Proceedings of the International Conference on Applied Mechanical Engineering Research (IC-AMER2019)
NIT Warangal, India – May 02-05, 2019
Paper No. XXX (The number assigned when the papaer is accepted)

Design, Development and Analysis of Intake Manifold of Single Cylinder Diesel Engine

Nikhil A Bhave[a*], Mohan Trivedi[a], Sandeep Joshi[a] Neeraj Sunheriya[b] Mahendra M Gupta[a]
[a]*Department of Mechanical Engineering ,Shri Ramdeobaba College of Engineering and Management, Nagpur 440013 Maharashtra, India*
[b]*Research Scholar, National Institute of Technology, Raipur 492013 Maharashtra, India*
*Corresponding author Email: bhavena1@rknec.edu

Engines are one of the greatest Mechanical Engineering application developed in 90s. Compression ignition engines are very robust, durable and efficient. The volumetric efficiency of CI engine is higher because of absence of throttle losses in comparison to SI engine. The flow of air through the intake manifold considerably affects the Power and Volumetric Efficiency of CI engines. In present investigation, existing intake manifold of CI engine is modified and manufactured by Additive Manufacturing Technique. The optimised design was finalised by undergoing Computational Fluid Dynamic Analysis. The new intake manifold is fitted to the engine intake and performance tests were performed. For all load conditions, the Volumetric Efficiency, Brake Power and Brake Thermal Efficiency were considerably improved. The Brake Specific Fuel Consumption was also reduced.

***Keywords*:** Intake Manifold, CI engine, CFD Analysis

1. Introduction

The air flow through the Engine Intake System greatly affects the performance of CI engine. The components involved in the gas exchange process are Intake Valve Opening and Closing Area, Dimensions of the intake and exhaust valves, Number of Valves per cylinder, Location of fuel injector, Intake and exhaust port geometry and Intake and Exhaust Manifold design. Small variations in any one of the above parameters can significantly affect the performance of engine. In the present investigation, the design of intake manifold is being considered[1].

2. Experimental Set-up details and Methodology

In the present investigation a stationary, single cylinder, 4-stroke, 5 BHP /1500 rpm, water cooled, direct injection (DI) Comet Diesel engine was used. The engine is coupled with a rope brake dynamometer. The brake drum can be loaded by maximum of 12Kg and Spring balance is attached to the other end. A calorimeter is installed to the exhaust manifold which is a concentric tube heat exchanger. The exhaust gases are cooled by cooling water through the heat exchanger. The dynamometer is continuously cooled by supply of cold water. Cooling water is circulated through the jackets to cool the engine. An air box fitted with an orifice plate is installed at the inlet air side to measure the air flow rate, while the calibrated burette is present for fuel measurement.

The engine is also equipped with power transducer for load measurement, piezo sensor and amplifier for pressure measurement, a crank angle encoder, water flow meter, RPM indicator and fuel flow sensor. Thermocouples are attached at intake and exhaust port, calorimeter inlet and outlet of water and exhaust gas and ambient cooling water inlet and outlet. A High Speed Data Acquisition System collects the data signal and send it to a computer equipped with an INTEL i7 core to duo processor. A Powerful software for studying different performance parameters of the engines such as BP, BSFC, A:F, BSEC and so on is also installed in the computer.

The design of existing engine manifold is as shown in the Figure5. Previously the overall length of the manifold was 160mm. It was having a 90˚ bend in the middle section. The bend is responsible for energy loss. Loss in energy results in decrease in air flow through the intake manifold. This results in lesser volumetric efficiency and power.

The aim of this research work is to design an intake manifold which will increase the power and efficiency of the engine. The design was to be optimised based on the overall length and the angle of bend. Thus, ANSYS 14 FLUENT, was used to perform CFD simulation of fluid flow through the pipe. Firstly the bend was considered with angles 90º, 60º, 30º, 15º, 0º. Secondly the further consideration was made to create a converging-diverging nozzle which will accelerate the flow at the outlet of manifold. Thirdly the manifold was manufactured by additive manufacturing technique as shown in Figure 8. The experimentation was performed on Kirloskar AV1 direct injection CI engine. The Air flow rate, Fuel Flow rate, Engine RPM and Brake Power were measured by flow anemometer, burette method, shaft encoder and rope brake dynamometer respectively.

3. Results and Discussions

The flow through the pipe has Mach number less than 0.3. Therefore the flow is considered to be incompressible. The roughness of the pipe was considered negligible and flow condition is considered as inviscid. The value of velocity was selected corresponding to mass flow rate calculated experimentally. At inlet to the manifold, the constant velocity of 7 m/sec was considered for all cases of calculations. The energy equation was solved for all cases and the air was considered as real gas. Standard initialisation with calculation from inlet was performed by ANSYS Fluent.

In the first case, the pressure drop went on decreasing with the increase in angle as seen in the Figure 1, 2 , 3, 4, 5. If the Bernoulli's Equation is applied to the pipe flow, the head loss due to friction is very small. However the head loss due to velocity component in bend is higher. The k value for [$k(0.45 \text{ to } 4.5)V^2/2g$] was selected based on the bend angle. The calculation of pressure drop is in the range of 0 to 10 Pa. The simulations results are close to analytical results performed. The value of pressure drop seems to be very small since the length is small. The details have been presented in the Table1.

Table 1: Variation of Pressure Drop and Velocity with Angle of Bend

Angle of bend	Pressure Drop(Pascal)	Exit Velocity(m/s)
90	5.975	6.93
60	2.239	6.99
30	0.605	7.17
15	0.001	7.07
0	0.000001	6.999999

It was inferred that a straight pipe will definitely increase the volumetric efficiency of the engine [2]. The acceleration of the charge can further enhance the efficiency of the engine. So another consideration was performed to model a nozzle with same length of pipe. Various design of converging- diverging nozzle was done by varying the convergence length and throat diameter. The convergence length was varied from 40 mm to 75mm while the throat diameter was varied from 12mm to 20mm such what the nozzle angle doesn't exceed 6°. Simulations were performed and results for pressure drop and exit velocity is presented in the Table 2.

Table 2: Variation of Pressure Drop and Velocity with different geometry

Length of Converging Section	Diameter of Pipe in mm (D1)	Diameter of Throat in mm (D2)	Divergence Angle	Pressure Drop in Pascal	Velocity at Exit in m/s
40	32	12	4.764	72	1.31 to 15.31
	32	16	3.814	-65.94	0.31 to 9.38
	32	20	2.862	-240.35	1.89 to 5.36

45	32	12	4.97	-40.18	8.49 to 9.66
	32	16	3.979	183.28	5.44 to 6.96
	32	20	2.987	21.94	1.34 to 12.64
50	32	12	5.194	406.27	3.78 to 9.16
	32	16	4.16	-67.88	7.43 to 8.55
	32	20	3.122	79.31	5.31 to 6.36
55	32	12	5.44	354.42	0.971 to 10.57
	32	16	4.357	-85.12	3.05 to 10.51
	32	20	3.27	102.81	7.24 to 8.04
60	32	12	5.711	281.15	0.21 to 7.43
	32	16	4.574	0.059	1.012 to 12.29
	32	20	3.434	106.93	6.56 to 11.71
70	32	12	6.34	295.98	0.375 to 7.37
75	32	12	6.72	-333.77	3.56 to 21.74
162	45	32	-	80	13.47 to 14.30
162	50	32	-	135	16.37 to 17.50

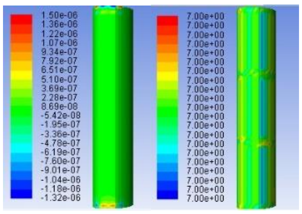


Figure 1:Pressure and Velocity Profile for 0° Bend

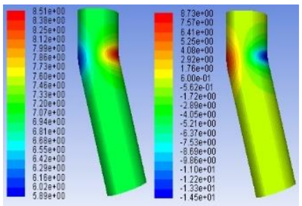


Figure 2:Pressure and Velocity Profile for 15° Bend

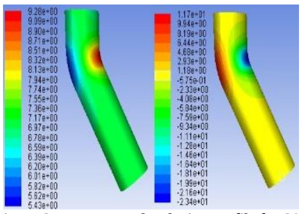


Figure 3:Pressure and Velocity Profile for 30° Bend

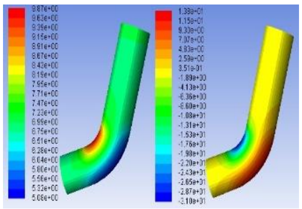


Figure 4:Pressure and Velocity Profile for 60° Bend

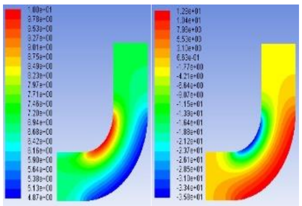


Figure 5:Pressure and Velocity Profile for 90° Bend

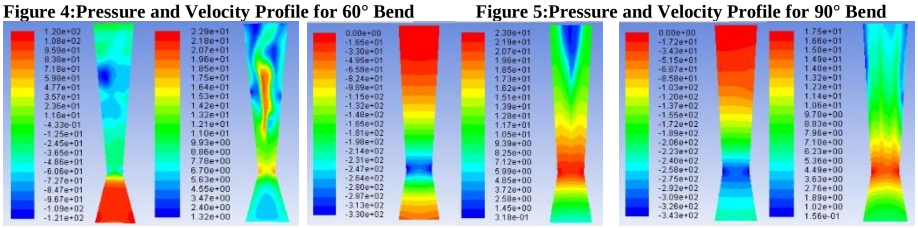


Figure 6:Pressure and Velocity Profile for 40-12, 40-16, 40-20

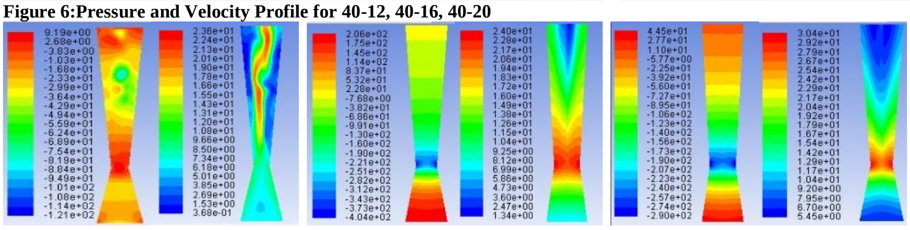


Figure 7:Pressure and Velocity Profile for 45-12, 45-16, 45-20

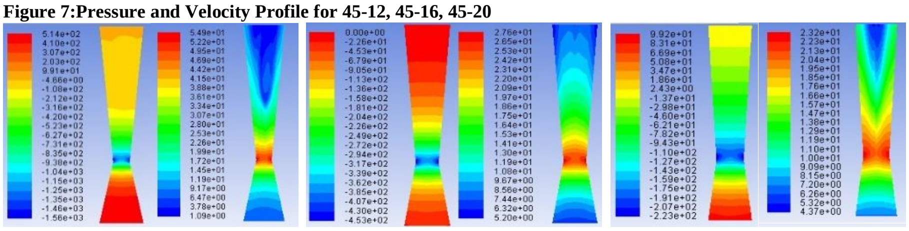


Figure 8:Pressure and Velocity Profile for 50-12, 50-16, 50-20

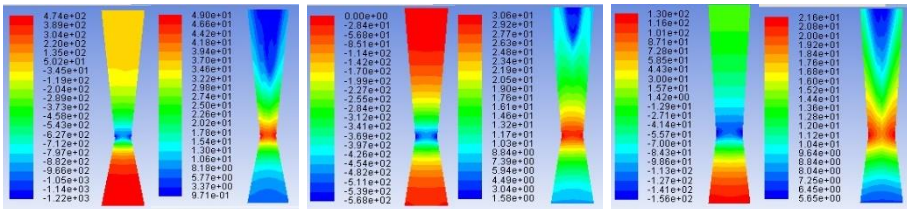


Figure 9:Pressure and Velocity Profile for 55-12, 55-16, 55-20

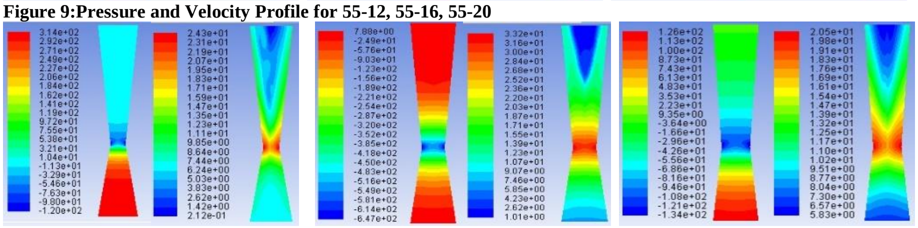


Figure 10:Pressure and Velocity Profile for 60-12, 60-16, 60-20

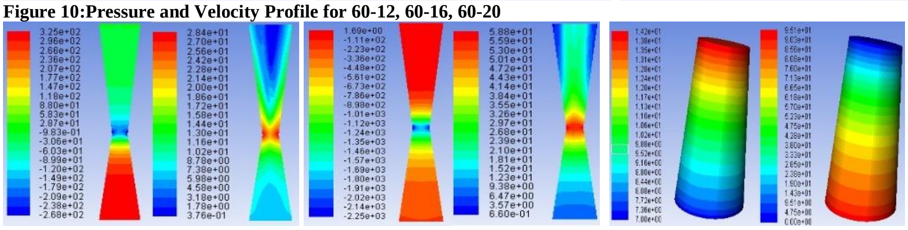


Figure 6:Pressure and Velocity Profile for 70-12, 75-12, 45-32

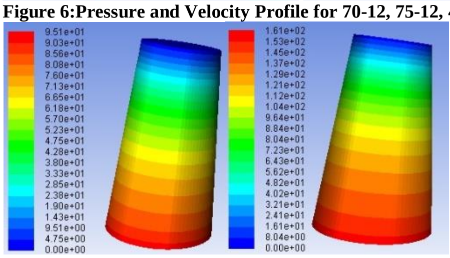


Figure 7:Pressure and Velocity Profile for 50-32

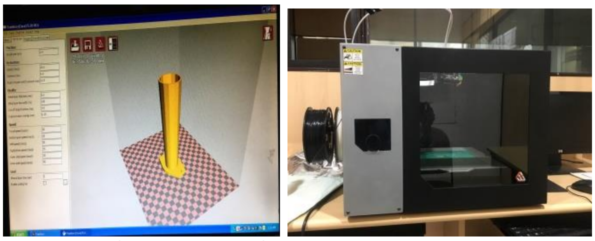
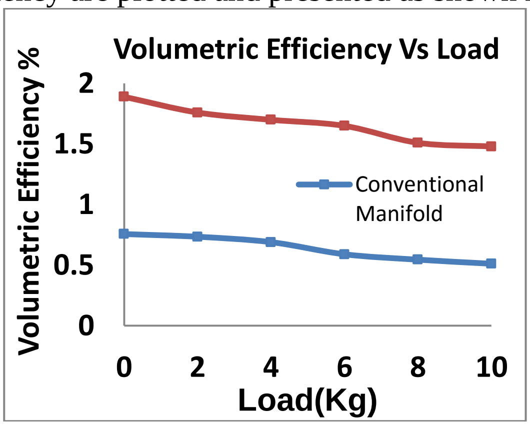


Figure 8: GUI for Additive Manifacturing and 3D Printer

Following are the observations from the above CFD simulation.

1) As the converging length was increased, the pressure drop is found to be less and uniform over the length of nozzle.
2) Formation of Turbulent Eddies at the exit to diverging length is more if the throat diameter is small.
3) If the throat diameter is half of the exit diameter, the least pressure obtained is exactly at the throat. The value of velocity obtained in most cases is high in this case.
4) Instead of converging-diverging nozzle, only a converging nozzle can be the most appropriate solution for the present case. The 50-32 Nozzle gives exit velocity of 17 m/sec by analytical and CFD simulation.

The nozzle (50-32) was manufactured by using additive manufacturing technique and fitted to the engine and the performance test was conducted before and after the use of modified manifold. The variation of Volumetric Efficiency, Brake Specific Fuel Consumption and Brake Thermal Efficiency are plotted and presented as shown in the Figure 9, Figure 10 and Figure 11.



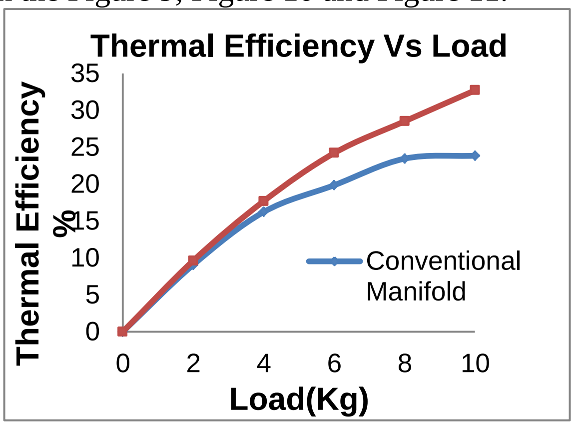


Figure 9: Volumetric Efficiency Vs Load

Figure 10: Thermal Efficiency Vs Load

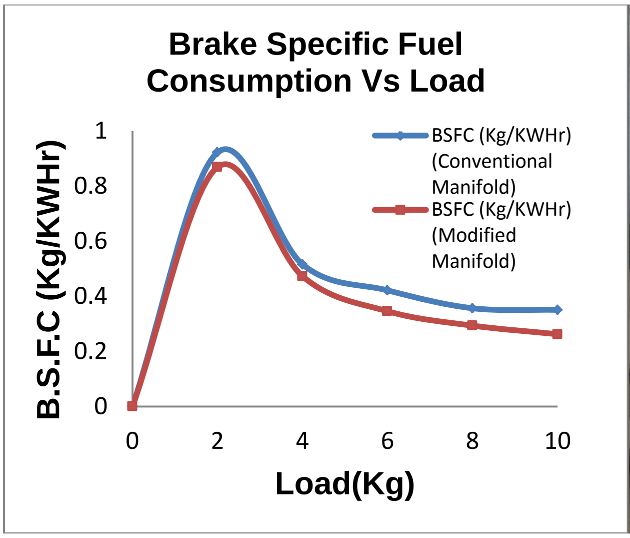


Figure 11: BSFC Vs Load

Figure 12: Experimental Set-up

4. Conclusions:

1) A small change in dimension of intake manifold can bring about huge change in the performance of the engine
2) Not much work is available on the co-relations for prediction of nozzle performance based on converging length, diverging length, throat diameter and exit diameter. CFD analysis is accurate to predict the pressure and velocity variations accurately.
3) The intake manifold modified into 50-32 Nozzle resulted in considerable increase in Volumetric and Thermal Efficiency by approx. 65.38% and 57.92% for full load. The fuel consumption was greatly reduced due to better combustion.

References

1. J. B. Heywood, Internal Combustion Engines Fundamentals, Edition 2, McGraw Hill Publishers, India (2017)
2. Mohd Faisal Hushim, Ahmad Jais Alimin, Mohd Azahari Razali, Akmal Nizam Mohammed, Azwan Sapit, Julio Cesar Mendez Carvajal. ARPN Journal of Engineering and Applied Sciences. 10,2006
3. Giovanni Vichi, Luca Romani, Lorenzo Ferrari, Giovanni Ferrara. ATI2014 Proceedings of the 69th Conference of the Italian Thermal Machines Engineering Association 2014. Energy Procedia, 2015
4. Chitnis P P, Juttu S, Deshmukh A G, Sampathkumar K R, and Marathe N V. IMECE2010-37154 Proceedings of the ASME 2010 International Mechanical Engineering Congress & Exposition IMECE2010 November 12-18, 2010, Vancouver, British Columbia, Canada, 2010