Design of intake manifold of IC engines with improved volumetric efficiency

Suresh .Aadepu
HOD & Associate professor
Department of Mechanical Engineering, Aurobindo College of Engineering and Technology,
Ibrahimpatnam-501506, Andhra Pradesh, India
I.S.N.V.R.Prasanth
Assistant professor
Department of Mechanical Engineering, Bharat Institute of Engineering & Technology, Sagar road,
Mangalpally, Ibrahimpatnam, Ranga Reddy Dist.-501510
Jarapala.Murali Naik
Department of Mechanical Engineering, Dhruva Institute of Engineering & Technology,
JNTUH-HYDERABAD.

INTRODUCTION

In today's world, major objectives of engine designers are to achieve the twin goals of best performance and lowest possible emission levels. Excellent engine performance requires the simultaneous combination of good combustion and good engine breathing. Whilst good combustion depends only in part on the characteristics of the flow within the combustion chamber, good engine breathing is strongly affected by the unsteady flow in the intake manifold, and to a lesser extent, that in the exhaust manifold. History tells us that correctly harnessing the flow in the intake manifold of a naturally aspirated I.C. (Internal Combustion) engine can yield improvements in engine torque of 10% or more, whereas performing the equivalent in the exhaust manifold yields a more modest 3-5%.

To maximize the mass of air inducted into the cylinder during the suction stroke, the intake manifold design, which plays an important role, has to be optimized. The design becomes more complex in case of a multi-cylinder 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.

Traditional intake manifold optimization has been based on the direct testing of prototypes. This trial and error method can be effective, but expensive and time consuming. Moreover this method cannot provide any information about the actual flow structure inside the intake manifold. Without this information, the design engineer can never really understand whether a particular intake manifold performs correctly or not. One of the possible ways to obtain this information within a reasonable amount of time and cost is to conduct computational analysis. There is a need for CFD (Computational Fluid Dynamics) method which could estimate the volumetric efficiency of the engine during the design stage itself, without undergoing time consuming experiments.

OBJECTIVE

This project intends to design an Intake Manifold for an 870cc naturally aspirated diesel engine for Greaves Cotton Limited. The current manifold delivers a maximum volumetric efficiency of 84% at rated torque, i.e. 2400 RPM. The objective of the project is to achieve higher volumetric efficiency taking the space considerations into account.

In terms of function, the best plenum design would have the air duct feed the center of the plenum. Unfortunately, due to space limitations and production costs, manufacturers tend to build plenums that are fed from one end, with the plenum blocked off at the other end. This results in air rushing to the far end of the plenum and creates a slight imbalance of air flow to the individual cylinders are the air will tend to flow past the first cylinder and collect at the far end of the cylinder, which is usually at the last cylinder. The diameter and length of the intake manifold runners influence the power curve of the engine. The intake runner diameter influences the point at which peak power is reached while the intake runner length will influence the amount of power available at high and low RPM.

The criterion used to compare two manifolds is the amount of volumetric efficiency the manifolds can achieve for the particular engine. The volumetric efficiency is a misleading term, in that it refers to the mass flow of air into the engine. The volumetric efficiency takes into account the losses throughout the system from the air filter to the intake valves themselves. When the right kind of components the efficiency can actually be greater than 100% due to the addition of a super charger or turbo charger by increasing the density of the air charge going into the engine. Volumetric efficiency itself is not a constant as the values vary for various engine speeds and pipe lengths as illustrated by the jaguar D-type engine in Figure 3.3. In this figure it shows that at a higher rpm value the longer pipes will actually hinder the efficiency of the engine leading to a poor performance curve. This phenomenon has led to the development of a new intake technology that involves manifold folding. Here the intake is able to vary its length based on engine rpm, by doing so the power band is maintained and an increase in the overall performance of the engine is gained. In order to create more horsepower the runners become shorter in length allowing the air a more direct path, and when torque is desired the pipe length is extended though the use of a valve to control the flow through a separate set of tubing.

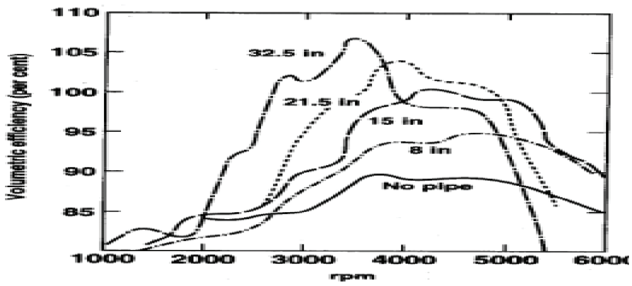


Figure2.1: Volumetric efficiency Vs RPM's for different pipe lengths for a Jaguar D-type engine (for factory built race car).

2.1 Project objective

To design an intake manifold for a twin-cylinder CI engine and to suggest necessary design modifications for improving the flow behavior.

2.2 Methodology adopted to meet the objective

This is 3-D computational fluid dynamics (CFD) method

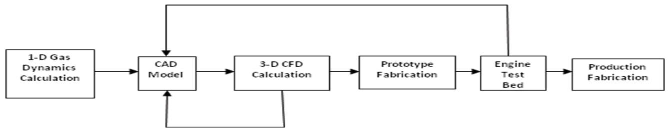


Figure 2.2: A typical process chain for design of intake manifold

2.2.1 Theory behind CFD

Applying the fundamental laws of mechanics to a fluid gives the governing equations for a fluid. The conservation of mass equation is

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{V}) = 0$$

and the conservation of momentum equation is

$$\rho \frac{\partial \vec{V}}{\partial t} + \rho(\vec{V} \cdot \nabla)\vec{V} = -\nabla p + \rho \vec{g} + \nabla \cdot \tau_{ij}$$

These equations along with the conservation of energy equation form a set of coupled, nonlinear partial differential equations. It is not possible to solve these equations analytically for most engineering problems.

However, it is possible to obtain approximate computer-based solutions to the governing equations for a variety of engineering problems. This is the subject matter of Computational Fluid Dynamics (CFD).

COMPUTED CASES

Different intake manifold configurations were tested. By saying different configurations, it means that we can change the five variables, i.e. primary pipe runner and diameter, plenum volume, and secondary pipe length and plenum volume. But the primary pipe diameter cannot be changed as it is same as port outlet and the secondary pipe too cannot be modified as it connects air cleaner. Hence, the primary pipe length, plenum volume and secondary pipe length as used as variables.

 ISSN No: 2348-4845

Initially, the manifold configurations were optimized at 2400RPM for each variant as the highest volumetric efficiency is found to occur at that rpm. Then the speed characteristics were computed using these optimized configurations for engine speed range from 1600RPM to 3600RPM.

The optimized manifold for this particular engine has been found with the below given dimensions. The plenum volume has been increased to 514cc from the present 390cc. Also the optimum runner length is found to be 150 mm. The following table and graph compares the current manifold with the optimized one.

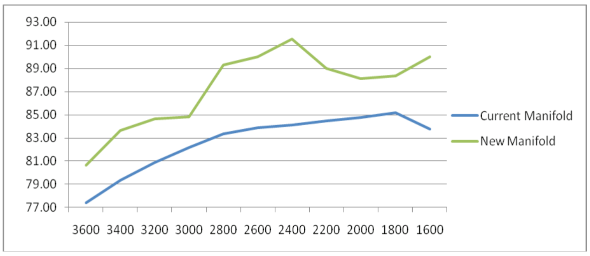


Figure 3.1: Graph plotted between volumetric efficiency and RPM for current and new manifolds.

Table 3.1: Comparison between the existing manifold and the new optimized manifold

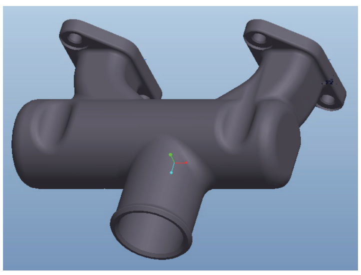
		Current manifold	New Manifold	Efficiency Improvement
Volumetric Efficiency	Speeds (RPM)	Volumetric Efficiency (%)	Volumetric Efficiency (%)	
	3600	77.42	80.67	3.25
	3400	79.36	83.63	4.27
	3200	80.88	84.66	3.79
	3000	82.20	84.81	2.61
	2800	83.38	89.28	5.90
	2600	83.91	90.03	6.13
	2400	84.14	91.55	7.41
	2200	84.45	88.99	4.54
	2000	84.76	88.14	3.38
	1800	85.20	88.36	3.16
	1600	83.79	90.01	6.22
Intake Manifold Dimensions	Primary Pipe dia, mm	35.00	35.00	
	Primary pipe length, mm	51.00	150.00	
	Plenum Dia, mm	52.00	60.00	
	Plenum Length, mm	184.00	182.00	
	Plenum Volume, cc	390.76	514.59	
	Secondary pipe Dia, mm	45.00	45.00	
	Secondary pipe length, mm	52.00	120.00	
	Notes	For the first cylinder, the end is 56mm offset from cylinder axis and in the 2nd cylinder, it is 24mm.	Equal length on both sides i.e. 24mm	

3-D ANALYSIS OF THE MANIFOLD

From the manifold dimensions achieved form GT-power software, several shapes of manifold can be devoloped. Two different manifolds are proposed using the configurations. These manifolds are modeled using Pro-Engineer (Pro-E) software. For the 3-D analysis of the manifold, the CFD domain is extracted from the models.

4.1 Current Manifold

The current manifold that is used in the current 870cc BS-II engine is asymmetric in nature. The CFD domain in the manifold was obtained using the cut-out feature in Pro-E. Figure 6.1 shows the physical model of the manifold and its CFD domain. CFD domain involves the volume of space in components whose analysis is required.



Subfigure 6.1.1: The Pro-E model of the current manifold used in the 870cc engine.

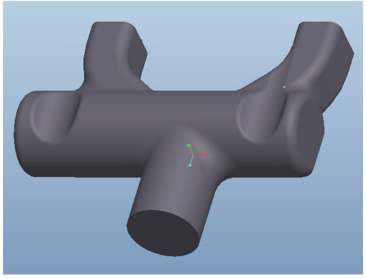


Figure 6.1: The current manifold model and its CFD domain.

4.2 Basis for Geometry Creation

A new manifold needs to be designed using the dimensions given in table 5.1. Two such configurations have been made as shown the figures 6.2 and 6.4. The first manifold is made such that the axis of primary and secondary pipes lies on the same plane while the second manifold is constructed otherwise. This gives the opportunity to observe the flow motion in each of the cases. The fillets on the manifold are chosen randomly. The CFD domains of these manifolds are shown in figures 6.3 and 6.4 respectively. The first manifold is designated "manifold model 1" and the second as "manifold model 2".

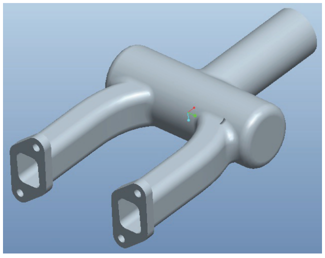


Figure 4.2: Pro-E model of manifold model 1

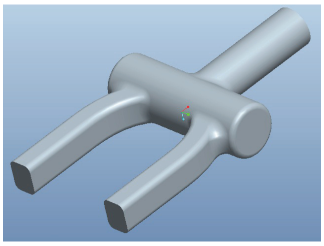


Figure 4.3: Pro-E model of CFD domain of manifold model1

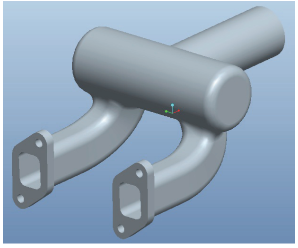


Figure 4.4: Pro-E model of manifold model 2

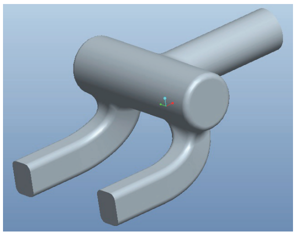


Figure4.5: Pro-E model of CFD domain of manifold model 2

CFD Analysis

5.1.1 Steady-State Flow Assumption

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.

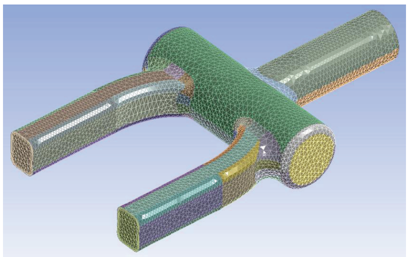


Figure 5.1: Mesh of manifold model 1

The mesh statistics:

Total number of nodes – 54033

No. of tetrahedra elements-106369

Number of pyramid elements – 0

No. of prism elements – 64410

Total no of elements – 170779

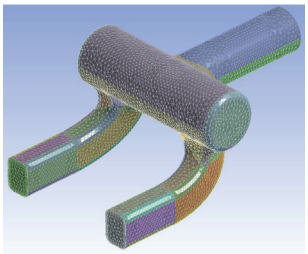


Figure5.2: Mesh of manifold model 2

The mesh statistics:

Total number of nodes 55637

No. of tetrahedra elements- 106893

No. Of pyramid elements- 23

No. of prism elements - 67145

Total no of elements - 174061

5.2.1 Boundary Conditions

Air was used as fluid media, which was assumed to be steady and incompressible. Measurements have been taken out from GT-Power for providing the necessary boundary conditions to the prediction. Boundary conditions are taken at two points in the crank rotation, first where mass flow rate is maximum in first runner and second where mass flow rate is maximum in second runner. Since the best volumetric condition of the engine occurs near 2400RPM, boundary conditions are taken at this speed.

Table 5.2: Boundary conditions for the manifold models

Location	Boundary condition	Crank angle	
		448.83	-83.35
Inlet of manifold	Pressures (bar)	0.991	0.980
Exit of runner to Cylinder 1	mass flow rate, g/s	60.88	0.45
Exit of runner to Cylinder 2	mass flow rate, g/s	-0.27	62.32

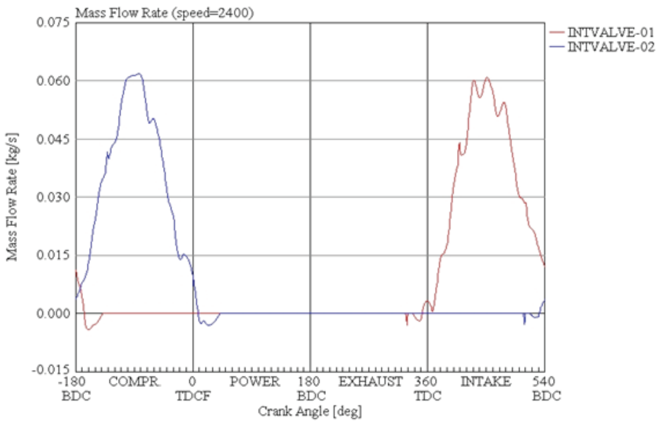


Figure 5.2: Mass flow rates for the two runners with respect to crank angle from the 1-D GT-Power code.

At the inlet of the plenum chamber, fixed pressure rates are taken and at the exit of the plenum chamber, fixed mass flow condition were given corresponding to the crank angle as boundary conditions. A turbulence intensity of 5% was presented to estimate the inlet boundary condition on turbulent kinetic energy *k* and its dissipation rate *e*. This turbulence model is widely used in industrial applications. No slip boundary condition was applied on all wall surfaces. Whole domain was considered at 1 atm and at 300 K as initial condition. Other fluid properties were taken as constants.

5.3.1Turbulence Model

The CFD analyses were conducted using the regular k-e turbulent model, and a first order upwind difference scheme. This model employs two additional transport equations one for turbulence kinetic energy (k) and another one for the dissipation rate (e).These solver settings guaranteed existence of a fully converged solution for each set of parameters. The optimization technique required high CFD accuracy for the evaluation of objective and constrain functions to compute the sensitivity for each variable. In order to keep the analysis at a manageable cost; the CFD mesh was kept relatively coarse, namely around 170000 cells defining the flow domain. Near wall treatment is handled through generalized wall functions. This model is well established and the most widely validated turbulence model.

6. RESULTS AND DISCUSSIONS

Table 6.1: Results of CFD analysis

Model and description	Inlet of manifold		Exit of runner 1		Exit of runner 2		Pressure Drop (Pa)
	Pressure (Pa)	Velocity (m/s)	Pressure (Pa)	Velocity (m/s)	Pressure (Pa)	Velocity (m/s)	
Manifold model 1 when first runner has maximum flow rate	98310	32.37	96810	53.95	N/A	N/A	1500
Manifold model 1 when second runner has maximum flow rate	97270	33.15	N/A	N/A	95690	55.25	1580
Manifold model 2 when first runner has maximum flow rate	98740	35.57	96310	60.93	N/A	N/A	2430
Manifold model 2 when second runner has maximum flow rate	97650	35.62	N/A	N/A	95020	58.17	2630

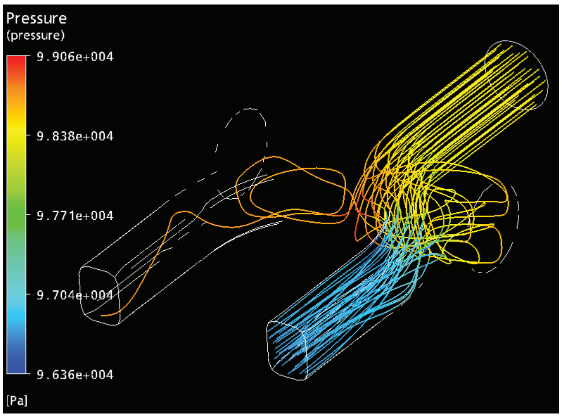


Figure 6.1: Pressure streamlines in manifold model 1 when first runner has maximum flow rate

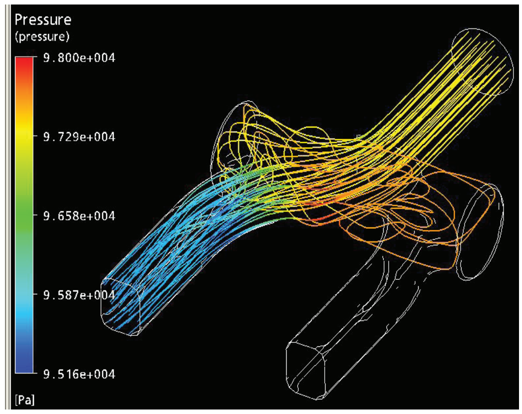


Figure 6.3: Pressure streamlines in manifold model 1 when second runner has maximum flow rate

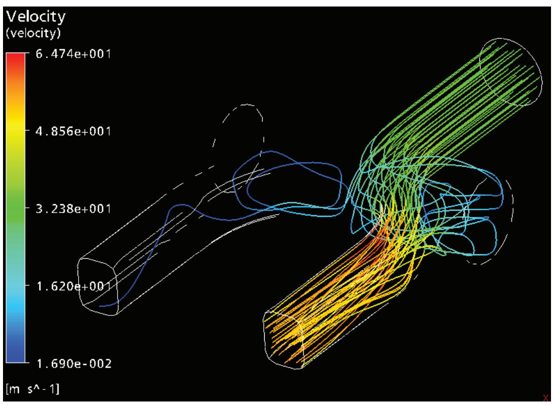


Figure 6.2: Velocity streamlines in manifold model 1 when first runner has maximum flow rate

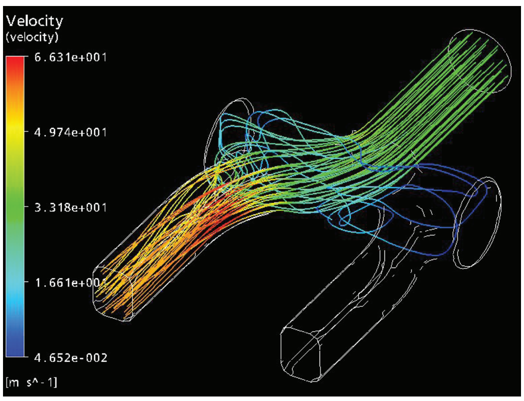


Figure 6.4: Velocity streamlines in manifold model 1 when second runner has maximum flow rate

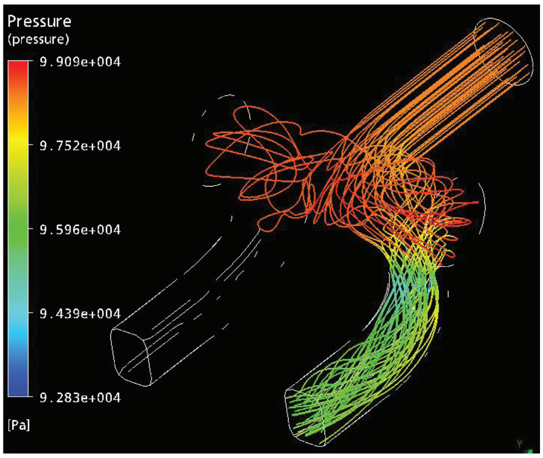


Figure 6.5: Pressure streamlines in manifold model 2 when first runner has maximum flow rate

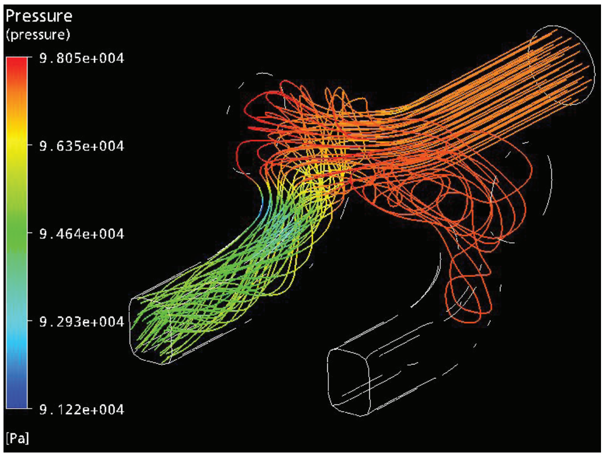


Figure 6.7: Pressure streamlines in manifold model 2 when second runner has maximum flow rate

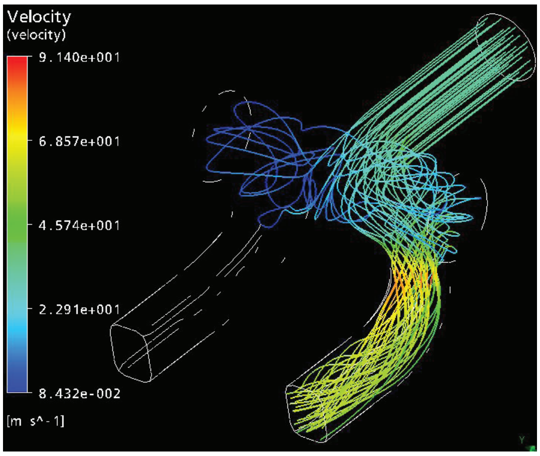


Figure 6.6: Velocity streamlines in manifold model 1 when first runner has maximum flow rate

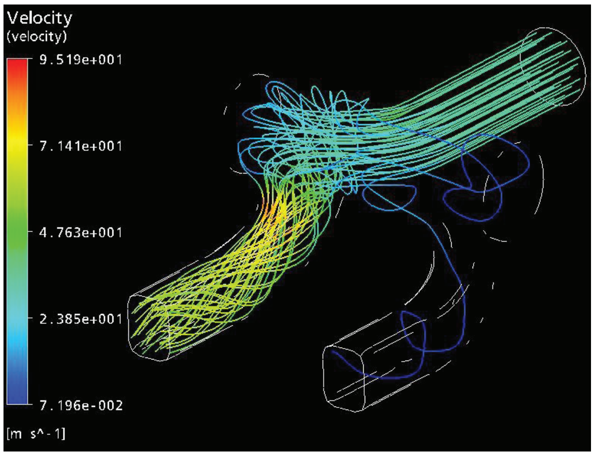


Figure 6.8: Velocity streamlines in manifold model 2 when second runner has maximum flow rate

From the results shown in the table 7.1 and in the figures 7.1 to 7.8, we can arrive at the following statements:

The pressure drop in the manifold model 2 is more than in manifold model 1.

The pressure drop in fluid flow is found to be more when the flow is through runner 2 than when compared to runner 1 in both manifold models 1 and 2. This difference can be reduced by giving smoother bends for runner 2.

The velocities in the manifold model 2 are more than in manifold model 1.

The flow in the runners in manifold model 2 is found to aid more swirl. This is beneficial because it aids better mixing of fuel and air in combustion chamber.

Manifold model 2 offer better space savings than manifold model 1.

Since manifold model delivers better velocities of fluid flow and offers space reduction, manifold model 2 is preferred over the manifold model 1.

CONCLUSIONS

A methodology for design of intake manifold of IC engines with improved volumetric efficiency has been presented. This methodology combines optimization using 1-D engine simulation software and three-dimensional, steady state CFD technique, rather than experimental comparison. The 1-D software served as a platform to obtain the configuration of the manifold which gives better volumetric efficiency, while the CFD simulations enabled to visualize the flow within the manifold. Such simulations give better insight of flow within the manifolds. Design modifications could be done using CFD simulations to enhance the overall performance of the system. Using this method, a better design of manifold giving 7% increase in volumetric efficiency could be achieved.

For better results, unsteady state analysis can be carried out to predict how the intake manifold performs under real conditions. The boundary conditions for these can be obtained form 1-D software. The joints can also be made smoother to reduce pressure losses, but the effect of increase in plenum volume has to be correlated with 1-D software. Also if some changes to the plenum perform in order to guide the flow to the runners by the geometry of the plenum, the performance of intake manifold will be improved

REFERENCES

Heywood, J .B., (1988), "Internal Combustion Engine Fundamentals," McGraw-Hill International Editions, pp. 309-321, ISBN 0-07-100499-8.

Rozsas.T., & Brandsteller,W.,(1988), "Optimization of Charge Process in Modern Motor Vehicle Spark Ignition Engines", SAE 885058.

Safari, M., Ghamari, M. and Nasiritosi, A., (2003) "Intake manifold optimization by using 3-D CFD analysis", SAE 2003-32-0073.

Bensler, H.,(2002), "Intake manifold optimization by using CFD analysis", Volkswagen AG, Powertrain Calculations Dept.

Kale, S.C., and Ganesan, V., (2004) "Investigation of the flow field in the various regions of intake manifold of a S.I. engine," Indian journals of Engineering and Material Sciences. pp 134-148.

Makgata, K.W, (2005) "Computational analysis and optimization of the inlet system of a high-performance rally engine", University of Pretoria.

Kale, S.C., and Ganesan, V., (2004) "Investigation of the flow field in the various regions of intake manifold of a S.I. engine," Indian journals of Engineering and Material Sciences. pp 134-148.

Trigui,N., Griaznov,V., Affes,H., and Smith,D.,(1999) "CFD Based Shape Optimization of IC Engine", Oil & Gas Science and Technology – Rev. IFP, Vol. 54 (1999), No. 2, pp. 297-307.

Ravinder Yerram and Nagendra Prasad, (2006)" Optimization of Intake System and Filter of an Automobile using CFD analysis", SAE