Performance Analysis of Heavy Duty SPFI CNG Engine Manifold System

[1]Kiran N. Dadas, Ashok P. Tadamalle
[1]Department of Mechanical Engineering,
Sinhgad College of Engineering, Vadgaon, BK, Pune, 411041, Maharastra, India

[2]S. S. Thipse, K. P. Kavathekar, S. D. Rairikar, S. B. Sonawane, P.S. Sutar
[2]Powertrain Engineering Department,
The Automotive Research Association of India, Pune

Abstract—**The primary modifications are required in Internal Combustion Engine System viz intake manifold, piston and Exhaust after treatment system to obtain optimum power, torque, fuel efficiency and to reduce harmful emission gases. These parameters are responsible for improvement in the performance of Engine. The purpose of the intake manifold is to distribute air-fuel mixture uniformly for all cylinders. The paper aims to design Intake Manifold and investigate the effect of Fuel Injection location and Air-Fuel mixture on the Performance of the CNG Single point fuel injection Engine. The intake manifold is modeled using reverse engineering data. The manifold system is analyzed for stress and deflection. The Air-Fuel mixture is evaluated using Computational Fluid Dynamics analysis based on the uniformity index by varying injection location. The effects of turbulence are represented by k-Ɛ turbulence model. The Engine Performance is estimated using 1D simulation software and the results are compared with Experimental results.**

Keywords—Intake Manifold, SPFI, CFD, Uniformity Index, Design of Injection Location

I. INTRODUCTION

In Internal Combustion engine the intake manifold is the part of the engine between the Cylinders and the throttle body. In a multi-cylinder engine its primary purpose is to evenly distribute the air flow between each cylinder, and to create the homogeneous fuel air mixture. The mass flow rate of air which is entering in the engines cylinders does large impact on the volumetric efficiency. There are two types of fuel injection systems Single Point Fuel Injection (SPFI) and Multi Point Fuel Injection (MPFI) System. SPFI is a system that has a single injector, or a group of injectors clustered together in one, usually centralized spot on the intake manifold. In this system the injection location plays vital role to improve the uniformity of the Air-Fuel Mixture.

Maji Luo Guohua et. al [1] have numerically simulated Three-dimensional steady flowin two types of inlet manifold using the arbitrary Lagrangian – Eulerian(ALE) method. The effects of turbulence are represented by k-Ɛturbulence model. Mass flow rates of the systems are calculated and compared to choose the efficient intake manifold design. Harishchandra Jagtap et.al [2] have studied the air-fuel flow pattern of long and short runner intake manifold with different plenum chambers and the flow distribution of air from plenum to individual runners using CFD analysis. LuizOtavioF.T.Alves et.al [3] have investigated the effects of different intake runner length and diameter on the performance of a four stroke and single cylinder IC Engine and results are validated using GT-Power simulation software.

Massimo Masi et.al [4] have showed that the design of intake manifold and the valve port in IC Engine effects on volumetric efficiency and flow intensity. CFD analysis has been performed on the intake valve of the high speed spark ignition Engine to investigate the reliability of polyhedral grids of different size and to assess the required mesh size. A Manmadhachary et.al [5] have carried out the study to develop a spiral intake manifold model to predict gas flow in the intake system of single cylinder IC Engine. intake manifold has significant effect on the volumetric efficiency and the performance of the engine.

Shashank Ghodke et.al [6] have worked on the intake runner diameter and valve timing of manifold system by individually varying them. simulation were carried out using Engine simulation software Ricardo wave to find the effect of intake runner diameter and timing on the engine performance and the results are compared with chassis dyno test results. Dileep Namdeorao Malkhede et.al [7] have investigated the effect of intake length for different speed of the Engine on volumetric efficiency. 1-D simulation was carried out to predict the pressure wave at two different locations on intake manifold and compared with test data. Kriti Gupta et.al [8] have carried out the fluid flow analysis for the flow through the intake manifold with different cross section of throttle body using CFD analysis. The nature of the flow through the valve and the comparative study of the pressure and velocity variation have been made. Michal Bialy et.al [9] have carried out CFD analysis of Engine head with different CNG injector location to demonstrate air fuel stratification using Influence of the injector nozzle position

The CFD is successfully used based on air fuel uniformity mixture [1-3,5].several authors have performed analysis using different turbulence models and same boundary conditions [5,9,].The effect of Air-Fuel mixture on engine performance parameters like power, torque & BSFC are studied using Engine simulation software Ricardo wave and 1D simulation software [7, 8]. Several authors have done simulation on single injection geometry the work has not been done for multiple injection location [1,2]. This paper aims to study effect of Air-Fuel mixture on engine performance using 3D steady state analysis of intake manifold with five different injection locations. Another application of this study is the observation of the streamline (flow path line) inside the manifold. Performance of injection location is simulated by using k-ε turbulence model. Different mass flow rate are provided during analysis on CNG inlet and air inlet whereas different pressure conditions are given at inlet and outlet. The

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 06, June-2019

results of analysis are validated with GT Power Software. The work has been carried out on a 6 cylinder CNG Engine of 6 liter capacity. This produces a power of 130 HP at 2400 rpm.

II. STRESS AND DEFORMATION ANALYSIS

This section deals with the prediction of stress and deformation induced in the intake manifold. The database for intake manifold is obtained by Re-engineering of existing box manifold used in automobiles. The Existing H6NA CNG Engine SPFI Intake manifold system is modified with different Injection location is then modeled using CATIA software. The assembly of manifold system consists of plenum, hub, throttle body; these are created separately and assembled by applying appropriate tolerances and constraints. In existing box manifold system the injection position 175 mm away from throttle body. The developed model was then extruded to meet 3.65 liter volume requirements. To obtain improved uniformity index the modified intake manifold is Re-modified with a different injection location at 155mm, 225mm, 145mm, 125mm away from throttle body excluding 175mm. The Intake manifold refers to an engine part that supplies the air and fuel mixture to the cylinders. Intake manifold plenum facilitates the distribution of this mixture. It's of rectangular shape having volume 2.65 litres, length and width is 655mm and 113mm respectively. Hub is the part between throttle body and plenum. Its of circular shaped pipe having 60mm diameter. Throttle body is responsible for controlling the amount of Air-Fuel that flows into an engine. In Single point fuel injection system the fuel injection location plays a crucial role on Air-Fuel mixture. The system has 14 inlets of 6mm diameters and 175mm length away from throttle body.

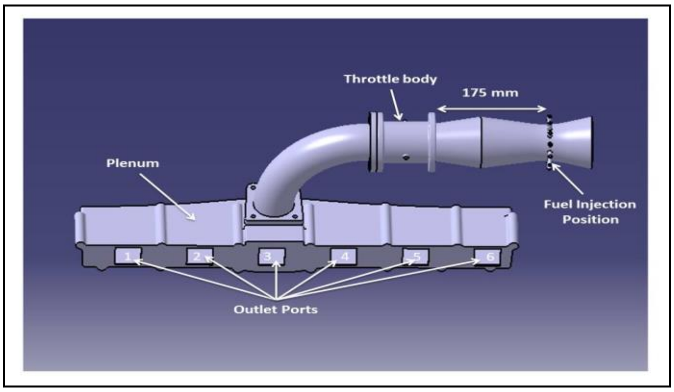

Fig. 1. Existing Intake Manifold

The CAD model is saved in STP file and imported into ANSYS environment. The imported geometry was then cleaned for its missing line, gap filling and ambiguous sections etc. Imported and meshed cleaned geometry is meshed with tetrahedral element having 0.5mm minimum size and 40mm maximum size of the element.

The minimum size of meshing has been generated at the critical part of the geometry. Tetrahedral elements can fit better complex geometry. The type and size is decided based on suitability and type of analysis. Meshed Intake manifold system consists of 356797 elements and 620319 nodes.

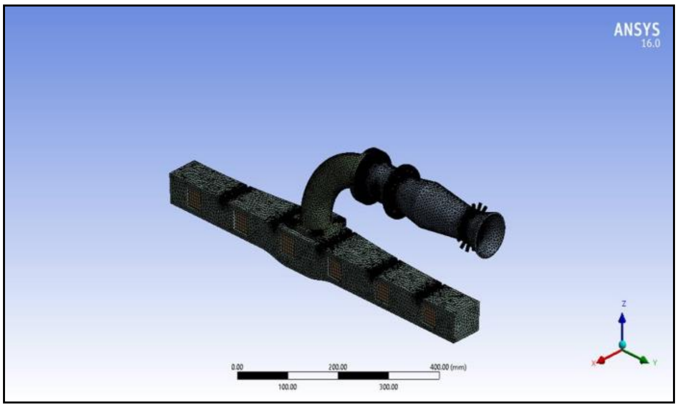

Fig. 2. Mesh Model of Intake Manifold for Structural Analysis

Different boundary conditions are provided at inlet and outlet of inlet manifold. 35N vertically downward Force is given on the manifold pipe by considering self weight and mixer weight of the manifold system. The intake manifold is connected to the intake side of the engines head so the fixed support is given to the outlet side of the manifold.

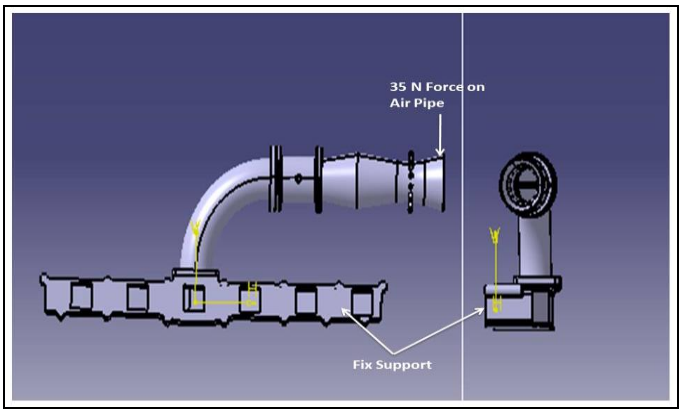

Fig. 3. Boundary Conditions for Static Structural Analysis

A static structural analysis determines the displacements, stresses, strains, and forces in structures or components caused by loads that do not induce significant inertia and damping effects. Steady loading and response conditions are assumed that is the loads and the structure's response are assumed to vary slowly with respect to time.

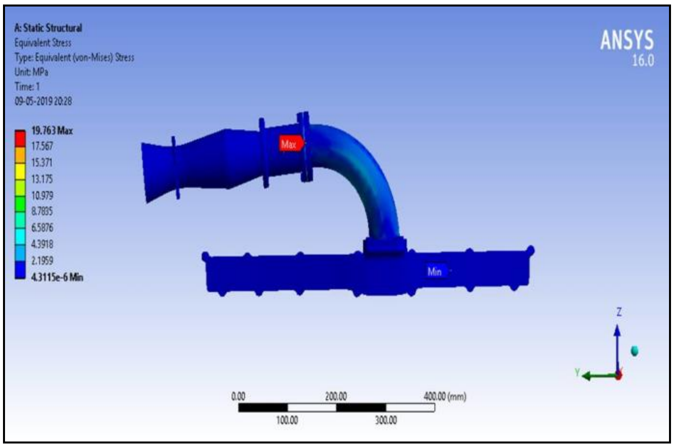

Fig. 4. Static Structural Analysis of Manifold System

If the Von-mises stress is less than yield stress of the material, then the product analyzed is safe, else it is of failure type. The maximum Equivalent stress is 19.76 MPa. The maximum Stress is observed on Manifold System. Therefor there is not significant stress is coming on the intake manifold. Stress on the manifold is found very less from yield stress so the system is structurally stable.

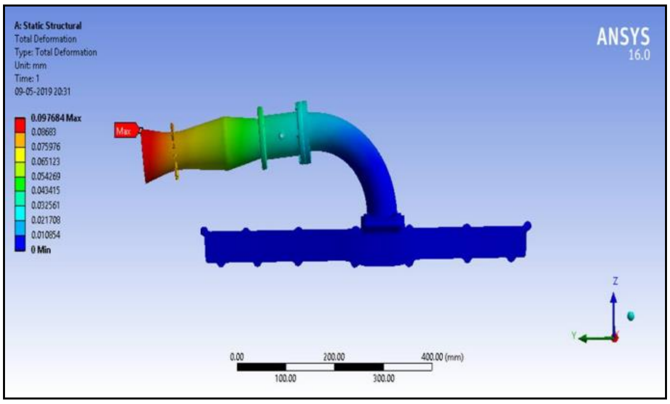

Fig. 5. Total Deformation of Manifold System

The Maximum deflection on intake manifold is observed. The system has a Maximum deflection which is 0.097mm. Therefor there is not significant deformation found on the intake manifold. Totally it is found that the system is having negligible deformation so the system is structurally stable.

III. EFFECT OF INJECTION POSITION ON UNIFORMITY INDEX

The CAD model is saved in STP file and imported into ANSYS environment. The imported geometry was then cleaned for its missing line, gap filling and ambiguous sections etc. Imported and meshed cleaned geometry is meshed with tetrahedral element having 0.6433mm minimum size and 82.34mm maximum size of the element.

The minimum size of meshing has been generated at the critical part of the geometry. Tetrahedral elements can fit better complex geometry. The type and size is decided based on suitability and type of analysis. Meshed Intake manifold system consists of 562007 elements and 111168 nodes.

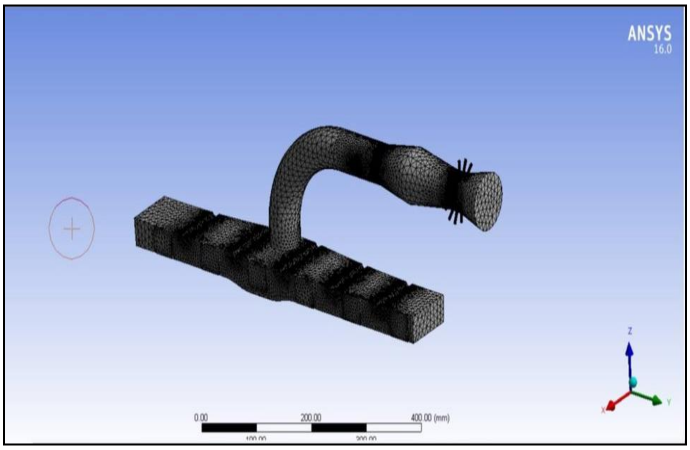

Fig. 6. Meshing of Intake Manifold for CFD Analysis

In a CFD analysis, we need to assign many boundary conditions to "define" how the system operates. Out of many boundary condition types (inlet, outlet, wall, symmetry, etc.), a wall boundary condition is without a doubt the most commonly used boundary condition in a CFD analysis.

Different mass flow rate are provided during analysis on CNG inlet and air inlet whereas different pressure conditions are given at inlet and outlet. Ambient pressure and temperature is given at Air inlet and CNG inlet. Mass flow rate are provided 0.09638 kg/s at air inlet and 0.005608kg/s at CNG inlet during analysis. When suction stroke occurs piston sucks air through manifold so the negative pressure comes at the outlet of the manifold which is -3600 Pascal.

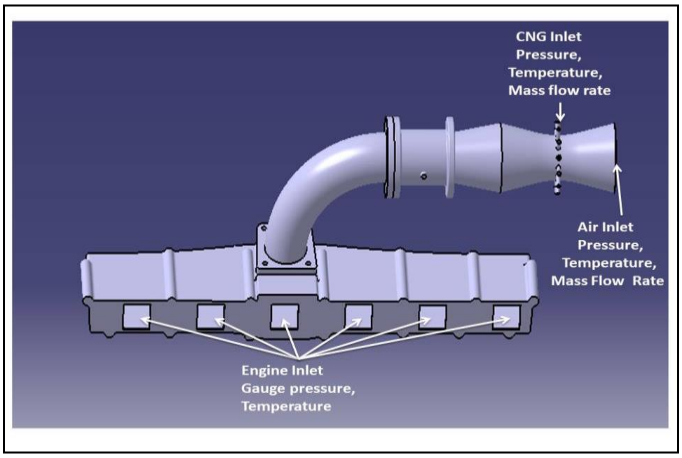

Fig. 7. Boundary Condition for CFD Analysis

To verify the velocity distribution and uniformity index of the Air\Fuel mixture computational fluid dynamics of manifold system has been carried out. A three dimensional CFD model has been built in order to accomplished the targets involved. All the relevant aspects of the flow has been taken into account and fully incompressible viscous flow simulation with turbulence modeling for steady state condition has been performed. Five different injector positions geometry were tested and the comparison among them was conducted in order to verify which one is having better uniformity index. The uniformity of the flow at the outlet of each runner is evaluated. The maximum velocity magnitude and are calculated for different section of pipe, hub and plenum.

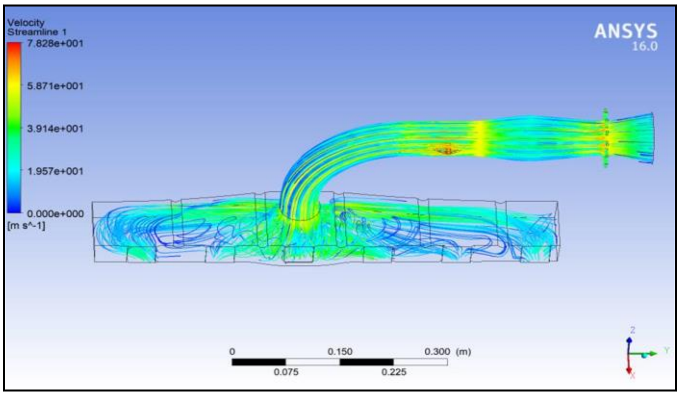

Fig. 8. Velocity Streamline with Injection Position 175mm from Throttle Body

From above result it is found that the flow uniformity index at outlet of each runner is 0.8295 and maximum velocity magnitude in the manifold is 78.28 m/s. it is observed that the streamlines are laminar in the region of hub, pipe and more turbulent in the region of plenum and throttle body. More turbulent mixture is observed in the region of cylinder 4, 5 and 6. It is observed that the cylinder no 1 is having more Air-Fuel mixture than the other cylinders. The injection mixer and throttle shaft provides restriction to the intake flow. Lesser is the restriction across the flow, lower is the loss of energy. This increases velocity at the outlet of the flow supply and throttle body.

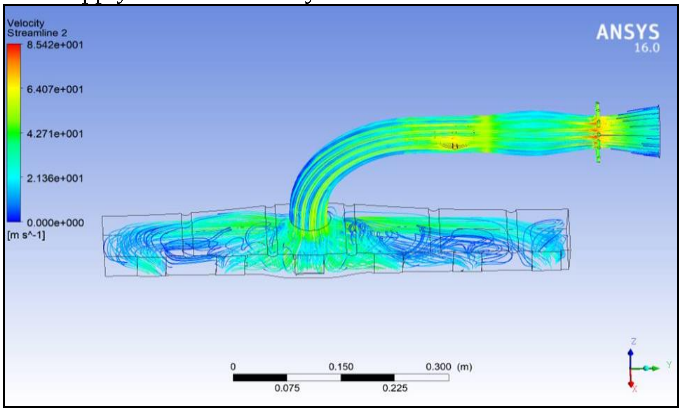

Fig. 9. Velocity Streamline with Injection Position 155mm from Throttle Body

From fig 9 it is found that the flow uniformity index at outlet of each runner is 0.8331 and maximum velocity magnitude in the manifold is 85.42 m/s.

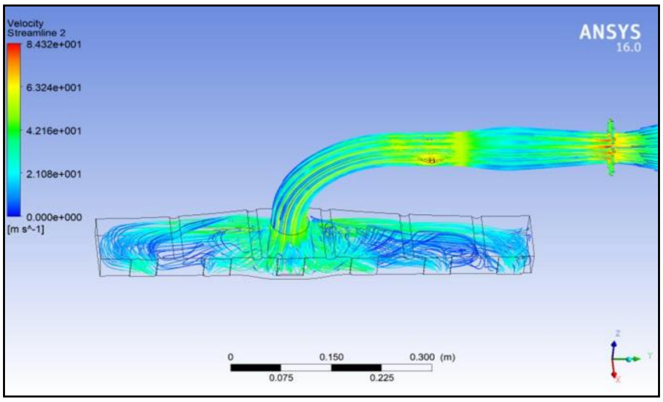

Fig. 10. Velocity Streamline with Injection Position 225mm from Throttle Body

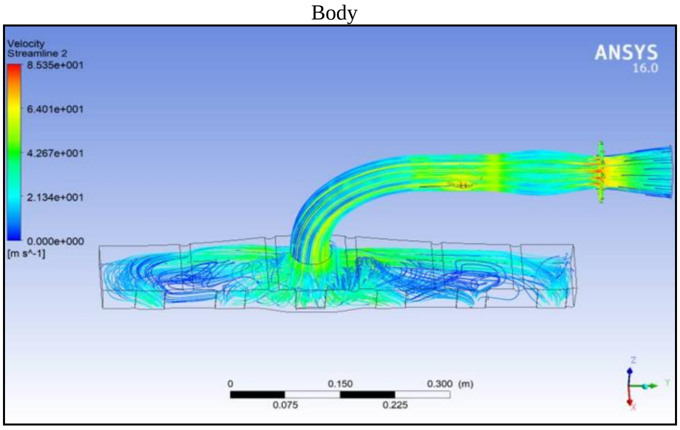

Fig. 11. Velocity Streamline with Injection Position 145mm from Throttle Body

From fig 10 it is found that the flow uniformity index at outlet of each runner is 0.8104 and maximum velocity magnitude in the manifold is 84.32 m/s.
From Fig 11 it is found that the flow uniformity index at outlet of each runner is 0.8356 and maximum velocity magnitude in the manifold is 85.35 m/s.

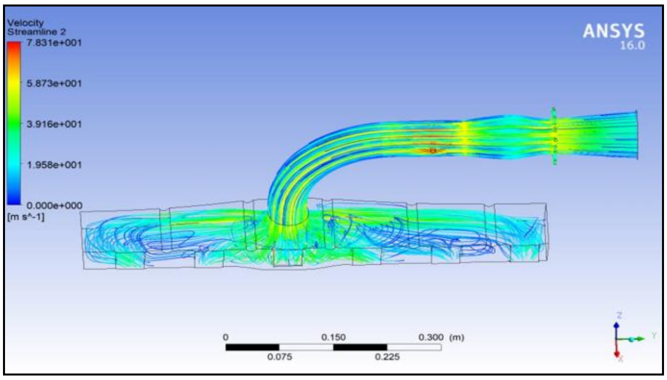

Fig. 12. Velocity Streamline with Injection Position 125mm from Throttle Body

From above result it is found that the flow uniformity index at outlet of each runner is 0.8441 and maximum velocity magnitude in the manifold is 78.31 m/s.
Boundary conditions (Air inlet, CNG inlet, Mixture at Engine inlet) are given while solving the problem. Uniformity index, velocity distribution have been calculated for five different geometries.

TABLE I. Uniformity Index and Maximum Velocity Magnitude for Different Injection Location

Parameters	**Injection position from throttle valve**				
	175mm	*155mm*	*225mm*	*145mm*	*125mm*
Uniformity index	0.8295	0.8331	0.8104	0.8356	0.8441
Max Velocity magnitude (m/s)	78.28	85.42	84.32	85.35	78.31

IV. PERFORMANCE PREDICTION FOR CNG ENGINE

The benchmarked data of CNG Engine is used for preparation of GT model. Fig 17 shows the 1-D model which is used for simulation of CNG to validate power, torque and of the engine. From the results, we can conclude that the power, torque parameters of the CNG engine are within acceptable limits for the simulated results.

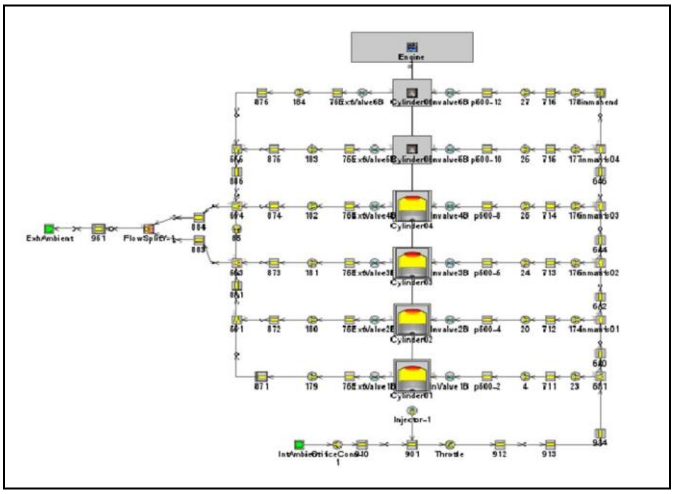

Fig. 13. CNG Engine 1-D Simulation Model

V. RESULT AND DISCUSSION

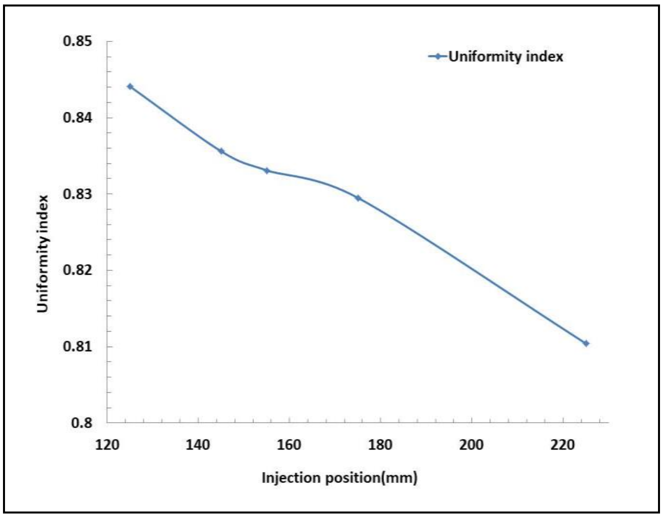

Fig. 14. Uniformity Index for each Injection location

Fig 14 shows Injection position away from throttle body and towards air filter shifting reveals that, as it shifts towards air filter uniformity index decreases and as it shifts towards throttle body uniformity index increases. Intake plenums flow does not create turbulent area and the lines are somewhat well behaved due to expansion in the manifold volume. As the injection position shift towards throttle body the velocity is increasing and more turbulent flow is created. Due to this when the injection location near to throttle body the better uniformity index is obtain at the outlet of the manifold system.

Fig 15 shows the maximum velocity magnitude at the inlet of the plenum for different injection location. The throttle shaft and throttle plate provide restriction to the flow of intake air. This increases velocity at the outlet of throttle body. Injection location does important role on the velocity magnitude. As the injection position shift towards throttle body the velocity is increasing and more turbulent flow is created.

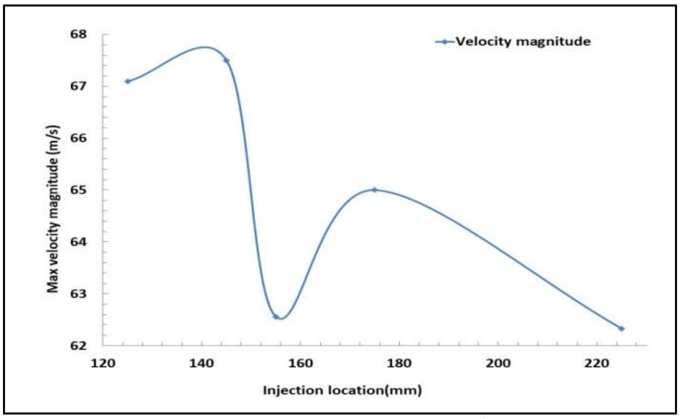

Fig. 15. Max Velocity Magnitude at Inlet of Plenum

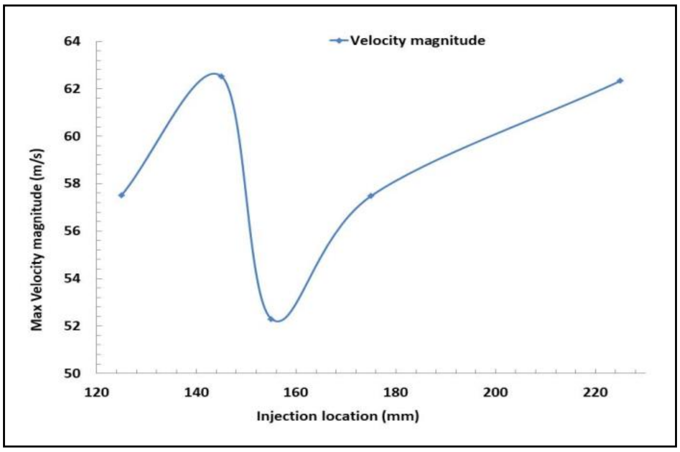

Fig. 16.Max Velocity Magnitude at Cross Section of Plenum

Fig 16 shows the maximum velocity magnitude at cross section of plenum for different injection location. It can be seen from this figure that there is uneven velocity magnitude between the different injection locations. From 225 to 155 it decreases and suddenly for 145 it increases. This is due to the change of geometry and injection change.

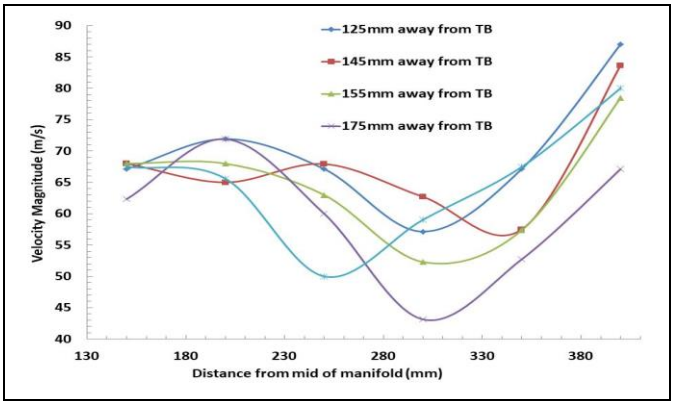

Fig. 17. Velocity Magnitude Representation for Different Injection location

Fig 17 shows the velocity magnitude for different injection location over different section of hub and throttle body. From the fig it is seen that for every injection location the nature of the velocity magnitude is same the velocity is increases in the region of injection position and throttle body. Because the throttle shaft and throttle plate provide restriction to the flow of intake air. This increases velocity at the outlet of throttle body. But the velocity is high for the 125 mm injections location due to this the more turbulent flow is created and more uniformity index is obtained.

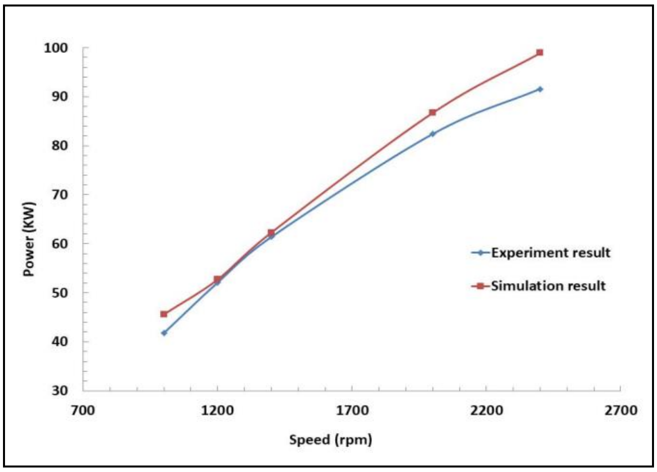

Fig. 18. Power comparison for Experimental and Simulation

Fig 18 shows the effect of speed on the engine performance parameter power. It is observed that with the increases in engine speed the power of engine also increases. The highest power is observed at 2400 rpm. The simulation results are validated using experimental dynamometer test result

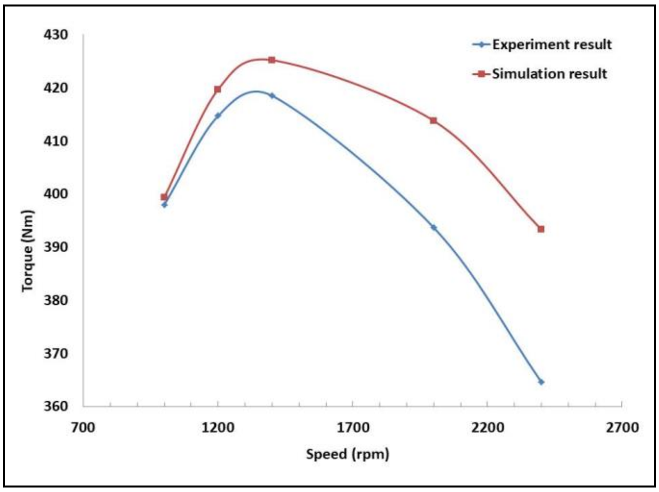

Fig. 19. Torque Comparison for Experimental and Simulation

Fig 19 shows the effect of speed on the engine performance parameter torque. It is seen that the engine efficiency is at the maximum at a speed where it produces its peak-torque. If you raise the engine above this speed, its torque starts to decrease because of the increased friction of the engine's moving parts. Here the peat torque is 425 Nm at 1400 rpm

CONCLUSIONS

1. The maximum Stress and Deformation are observed on Manifold System. Stress on the manifold is found very less from yield stress so the system is structurally stable.
2. Improvement in uniformity index achieved by taking injection position towards throttle body.
3. As the injection position shift towards throttle body the velocity is increasing and more turbulent flow is created.
4. From CFD Flow analysis it is found that the cylinder number one is having more Air/Fuel mixture than other cylinders.
5. It is observed that with the increases in engine speed the power of engine also increases. The highest power is observed at 2400 rpm.
6. It is seen that the engine efficiency is at the maximum at a speed where it produces its peak-torque if you raise the engine above this speed, its torque starts to decrease because of the increased friction of the engine's moving parts.

ACKNOWLEDGEMENT

The authors would like to thank Automotive Research Association of India, Pune and all Supportive Staff at ARAI for their encouragement and Guidance during the project. Author would also thankful to Dr. T. A. Jadhav PG Coordinator, Department of Mechanical Engineering, prof. V. N. Kapatkar, Head, Dr. S. D. Lokhande, Principal, Sinhgad College of Engineering, Pune.

REFERENCES

[1] Cesareo de La Rosa Siqueira, Martin Poulsen Kessler, Luiz Augusto Rachid de Araujo, Erlon Castro Rodrigues. (2006) "Three-dimensional Transient Simulation of an Intake Manifold using CFD Techniques" SAE International, Paper No 2006-01-2633.

[2] Maji Luo Guohua Chen Yankun Jiang Yuanhao Ma. (2001) "Numerical Simulation of Flows in Multi-Cylinder Diesel Engine Inlet Manifold and its Application" SAE International, Paper No 2001-01-001.

[3] Harishchandra Jagtap, Chavan Vinayak and Ravindra Koli (Mahindra & Mahindra Ltd, India). (2011) "Intake System Design Approach for Turbocharged MPFI SI Engine" SAE International Paper No 2011-26-0088.

[4] Luiz otavio ferrao texeiraaalves, Vagner abramchuk. (2017) "Design of a new intake manifold of a single cylinder engine with three stages" SAE International, Paper No 2017-36-0172.

[5] Massimo Masi, Lorenzo Artico, Paolo Gobbato. (2015) "Measurement of the Intake and In-Cylinder Flow Field to Investigate the Reliability of CFD Steady-State Simulations for Actual Engines" SAE International, Paper No 2015-24-2404.

[6] Manmadhachary, M. Santosh Kumar, Y. Ravi Kumar. (2017). "Design and manufacturing of spiral intake manifold to improve volument efficiency of injection diesel engine by AM process" Sciencedirect.Material today proceeding.PP.1084-1090.

[7] Shashank Ghodke, Saiful Bari. (2018) "Effect of Integrating Variable Intake Runner Diameter and Variable Intake Valve Timing on an SI Engines Performance" SAE International, Paper No 2018-01-0380.

[8] Dileep Namdeorao Malkhede (College of Engineering), Hemant Khalane(Greaves Cotton Ltd).(2015) "Maximizing Volumetric Efficiency of IC Engine through Intake Manifold Tuning" SAE International, Paper No 2015-01-1738

[9] Kriti Gupta, Saumya Sharma, Jayashree Aseri, Anupriya. (2016 August) "CFDAnalysis of Flow through a Throttle Body of a Spark Ignition Engine for different Throttle Valve Shaft Configurations" International Journal of Engineering and Technical Research (IJETR) ISSN: 2321-0869 (O) 2454-4698 (P)

[10] Chawin Chantharasenawong (KMUTT) (2017) "Improvement of Formula Student Racecar Performance with Variable Runner Length for the Intake System" SAE International, Paper No 2017-01-1083