Design Modification of Intake Manifold of a Three Cylinder IC Engine to Improve Volumetric Efficiency

B. Dasendra Kumar[1] Dr. P. Ravindra Babu[2] Dr. J.A. Ranga Babu[3]
[1,2,3]Department of Mechanical Engineering
[1,2,3]Gudlavalleru Engineering College, India

Abstract—*Geometrical design of intake manifold is a vital parameter to improve the performance of an I.C. Engine. Unequal velocity distribution of intake air at runner's outlets of intake manifold makes it less efficient. The present work aims to make this unequal distribution of velocity nearly equal manner with increase of velocity at outlets without any major modification in design of intake manifold. A 3 Dimensional model is developed in ANSYS workbench for original geometry of intake manifold and numerical analysis is carried out on it using commercial CFD software ANSYS-FLUENT 16.0. An experimental test rig is developed to validate the numerical model and noticed a closer approximation with less than 7 % deviation between numerical and experimental outcomes. In view of enhancing the performance of IC engine the design of intake manifold numerical analysis is carried out by modifying the design of intake manifold and analyzed using ANSYS FLUENT. The results of original and modified designs are compared in terms of pressure developed and velocity of air (charge) flow. It is noticed from the results that velocity distribution is more uniform compared to original model pressure losses are reduced. The nearly equal velocities in all three runners are achieved in the inlet manifold by redesigning the plenum of intake manifold.*

Keywords— IC engine, Cylinder, Volumetric Efficiency

I. Introduction

An inlet/intake manifold is one of the salient part of IC engine that supplies the fuel/air mixture to the engine cylinder. The primary function of the intake manifold is to evenly distribute the intake charge to each intake valve/port in the cylinderhead. The ideal intake manifold evenly distributes pre-combustion mixture to the piston valves. Even distribution is important to optimize the efficiency and performance of the engine. It may also serve as a mount for the carburetor, throttle valve, fuel injectors and other components of the engine. The intake manifold has historically been manufactured from aluminium or cast iron but days are turned towards the use of composite plastic materials is gaining popularity in recent past. The photographic view of conventional intake manifold as shown in Fig. 1.

Fig.1.1: Intake manifold

The intermittent or pulsating nature of the airflow through the intake manifold into each cylinder may develop resonances in the airflow at certain speeds. These may increase the engine performance characteristics at certain engine speeds, but may reduce at other speeds, depending on manifold dimension and shape. The intake manifolds can broadly classified into Static length intake manifold and Variable length intake manifold.

A. Static Length IntakeManifold

Static intake manifolds for vehicles have fixed air flow geometry and static intake manifold. With a static intake manifold, the speed at which intake tuning occurs is fixed. A static intake manifold can only be optimized for one specific rpm.

B. Variable Length IntakeManifold

Variable length intake manifold technology uses the pressure variations generated by the pulsating flow due to the periodic piston and valve motion to produce a charging effect. By varying the intake length/volume, we can operate engine over a broad speed range. Various designs for variable intake geometry have met with varying degrees of success. The designs of the variable intake manifolds may be rather complex and expensive, to produce. Difficulty in servicing and a limited range of variable tuning may also be disadvantageous design results of variable intake manifolds.

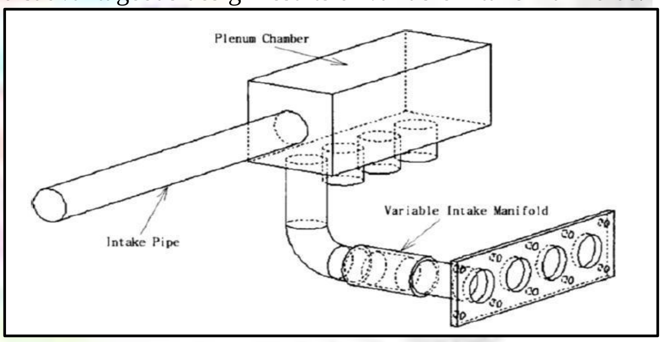

Fig.1.2: Variable length intake manifold

C. Scope for Modification of IntakeManifold

The main task of an inlet manifold is to distribute air inside the manifold runner uniformly, which is essential for an optimized inlet manifold design. The inlet manifold design has strong influence on the volumetric efficiency of the engine. An uneven air distribution leads to less volumetric efficiency, power loss and increased fuel consumption. Depending on the amplitude and phase of pressure waves inside the inlet manifold, filling of cylinders by air can be affected positively or negatively. The amplitude and phase of these pressure waves depend on inlet manifold design, engine speed and valve timing. The unsteady nature of the induction means and the effect of the manifoldonchargingareextremelydependentupontheenginespeed.Thisisbecausetheentryof air inside the inlet manifold is a function of varying pulses. Therefore these pulses should be fine- tuned in engine manifolds to give required power. The main objective of the current study is the development in the field of geometrical modifications in IC engine are still in progressive state, improvements in design can be further done for achieving good performance. Therefore, the current work is emphasis on design modification of intake manifold in view of achieving high volumetric efficiency as well as emission control.

II. Methodology

To achieve a best intake manifold of I.C engine, a methodology is proposed as described below:

- Experimental analysis of intakemanifold.
- Modelled the proposed intake manifold in Ansys workbench.
- CFD analysis of the intakemanifold.
- Comparison of experimental and CFD analysis results and
- Proposed modifications for a given intakemanifold.

III. Experimental analysis of intakemanifold

The Experiment is planned to conduct on FORD FIGO, 8V TDCI Diesel Engine, indirect injection, 3 Cylinder water cooled having 1499 cc.

Fig.3.1: Intake Manifold

For making experiment setup various instruments are required like Anemometer to measure the air flow, U-tube manometer to check the pressure at inlet and outlets andvarious attachments example pressure taps, pipes attachmentsetc.

Fig.3.2: U tube manometer

Fig.3.3: Anemometer

Fig.3.4 Pressure taps

A. Experimentalsetup:

The setup shown below is for taking out the reading at outlets of intake manifold with Anemometer and U-tube manometer.

Fig.3.5: Experiment setup

B. Experimental Procedure

In this experiment, we have to find the pressure & velocity of runner outlets of intake manifold. In this three runner outlets are fitted with pressure taps & these pressure taps are connected individual to the u-tube manometer to find pressure losses at each individual runner, after that we have to find velocity at each individual runner by using Anemometer.

IV. NUMERICAL APPROACH

A. Modeling of Intake manifold system

Proposed intake manifold is modelled in Ansys workbench in design module of ANSYS 16.0. The generated 3 Dimensional model is meshed in the same Ansys environment using standard Meshing tool. In the later phase, all the sections are properly named and interfaced. Modeler don't have the classified geometry of the intake system there for by using the reverse engineering process the geometry is modeled in software as precise as possible for exact similarity to our physical model. Here In reverse engineering process modeler simply measures the various dimensions of intake system and then created model in software.

B. CFD analysis of intakemanifold

Computational fluid dynamics is a tool to find numerical solution of governing equation using high speed digital computer. So 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. By using the CFD code, flow field can be predicted by solving the governing equations viz., continuity, momentum and energy. The renormalization group theory (RNG k-ε) turbulent model is used for analyzing the physical phenomena involved in the change of kinetic energy.

C. Strategy for study of intakemanifold

For finding out losses and optimized geometry, study will take place on following models of same intake manifold with small modifications:

– Model-1 (Model of original geometry)
– Model-2 (Model with modified geometry at end of runners & plenum)

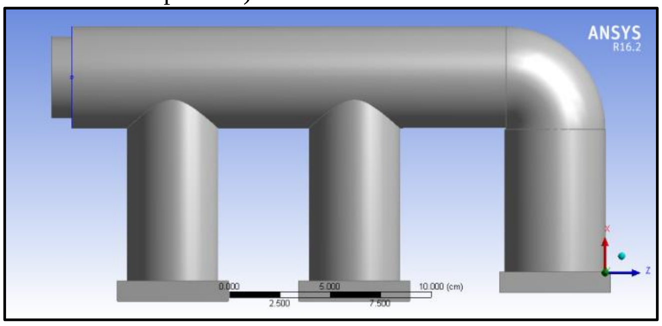

Fig.4.1: Model-1

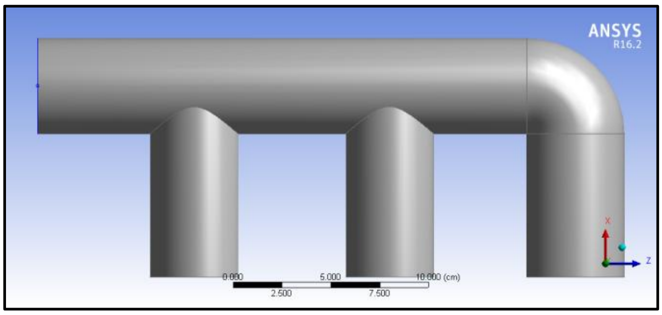

Fig.4.2: Model-2

D. Meshing properties

	Model-1	Model-2
Nodes	24799	33969
Elements	127678	181206

V. RESULTS & DISCUSSION

From the above experimental and CFD analysis the following results are observed

A. Experimental result of Intakemanifold

Inlet (m/s)	Outlet1(m/s)	Outlet 2 (m/s)	Outlet3(m/s)
32.37	9.68	23.04	26.57
33.15	8.09	23.38	27.12
35.57	11.36	23.64	29.22
35.62	10.75	23.82	29.92

Table 5.1 Experimental reading of Anemometer

INLET(m/s)	Outlet1 (pa)	Outlet2 (pa)	Outlet3 (pa)
32.37	809.21	805.69	804.36
33.15	790.17	787.21	784.58
35.57	736.41	734.29	730.07
35.62	735.39	733.46	730.89

Table 5.2 Experimental reading of U-tube manometer

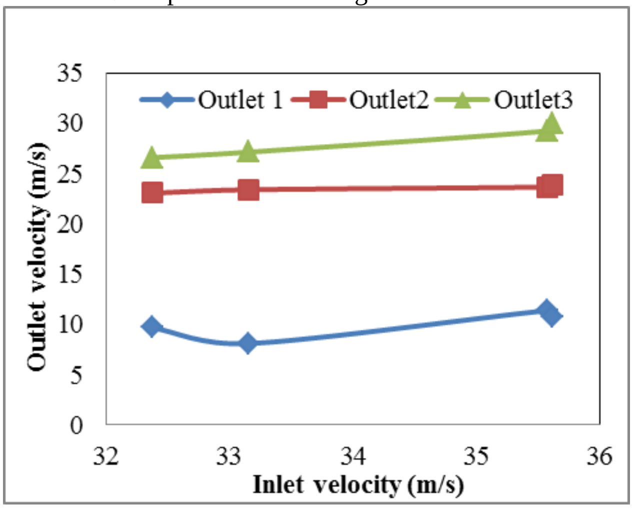

Fig.5.1: Velocities at outlets

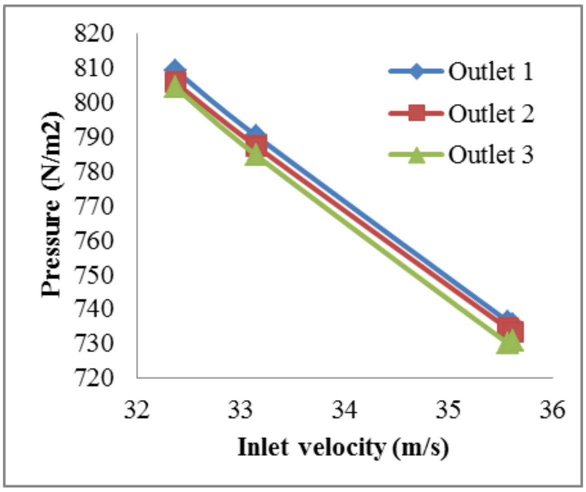

Fig.5.2: Pressure at outlets

B. CFD Simulationresult:

- Model-1 (Model of original geometry)
- Model-2 (Model with modified geometry at plenum & at end of runners)

INLET(m/s)	Outlet1(m/s)	Outlet 2 (m/s)	Outlet3(m/s)
32.37	10.87	25.32	28.57
33.15	9.09	25.69	29.16
35.57	12.76	25.98	31.42
35.62	12.08	26.18	32.17

Table 5.3 Model-1 results

INLET(m/s)	Outlet 1(m/s)	Outlet 2 (m/s)	Outlet 3(m/s)
32.37	25.91	32.43	36.22
33.15	26.53	33.23	37.11
35.57	28.53	35.69	39.85
35.62	28.58	35.74	39.91

Table 5.4: Model-2 results

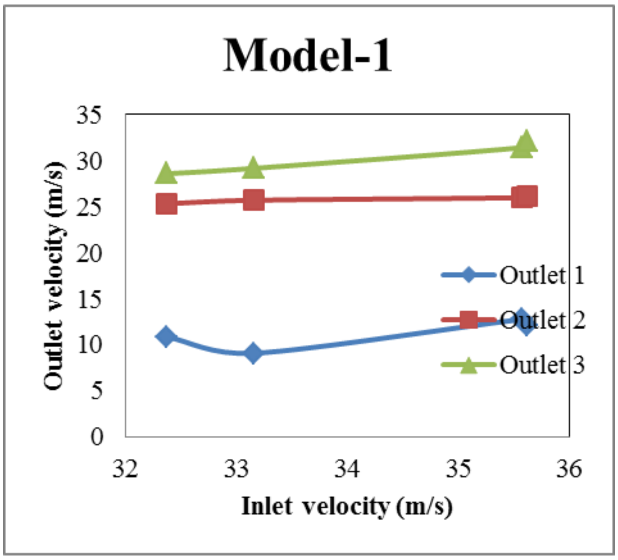

Fig. 5.3: (CFD Simulation of Model-1)

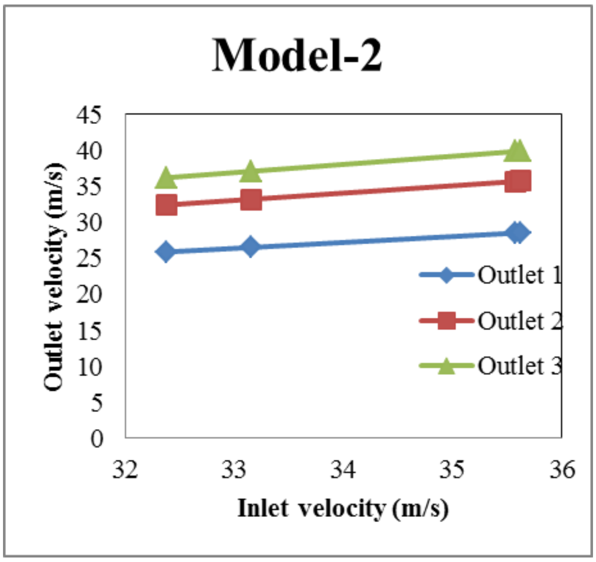

Fig.5.4: (CFD Simulation of Model-2)

C. Validation

1) CFD model validation with experimental model:

a) Experimental results:

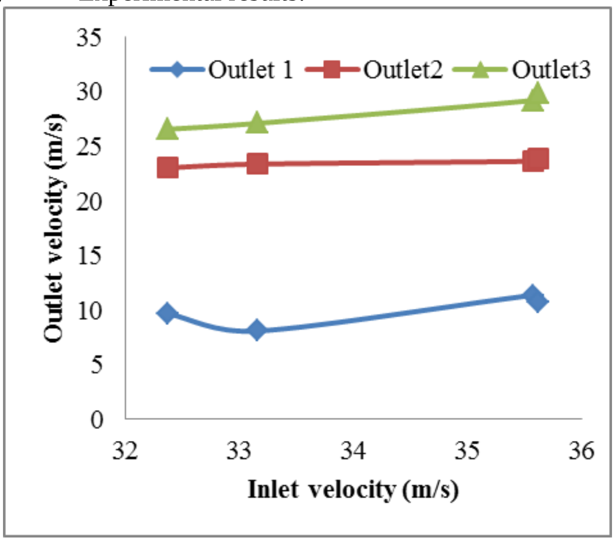

Fig.5.5: (Anemometer readings)

b) Simulation results:

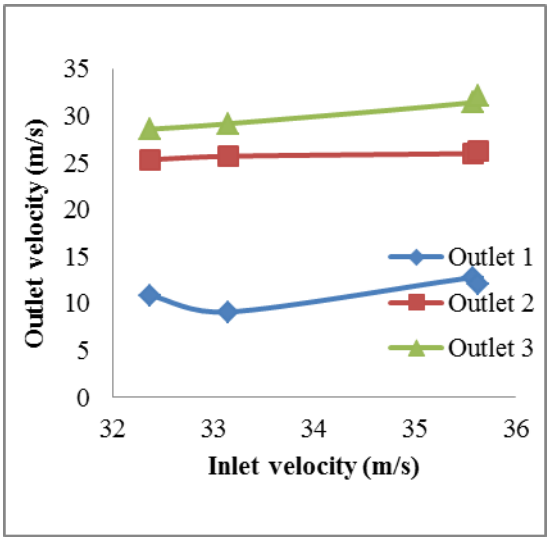

Fig.5.6: (CFD Simulation of Model-1)

The velocity at outlet-2 and outlet-3 are nearly same in both while velocity at outlet-3 is the highest one and velocity at outlet-1 is lowest in both model.

1) Outlet 1:
   At outlet 1 the actual model show lowest velocity when compared to the modified one and lowest velocity among three runners also, it shows that projection at runner 1 side has actual bad design configuration.
2) Outlet 2:
   At outlet 2 the actual model show large variation from outlet 1 to outlet 2 but when in modified model there is only small variation, it shows that projection at runner 2 side has good design configuration compared to runner 1.
3) Outlet 3:
   At outlet 3 actual model show highest velocity among these two models, the results at outlet 3 shows that may be the inside projection of depth cut at extreme side of plenum above runner 3 play significant role and help in improving results of intake manifold.

D. Proposed Geometry:

The purposed geometry has following design considerations for good results:

1) Depth cuts at extreme sides of plenum is carefully designed, depth cut at runner -1 side is of 10mm depth and runner 3 side it will same as original one of intakemanifold.
2) Curves design is good to obtain equal flow so not much alteration isrequired.
3) After all alteration geometry of good configuration obtain which giving not only nearly equal velocity in all runners but also discharge air at high velocity compare to previousdesign.

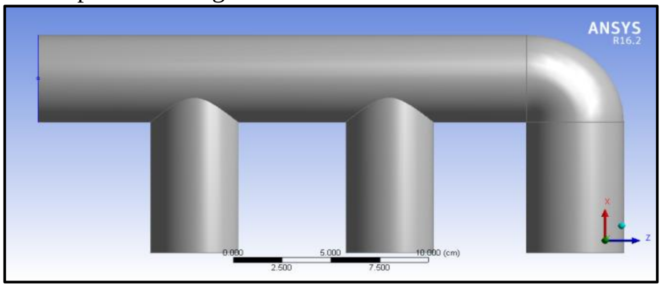

Fig.5.7: (3-D model of Proposed Geometry)

E. Comparison of result of proposed and actual intake manifold:

At inlet velocity 32.37(m/s):

	Outlet 1	Outlet 2	Outlet 3
Actual	10.879824	25.328396	28.578102
Purposed	25.914799	32.436039	36.224495

Table 5.5:

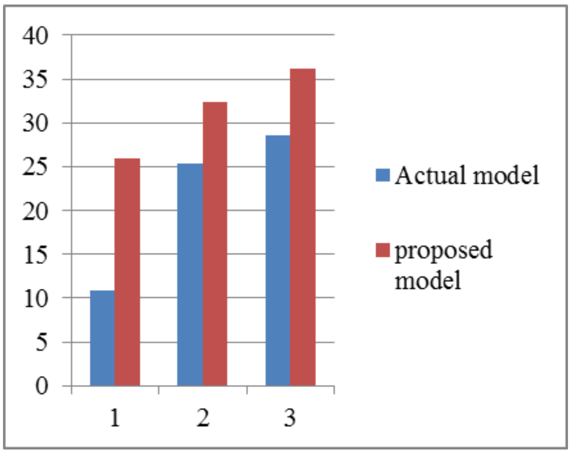

Fig.5.8: Comparison of velocity of Actual and Proposed model

Fig.5.8 show that, the velocity increases nearly equal in all three outlets of purposed geometry in comparison with original one.

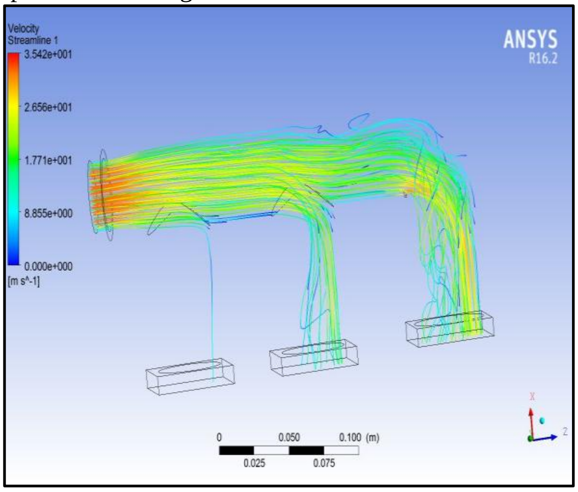

Fig.5.9: Velocity streamlines (Model-1)

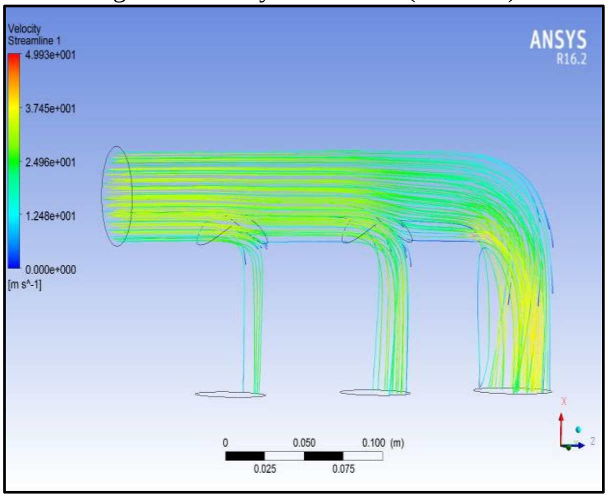

Fig.5.10: Velocity streamlines (Model-2)

VI. Conclusions

The purpose of this report is to perform an experimental and numerical analysis on intake manifold for three cylinder diesel engine. From the analysis, the following points are concluded:

1) From the experimental results it is noticed that air/charge distribution is non uniform. In particular, low

velocities i.e. lower discharge is observed in the intake manifold of first cylinder which is near to suction pump.

2) From the numerical analysis of similar kind of results are noticed. Thereby, in view of relatively more uniform distribution plenum geometry and runner outlet geometry are modified.
3) Numerical results are validated with the experimental results and it is noticed a marginal deviation of 7 % to 12 %.
4) Geometry of both plenum and runner outlets are modified and analyzed, by increasing the diameter of the plenum and changing the shape of runner outlet from square to circle.
5) As a result, relatively more uniform distribution is noticed in all the cylinders.
6) Air flow velocity not only raised in runner-1 by 42 %, but there is an enhancement of velocity in other runner outlets as well.

REFERENCES

[1] Karthikeyan S, Hariganesh R, Sathyanadan M, Krishnan S, " Computational Analysis Of Intake Manifold Design And Experimental Investigation On Diesel Engine For LCV", ISSN: 0975-5462, vol. 3 no. 4, (March2011).

[2] Benny Paul, Ganesan V, "Flow Field Development In a Direct Injection Diesel Engine With Different Manifolds", IJEST: Vol. 2, No. 1, pp. 80-91, (2010).

[3] Jemni M .A, Kantchev G, Abid M. S, "Intake Manifold Design Effect On Air Fuel Mixing And Flow For An LPG Heavy Duty Engine", IJEE:Vol. 3, Issue 1, pp.61-72, ,(2011).

[4] Min Ho Kim, Woo In Chung, In-Bum Chyun, "Three-dimensional Flow Characteristics and Engine Performance for the Geometry Modification of Intake Manifold in Multi-cylinder Diesel Engine", Seoul 2000 FISITA World Automotive Congress, June 12-15,(2000).

[5] Luo Ma-Ji, Chen Guo-Huo and MA Yuan-Hao, "Three Dimensional Transient Numerical Simulation For Intake Process In The Engine Intake Port Valve Cylinder System" , ISSN 1009 - 309.5 Journal of Zhejiang University SCIENCE V.4, No.3, PP .309-316, May- June, (2003).

[6] Yvonne S. H. Chang, Z. Yaacob and R. Mohsin, "Computational Fluid Dynamics Simulation of Injection Mixer for CNG Engines", San Francisco, U.S.A, WCECS 2007, October24-26, (2007).

[7] Kim J.N, Kim H.Y, Yoon S.S, S. D. SA, "Effect of Intake Valve Swirl on Fuel-Gas Mixing And Subsequent Combustion In a CAI Engine", Vol. 9, No. 6,pp. 649−657 ,(2008).

[8] Jae-soon Lee, Keon-Sik Yoon, "A Numerical and Experimental Study on the Optimal Design for the Intake System of the MPI Spark Ignition Engines" , KSME Journal, Vol 10, No. 4, pp.471-479,(1996).

[9] B. Murali Krishna, A. Bijucherian, and J. M. Mallikarjuna, "Effect of Intake Manifold Inclination on Intake Valve Flow Characteristics of a Single Cylinder Engine using Particle Image Velocimetry" , World Academy of Science, Engineering and Technology 68(2010).