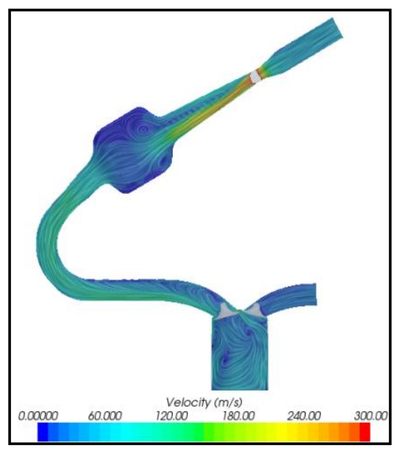
1000 RPM



3000 RPM



4000 RPM



5000 RPM

Figure 11: Velocity flow field in the intake manifold

4.2 Volumetric efficiency Appraisal

Volumetric efficiency is the mass of air equals to density times displacement. During the study, mass flow rate and the density change during the intake valve open period at different engine speed is measured. The measurement has been taken at the cross section just before the intake valve. The mass flow at different engine speed has been represented in figure 12 (a), it has been demonstrated from the plot that during the initial suction period (0-110˚ of crank angle) mass flow is greater. This could be the effect of the plenum chamber. The mass flow and density change obtained from the computational results in the volume flow calculated using Eqn. 6 and is summarized by the numerical method to obtain the total volume during the cycle.

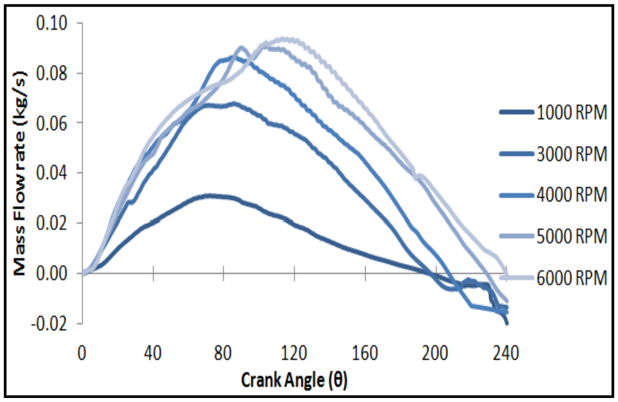


Figure 12 (a): Mass flow rate at various engine speed

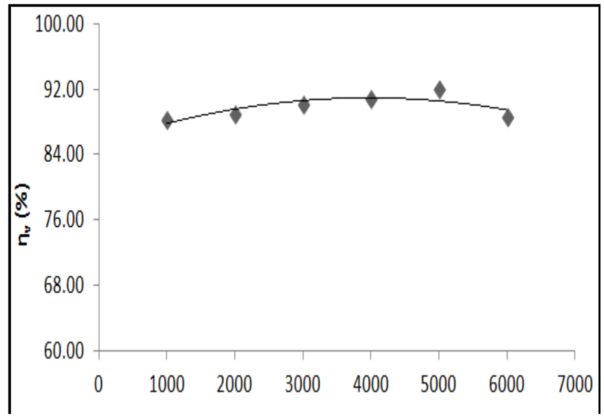


Figure 13 (b): Volumetric efficiency vs. Engine speed plot

$$V = \frac{1}{\rho}\int_0^T \dot{m}dt \qquad (6)$$

$$V_{actual} = \frac{T-0}{2N\rho}\sum_{T=1}^{N}[\dot{m}_1 + 2f\,\dot{m}_2 + ... + 2f\,\dot{m}_N + f\,\dot{m}_{N+1}] \qquad (7)$$

$$\eta_{volumetric} = \frac{V_{actual}}{V_{theoritical}} \qquad (8)$$

Similar method has been approached with the different engine speed data. And with the equation 14 the volumetric efficiency has been calculated which has been plotted in the figure 12 (b). The theoretical volume flow rate is 500cc.

5. CONCLUSION

The visualization and examination of cold flow in the cylinder physiognomies during the suction stroke is approved by standard CFD code. The investigation of flow has been visualized at two crank angles and it has been observed that the new design of intake is feasible with the engine. It has been demonstrated that new design provides significant turbulence. Air sucked by the engine primarily travels through plenum. During first 60° of crank rotation no turbulence or less movement in the upstream of plenum is observed. Plenum chamber works as a reservoir hence due to the large volume of plenum the pressure increases after approaching from restrictor section which essentially affects the volumetric efficiency.

ACKNOWLEDGEMENT

Author would like to thank IIT Gandhinagar to allow the use of computer resources at High Performance Computing Lab (HPCL@IITGN) and Prof. Murali Damodaran for being a mentor for this study.

REFERENCES

[1] Computational Fluid Dynamics and Multi-physics Engineering Software-STAR-CCM+ Version: 8.04.2013 User Manual, CD-Adapco, Lebanon, NH, USA, URL: (http://www.cd-adapco.com/products/star-ccm).
[2] Launder, B. E., & Sharma, B. I. (1974). Application of the energy-dissipation model of turbulence to the calculation of flow near a spinning disc. *Letters in heat and mass transfer*, *1*(2), (pp-131-137)
[3] McFarland Jr, J. D. (1976). *U.S. Patent No. 3,930,473*. Washington, DC: U.S. Patent and Trademark Office.

[4] Masumoto, I., & Takii, O. (1989). *U.S. Patent No. 4,809,647*. Washington, DC: U.S. Patent and Trademark Office.
[5] Masumoto, I., & Takii, O. (1991). *U.S. Patent No. 5,048,471*. Washington, DC: U.S. Patent and Trademark Office
[6] Narayanaswamy, K. (2006). *U.S. Patent No. 6,983,727*. Washington, DC: U.S. Patent and Trademark Office.
[7] Student Formula Rule book. (2014).
[8] Verkleeren, R. L. (1998). *U.S. Patent No. 5,762,036*. Washington, DC: U.S. Patent and Trademark Office.