Design & Optimization of Intake Manifold of Carburetor Using Computational Fluid Dynamics

Mukesh Kumar Verma, Devvrat Verma, Annand Shukla

***Abstract*— intake manifold is the part of an engine that supplies the fuel/air mixture to the cylinder. the intake manifold is essential for the optimal performance of an internal combustion engines. Intake manifold is the breathing system of the car engine which supplies air to the engine cylinders where the combustion of the fuel occurs.**

The primary function of the intake manifold is to evenly distribute the combustion mixture (or just air in a direct injection engine) to each intake port in the cylinder head. Even distribution is important to optimize the efficiency and performance of the engine. It may also serve as a mount for the carburetor, throttle body, fuel injectors and other components of the engine.

Due to the downward movement of the pistons and the restriction caused by the throttle valve, in a reciprocating spark ignition piston engine, a partial vacuum (lower than atmospheric pressure) exists in the intake manifold.

This review paper has main aim is the gain understanding of the flow characteristics inside an intake manifold which is fitted with an intake restrictor. Geometrical design of intake manifold is very important for the good performance of an IC engine. Unequal velocity distribution of intake air of runner's outlets of intake manifold makes it less efficient. The reported works aims to makes this unequal velocity distribution of velocity in nearly equal manner with increase of velocity at outlet without any major modification in design of intake manifold. To achieve the desired improve result two other models of same intake manifold with different design configuration are made in software then examine the results of this two models with original one to find out pressure and velocity losses. After analysis of models it noticed that the hidden projections of nut, projected stiffeners and depth cuts o extreme of plenum causes pressure loses due to which uneven distribution takes place at runner's outlets.

***Index Terms*— intake manifold, CI engine, plenum, restrictor, flow analysis, piston, fuel/air mixture, computational fluid dynamics.**

I. INTRODUCTION

S. karthikeyan et al (1) in this paper, pressure waves for the intake manifold is simulated using 1D AVL- Boost software, to study the internal air flow characteristics for 3- cylinder diesel engine during transient conditions. Based on the 1D simulation results, the intake manifolds design I optimized using 3D CFD software under steady state condition.

Benny Paul et al (2), observed the effect helical, spiral, and helical-spiral combination manifold configuration on air motion and turbulence inside the cylinder of a direct injection diesel engine motored at 3000 rpm. By using the CFD tool, they compared predicted CFD results of mean swirl velocity of the engine at different locations inside the combustion chamber at the end of compression. After the analysis they notice various things like, the helical-spiral manifold geometry creates higher velocity component inside the combustion chamber at the end of compression stroke. Swirl ratio inside the cylinder and turbulent kinetic energy are higher for spiral manifold. Volumetric efficiency for spiral-helical combined manifold is 10% higher than that of spiral manifold. Conclusion of results shows that helical spiral combined manifold creates higher swirl inside the cylinder than spiral manifold.

M.A. Jemni Et. al. (3) presents an investigation of mixture preparation in the intake manifold of a diesel engine into LPG spark- ignition engine operation. Two manifolds shapes are used in order to test the adequate design in view of flow and air gas homogenization. The first is designed according the acoustic- filling phenomenon, and the second present an unspecified design. The model of simulation done by solving **Navier- stokes** and energy equation in conjunction with the standard K-e turbulence model, using 3D CFD code flow works.

Min-HO Kim et al. (4) in this, the internal flow characteristics in the intake manifold of a six cylinder diesel engine are investigated computationally for the variation of spacer and chamber with under steady state. Analysis result of relative air distribution, rate of all ports was in the range of approximately 14%- 19%, and the air distribution rat into port no. 4 was the least one. Model with spacer is more efficient than the other model without spacer. In the case of the engine performance test, with regards to the fuel consumption rate and smoke at low speed, the case of the model without the spacer decreased more than the model with the spacer. But at high speed, it shows a tendency to increase, contrary to the finding at low speed. In case of the model with the spacer, as the chamber width increased at low speed, the fuel consumption and smoke level decreased, but increased at high speed.

Luo Ma-ji et al. (5) this paper presents a KIVA-3 code based numerical model for 3D transient intake flow in the intake port valve cylinder system of internal combustion engine using body- fitted technique, which can be used in numerical study on internal combustion engine with vertical and inclined valve, and has higher calculation precision. A numerical simulation (on the intake process of a two valve engine with a semi sphere combustion chamber and a radial intake port) is provided for analysis of the velocity field and pressure field of different plane at different crank angles. The results revealed the formation of the tumble motion, the evolution of flow field parameters and the variation of tumble ratios as important information for the design of engine intake system.

II. METHODOLOGY

Continuity equation:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u_i)}{\partial x_i} = 0$$

Momentum equation:

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_i u_j)}{\partial x_i} = -\frac{\partial p}{\partial x_i} + \frac{\partial \sigma_{ij}}{\partial x_i}$$

According to stokes's hypothesis which assumes that the bulk velocity can be neglected, the shear stress tensor for a Newtonian fluid is given by:

$$\sigma_{ij} = 2\mu(T)S_{ij} - \frac{2}{3}\mu(T)S_{kk}\,\delta_{ij}$$

Energy equation:

$$C_p\left[\frac{\partial(\rho T)}{\partial t} + \frac{\partial(\rho T u_j)}{\partial x_i}\right] = \frac{\partial p}{\partial t} + u_j\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_i}\left(K\frac{\partial T}{\partial x_i}\right) + \Phi$$

Where the viscous dissipation Φ is defined as:

$$\Phi = \sigma_{ij}\frac{\partial u_i}{\partial x_i}$$

Bernoulli's equation:

$$\frac{P_1}{\rho_1 g} + \frac{V_1^2}{2g} + Z_1 = \frac{P_2}{\rho_2 g} + \frac{V_2^2}{2g} + Z_2 + losses$$

Hydrostatic law:

Pressure (P) = ρ×g×h

ρ= Density
g = acceleration due to gravity
h = difference in mercury level in U-tube manometer

III. MESHING

The accuracy of the results depends highly upon the mesh quality. Thus the choice of meshing scheme (grid pattern) is very important for FLUENT to provide accurate results. For doing simulation of the intake manifold model we have to do first meshing, in this technique the flow is domain is converted or spilt into various sub domain primitives like hexahedral and tetrahedral. Care must be taken to ensure proper continuity of solution across the common interface between two sub domains, so that the approximate solutions inside various portions can be put together to give a complete picture of fluid flow in the entire domain.

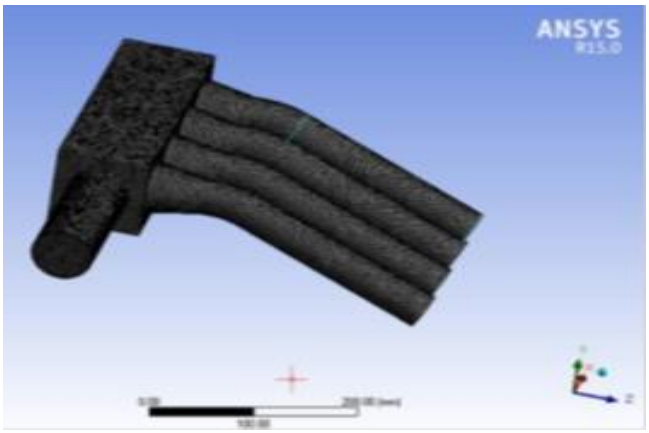

Fig 1: mesh of intake manifold

Table: Mesh details

Nodes	176396
Element	575599

IV. BOUNDARY CONDITION

Boundary conditions are essential to do a simulation. In this problem inlet is open to atmosphere and at outlet suction pressure will act due to pistons down motion. So inlet is chosen as pressure inlet and outlet is chosen as pressure outlet. Pressure inlet boundary condition needs total pressure at the inlet. So from atmospheric condition total pressure at inlet is 0 Pa (gauge pressure). The default value of reference pressure in the operation condition is given as 101325. So Gauge pressure (0) = Absolute pressure (101325) – reference pressure (101325).

Table: boundary conditions

Pressure at flow inlet	0 Pa
Intake	38
Pressure at runner outlet	-101325 Pa
Hydraulic diameter	60 mm
Turbulence intensity	4.2%
Pressure at runner outlet	-101325 Pa

V. RESULTS AND DISCUSSION

From the above experimental and CFD analysis the following results are observed at various runners outlet of the intake manifold.

Experimental result of intake manifold:

1. Anemometer,

U-tube manometer

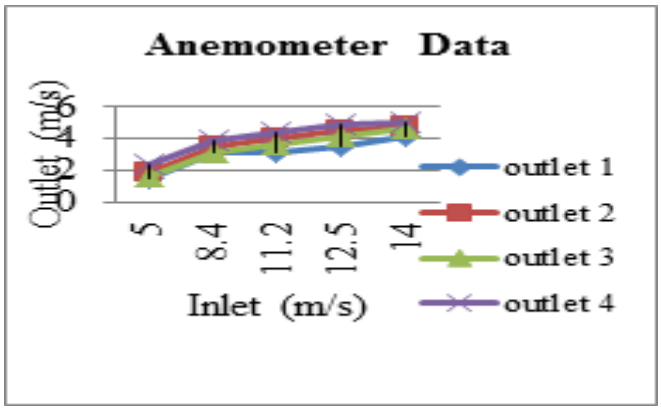

Fig 2: Velocities at outlets with variable inlet velocities

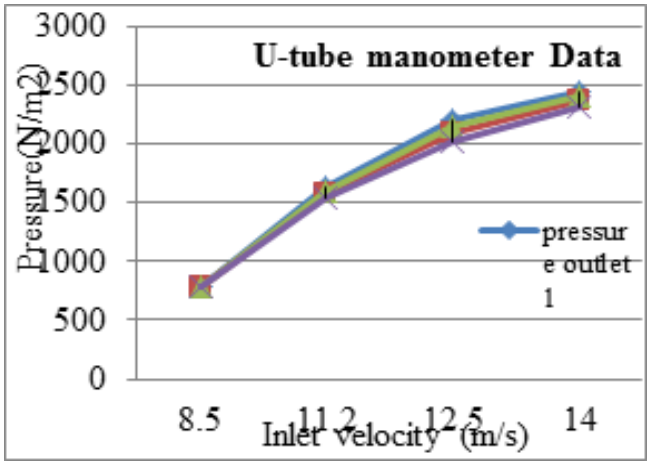

Fig 3: Pressure at outlets on variable inlet velocities

CFD ANALYSIS RESULTS OF INTAKE MANIFOLD:

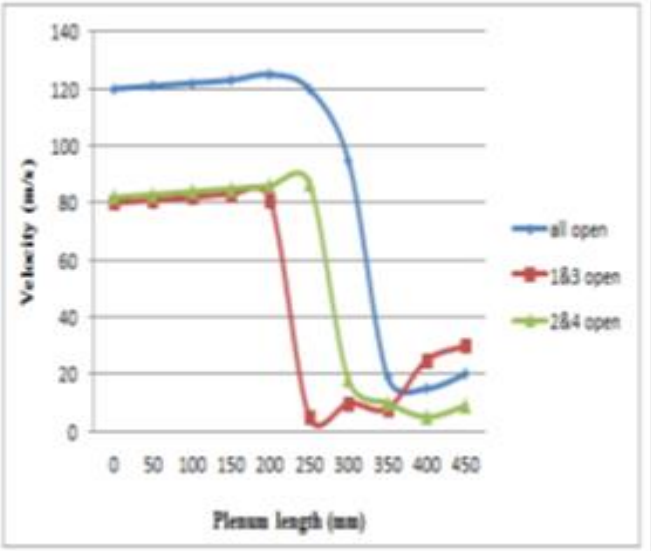

Fig 4: Velocity variations with plenum length

From Fig.4 it is observed that a sudden drop in velocity occurs for all the three conditions as the flow occurs from inlet of the plenum to the runner.

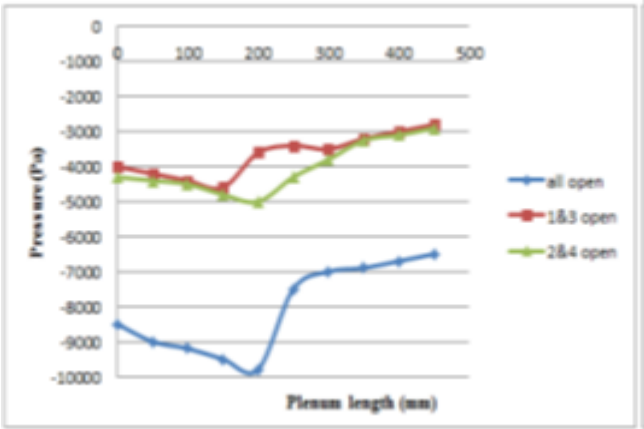

Fig 5: Pressure variations with plenum length

From Fig.5 it is observed that the pressure is higher when 1st and 3rd runners are open compared to 2nd and 4th runner open condition in the plenum. The pressure within is lower comparatively for the condition when all runners are open.

VI. CONCLUSION

After analysis, the concluded points are as follows:

- The variation in velocity is due to faulty design of plenum chamber.
- Plenum has casting and design defect.
- The outlet-1 has lowest velocity, so pressure losses are more in plenum chamber at runner-1 side.
- The inside projection of nuts as well as depth cut at runner-1 side block the pressure of air stream.
- All the cases show that runner 4 is working efficiently because it's located away from the pressure loss or recirculation region.

Geometry free from unwanted projections of nuts, stiffeners and depth cuts at extreme of plenum show good results. Air flow velocity not only increases in runner-1 by 16%, but improvement of velocity by 5% to 7% approx. in other runners outlets also take place. Nearly equal distribution of velocity in all.

REFRENCES

[1]Shinde, P.A., 2014. Formula SAE car engine. , 4(4), pp.4–8.
[2]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).
[3]Fluent. FLUENT 5 User Guide, 1999
[4] Ganeshan V. INTERNAL COMBUSTION ENGINES. New Delhi, TMH publication,2009
[5] R. K. Rajput, Internal Combustion Engines, Lakshmi Publications Pp 228, 238-240.
[6]John B. Haywood, Internal Combustion Engine Fundamentals, 2nd Edition, Tata McGraw-Hill Company,1988
[7]manifold, (adv.) "in the proportion of many to one, by many times". AD1526 Oxford English Dictionary.
[8]"History of Technology: Internal Combustion engines".*Encyclopædia Britannica.Britannica.com*. Retrieved 2012-03-20.
[9]*Pulkrabek, Willard W. (1997). Engineering Fundamentals of the Internal Combustion Engine. Prentice Hall. p. 2.* ISBN 9780135708545.
[10]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, (March 2011).
[11] 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).
[12]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).
[13]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,(2008),pp. 649−657

Mukesh Kumar Verma, M.Tech in Heat Power & Thermal engineering from VITS Jabalpur (MP), I am interested in Internal combustion engines: performance, emission, design and analysis using AUTOCAD, ANSYS & SOLID WORK.