Design and Optimization of Intake Manifold of Carburetor using Computational Fluid Dynamics

Mukesh Kumar Verma
Research Scholar
Department of Mechanical Engineering
VIT, Jabalpur (M.P.)

Jyoti Kale
Assistant Professor
Department of Mechanical Engineering
VIT, Jabalpur (M.P.)

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 losses due to which uneven distribution takes place at runner’s outlets.

Keywords: Intake manifold, CI Engine, Plenum, Restrictor, Flow Analysis, Piston, Fuel/Air Mixture, Computational Fluid Dynamics

I. INTRODUCTION

A research paper was published on Design of a new SI engine intake manifold with variable length plenum by M.A. Ceviz et al. This paper investigates the effects of intake plenum length/volume on the performance characteristics of a spark-ignited engine with electronically controlled fuel injectors. Previous work was carried out mainly on the engine with carburetor producing a mixture desirable for combustion and dispatching the mixture to the intake manifold. The more stringent emission legislations have driven engine development towards concepts based on electronic-controlled fuel injection rather than the use of carburetors.

In the engine with multipoint fuel injection system using electronically controlled fuel injectors has an intake manifold in which only the air flows and, the fuel is injected onto the intake valve. Since the intake manifolds transport mainly air, the supercharging effects of the variable length intake plenum will be different from carbureted engine. From the results of this study, the following conclusions can be deduced:

Research and optimization of intake restricter for Formula SAE car engine by Pranav Anil Shindhe This research paper aims to optimize a venturi type restrictor which is to be fitted in the intake manifold of a Formula SAE car engine. The main purpose of 20mm restrictor in intake manifold is to restrict mass flow passing to the engine thus reducing its maximum power. Objectives of this research is to optimize a venturi type design to allow maximum possible mass flow rate to the engine from 20 mm restrictor buy reducing the difference in pressure across venturi at all speeds. Analytical calculations are done based on standard results to get maximum mass flow rate and CFD tool is used to calculate minimum pressure drop across the restrictor buy varying converging and diverging angles of venturi. It can be observed from CFD results that for converging and diverging angle of 12 degrees and 6 degrees respectively minimum pressure drop can be achieved. A venturi, generally is used in diverse purpose for various applications in fluid dynamics involving either liquid or gas. In this project it is brilliantly used in reducing power of engine. As in this competition all the teams are busy trying to squeeze almost all single horse power available even with the restrictor attached, this gives rise to increasing research in optimization and finding out alternative technology for increasing mass flow rate to engine. One such technology used is supercharging of air downstream the venturi to increase the pressure on engine side. A venturi in itself can allow a maximum of 0.0703 kg/s of air flow to engine, considering no losses in friction and turbulence. From all the
research done till now it is clear that at converging angle of 12 degree and diverging angle of 6 degree we get maximum recovery of pressure. Computational fluid dynamics played important role in all analysis.

Design of intake manifold of IC engines with improved volumetric efficiency by Suresh Aadepu et. al 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. 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.

Jae-soon Lee performs a study for the optimal design of the intake system by varying the factors which can influence the volumetric efficiency, such as the volume of the plenum chamber, the length of the intake manifold and the pipe length between the surge tank and the plenum chamber.

II. METHODOLOGY

Continuity equation:

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

Momentum equation:

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

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:

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

Where the viscous dissipation $\Phi$ is defined as:

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

Bernoulli’s equation:

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

Hydrostatic law:

$$P = \rho \times g \times h$$

$\rho$ = Density

g = acceleration due to gravity

h = difference in mercury level in U-tube manometer

III. DIMENSIONS & MATERIAL PROPERTIES

<table>
<thead>
<tr>
<th>Section</th>
<th>Dimension (Diameter)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Intake</td>
<td>38 mm</td>
</tr>
<tr>
<td>Outlet 1 (inner diameter)</td>
<td>30 mm</td>
</tr>
<tr>
<td>Outlet 2 (inner diameter)</td>
<td>30 mm</td>
</tr>
<tr>
<td>Outlet 3 (inner diameter)</td>
<td>30 mm</td>
</tr>
<tr>
<td>Outlet 4 (inner diameter)</td>
<td>30 mm</td>
</tr>
<tr>
<td>thickness</td>
<td>mm</td>
</tr>
</tbody>
</table>
Table – 2

Material properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s modulus</td>
<td>7e+010 N-m²</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>0.346</td>
</tr>
<tr>
<td>Density</td>
<td>2710 kg/m³</td>
</tr>
<tr>
<td>Coefficient of thermal expansion</td>
<td>2036e⁻⁵/k</td>
</tr>
<tr>
<td>Yield strength</td>
<td>9.5e⁻⁷ N-m²</td>
</tr>
</tbody>
</table>

IV. 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 runners’ outlets achieved as compared to original intake manifold.

ACKNOWLEDGEMENT

I would like to express my deepest appreciation to all those who provided me the possibility to complete this paper. A special gratitude I give to our M.Tech Heat Power and thermal engineering project co-coordinator, MISS JYOTI KALE whose contribution in stimulating suggestions and encouragement, helped me to coordinate my project students especially in writing this paper

REFERENCES