Scientific Journal Impact Factor (SJIF): 1.711
International Journal of Modern Trends in Engineering and Research
IJMTER
ISSN (ONLINE) : 2349 - 9745
ISSN (PRINT) : 2393 - 8161

A Review on Design, Development and Optimization of Air Intake Manifold

Mr. Nilesh B. Kakad[1], Dr. Sanjay B. Zope[2]

[1,2]Department of Mechanical Engg, SVCET, Rajuri, Pune

Abstract— The main function of the intake manifold is to uniformly distribute the combustion mixture to each intake port in the cylinder head(s). The proper distribution of air is important to optimize the efficiency and performance of the engine. If the air supply is uneven it leads to less volumetric efficiency, power loss and increased fuel consumption. There is possibility to change the geometry of intake manifold to increase its volumetric efficiency. The finite element method is used to optimize the intake manifold. The factor which affects the swirl strength and mass flow rate in intake manifold .

Keywords— Intake manifold, CFD, Pressure, Swirl Strength, Smoke

I. INTRODUCTION

The principle assignment of an Inlet Manifold (IM) is to convey air inside the complex runner consistently, which is crucial for an improved IM plan. The IM outline has solid impact on the volumetric proficiency of the motor. An uneven air appropriation prompts less volumetric productivity, power misfortune and expanded fuel utilization. Contingent upon the sufficiency and period of weight waves inside the IM, filling of barrels via air can be influenced absolutely or contrarily. The plentifulness and period of these weight waves rely on upon IM outline, motor pace and valve timing. The insecure way of the affectation implies the impact of the complex on charging is greatly reliant upon the motor rate. This is on account of the passage of air inside the IM is an element of shifting heartbeats into it. Along these lines these heartbeats ought to be adjusted in motor manifolds to give obliged force [1].

Figure1.Intake Manifold [1]

IM can be planned in different courses, in spite of the fact that for actually suctioned motors the main suitable strategy which considers the entry of waves in the funnels. The reason is huge impact of shaky stream which influences volumetric proficiency of the motors; such technique is characterized as wave activity count system. It is conceivable to utilize such systems with turbocharger motors in light of the fact that the part of turbocharger goes about as channels to the waves and power the gas into the barrels; such motors are less defenseless to wave's impact. The general objective of this work is to re-enact the wind stream of admission complex for both unfaltering and precarious conditions. This additionally serves to enhance the capacity of the IM to give air indistinguishably to all barrels with the minimum conceivable weight misfortunes for turbocharger engines [1]. The primary goal of the present work was to make a computational investigation of enduring low through admission complex, port, valve and valve seat for different valve lifts. Three-dimensional stream inside of the complex, port and valve was mimicked utilizing computational liquid flow (CFD) and the code Fluent. Stream structures for the different valve lifts were anticipated. The collective weight map from calculation gave exhaustive data on the admission locale stream. Examination was completed for runner 1 at three distinctive valve lifts for different speed. The numerical assessment of the wind stream through an admission complex was done utilizing business programming Fluent [3].

II. LITERATURE REVIEW

S. Karthikeyan et al [1] had indicated From the AVL BOOST software the sudden increase in pressure waves are observed with initial manifold design. The initial intake manifold is not able to provide uniform distribution of air to all the cylinders. Due to this performance of the engine is poor. This is observed by an increase in the smoke level. Therefore the initial IM is optimized for uniform flow, by using CFD software. From the CFD results, 76% mass fraction of air is observed for all the three runners at 1800 rpm. Further experimentally air pressure inside the runners are investigated and increased air pressure of 13% shows that flow of air has increased inside the runner for the optimized IM design. The reduced smoke level indicates better air and EGR mixing inside the engine using optimized manifold.

M. Khan, S. M. Salim [2] had perform investigation of the flow regime, within the intake manifold, using CFD analysis is possible. However, applying the correct parameters and CFD models, which can provide economical simulated results, is highly important. This study appraises different CFD turbulent models and the importance of realistic parameters to successfully predict the flow inside IM. It is noticed that each model shows variation with the experimental results. While, RNG and RSM shows promising results with lower margin of error compared to standard k- ε and k- ω. For future studies, it is important to analyse each model using transient flow conditions for better accuracy and predictability of the flow inside IM with part throttle conditions. Furthermore, for future validations high calibrated sensors should be used to improve the accuracy of experimental data obtained.

B. M. Angadi et al [3] had indicated that at steady state condition analysis, in the individual runners pressure drop were determined. It was determine that pressure drop near runners was non-uniform and maximum pressure drop was observed in runner 1, because of the large flow separation region near runner 1. The flow is highly three-dimensional. At higher valve lift flow separation is critical.

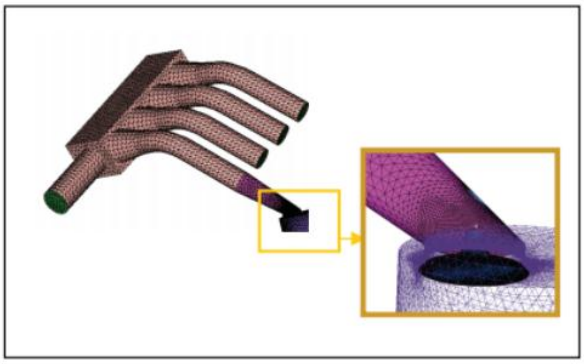

Figure 2. Meshing Model of Intake Manifold [3]

N.Maftouni et al [4] had investigated three hypothetical models of runner whose length is increased to 10%, 20% and 30% of initial value. No sensible change in volumetric efficiency is observed in the cases with 10% and 30% increased length, at any speed of engine. But the model with 20% extended runner, volumetric efficiency increases at the engine speed of 3500 and 4500 rpm. It means that if we increase 20% length of runners we increase volumetric efficiency. Optimizing the volumetric efficiency is a powerful method to evaluate an intake manifold performance. The results of both steady and unsteady simulations results suggest improving the performance of this intake manifold. According of this work, 3-D simulation can be used as a strong and powerful tool for design and optimization of intake manifolds.

Benny Paul et al. [5] observed the effect of helical, spiral, and helical-spiral combination manifold configuration on air motion and turbulence inside the cylinder of a Direct Injection (DI) diesel engine motored at 3000 rpm. By using the CFD tool (FLUENT), they compared predicted CFD results of mean swirl velocity of the engine at different locations inside the combustion chamber at the end of compression and the turbulence modeled using RNG k-□ model stroke with experimental results available in the literature. They also compared the volumetric efficiency of the modeled helical manifold. 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 the spiral-helical combined manifold is 10% higher than that of spiral manifold. Conclusion of result shows that Helical-spiral combined manifold creates higher swirl inside the cylinder than spiral manifold. Helical manifold provides higher volumetric efficiency. Helical spiral combined manifold provides higher mean swirl velocity at TDC of compression. Hence, for better performance a helical-spiral inlet manifold configuration is recommended by them.

M.A. Ceviz et al. [6] This paper investigates the effects of intake plenum length/volume on the performance characteristics of a spark-ignited engine with electronically controlled fuel injectors. 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. Engine performance characteristics such as brake torque, brake power, thermal efficiency and specific fuel consumption were taken into consideration to evaluate the effects of the variation in the length of intake plenum. The results showed that the variation in the plenum length causes an improvement on the engine performance characteristics especially on the fuel consumption at high load and low engine speeds According to the test results, plenum length must be extended for low engine speeds and shortened as the engine speed increases.

V. V. Naga Deepthi et al.[7] This paper aims at studying the effect of air swirl generated by directing the air flow in intake manifold on diesel engine performance. The turbulence was achieved in the inlet manifold by grooving with a helical groove of size of 1mm width and 2mm depth of different pitches to direct the air flow in inlet manifold. The tests are carried with different configurations by varying the pitch of the helical groove from 2 mm to 10 mm in steps of 2 mm inside the intake manifold. The measurements were done at constant speed of 1500 rpm. The results are compared with normal engine (without helical groove). The results of test show an increase in air flow, increases the brake thermal efficiency, mechanical efficiency and decrease in HC and Co emissions. On the other hand the volumetric efficiency is dropped by about 5%.

Martínez-Sanz et al. [8] the objective of this work was to develop a new design of a high performance intake manifold through a combination of CAD and FEM. First a FEA model was done, which included a complete thermal and structural analysis of the new intake manifold and the contact area between the aluminum coupling, using the combined tools of CATIA, ANSYS WORKBENCH, MATHCAD. Then several composite prototypes were made and analyzed. New materials field also in this idea was developed in order to study the different possibilities available to build an intake

Volume 02, Issue 10, [October – 2015] ISSN (Online):2349–9745 ; ISSN (Print):2393-8161

manifold. Aluminum was finally decided to be used due to its great thermal properties and the low weight in comparison with some other materials like steel. A new problem appeared when it was needed to calculate the way of joining this intermediate coupling to the runners. The right solution of problem was to join the both parts with an adhesive. In this case the contact stress is minimum and the fatigue calculation is suitable for the implementation.

III. PROBLEM STATEMENT

A detailed assessment of the problem of the intake manifold is done and it is realized that the performance of engine is affected by the uneven distribution of air to the runners. This uneven distribution of air may affect the volumetric efficiency of engine. To solve this problem the intake manifold is redesigned and optimized.

IV. OBJECTIVE

- To propose a good geometry model for intake manifold to improve the performance of engine.
- To achieve equal velocity of flow in all runners of intake manifold

V. PROPOSED METHODOLOGY

To achieve a best intake manifold of engine, a methodology is proposed as given below:

- Experimental analysis of intake manifold,
- Modelling-Creating prototype of given intake manifold in Design software for stress analysis,
- CFD analysis of the intake manifold
- Comparison of experimental and CFD analysis
- Propose modifications for a given intake manifold

A. Experimental Analysis of Intake Manifold

The setup shown in Fig 3. below is for taking out the reading at outlets of intake manifold with Anemometer and U-tube manometer. 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, Regulator to control the flow of air, Thermometer and various attachments example pressure taps, pipes attachments.

Figure3. Anemometer

B.3-D CFD Analysis of Intake manifold

Conventional admission complex enhancement had been in light of tests of IM. This experimentation system can be powerful however is exceptionally costly and tedious. Adjacent to this technique can't give any data about the genuine stream structure inside the admission complex. This essential data can be acquired utilizing 3D CFD examination. The outline architects can consider the stream structure and comprehend whether a specific admission complex performs accurately or not. 3-D CFD re-enactment can be isolated in two stages as below [4].

1. Steady State Analysis

The primary point of enduring state investigation is to find out the wind stream design for distinctive barrel opening conditions. Further swirls development amid suction stroke can be broke down. Relentless state examination can give the misfortune coefficients however it can't give any data around an IM execution in the transient working conditions. The limit condition (BC) utilized as a part of the enduring state re-enactment is steady pressure [1]. The weight misfortune coefficients for individual runners can be resolved with utilizing consistent state reproduction. This data can be gotten from an enduring stream test (stream seat) as well; however reason for this sort of reproduction is to be prepared for unstable re-enactment of IM. The limit conditions (BC) in enduring state re-enactment are steady pressure [4].

2. Transient Analysis

Consistent state study can be quick and can give the misfortune coefficients yet this data can't give any data around an IM execution in the working circumstance. Insecure state re-enactment can foresee how an IM function under genuine conditions. The limit conditions are no more steady however time variation. These limit conditions are acquired from the 1-D gas flow examination by utilizing the Wave code [4]. Flow through an admission complex is subject to the time since wrench point positions fluctuate regarding time. Shaky state reproduction can anticipate how an IM function under genuine conditions. The limit conditions are no more consistent yet differ with time. The IM weight information is acquired from 1D investigation utilizing AVL BOOST software [1].

3. Simulation Procedure

The following step shows steady and unsteady simulation procedures:

- Creation of 3D model of intake manifold using any mechanical software.
- Mesh generation in ICEM CFD software by using suitable mesh size.
- The information of cells is used as the input for CFD solver (CFX).
- From the post processing process eddies and mass fraction of air flow can be obtained.
- Finally the steady state simulation results are compared with experimentally investigated engine test rig data.
- Comparison between the results of the steady state simulation and with the steady-state (flow bench) test rig data.
- Analysis of the results and recommendation some suggestions to improve the performance of the intake manifold [4].

VI. EXPECTED CONCLUSION

- Equal distribution of air to all runner
- The volumetric efficiency may also increase due to proper supply of air
- Optimum mass flow rate can be obtained.

REFERENCES

[1] S.Karthikeyan, R.Hariganesh, M.Sathyanadan, S. Krishnan, "Computational Analysis Of Intake Manifold Design And Experimental Investigation On Diesel Engine For LCV, International Journal Of Engineering Science And Technology (IJEST)," ISSN : 0975-5462, Vol. 3 No. 4 Mar 2011,Pp 2359-2367.

[2] M. M. Khan, S. M. Salim, "Evaluation Of CFD Sub-Models For The Intake Manifold Port Flow Analysis," EURECA 2013 – Evaluation Of CFD Sub-Models For The Intake Manifold Port Flow Analysis,Pp 93-94.

[3] B.M.Angadi, Anandkumar S Malipatil, V.V.Nagathanr.S.Kattimani, "Modelling And Analysis Of Intake Manifold Of A Multi-Cylinder SI Engine," The 37th National & 4th International Conference On Fluid Mechanics And Fluid Power, December 16-18, 2010, IIT Madras, Chennai, India.

[4] NeginMaftouni , Reza Ebrahimi, "Intake Manifold Optimization By Using 3-D CFD Analysis With Observing The Effect Of Length Of Runners On Volumetric Efficiency," Proceedings of The 3 Bsme-Asme International Conference On Thermal Engineering 20-22 December, 2006, Dhaka, Bangladesh.

[5] Federico Brusiani, StefaniaFalfaria, Giulio Cazzoli, "Tumble Motion Generation In Small Gasoline Engines: A New Methodological Approach For The Analysis Of The Influence Of The Intake Duct Geometrical Parameters," Energy Procedia 45 (2014) 997 – 1006 68th Conference Of The Italian Thermal Machines Engineering Association, ATI2013.

[6] Long Xie, HarutoshiOgai, And Yasuaki Inoue, "Modelling And Solving An Engine Intake Manifold With Turbo Charger For Predictive Control," September 2006, Asian Journal Of Control, Vol. 8, No. 3, Pp. 210-218.

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

[8] Ceviz M.A and Akın M., "Design of a new SI engine intake manifold with variable length plenum" ELSEVIER JOURNAL, (2009).

[9] Prasad S. L. V, Pandurangadu V, Prathibha Bharathi V. V. and Naga Deepthi V. V.,"Experimental Study Of The Effect Of Air Swirl In Intake Manifold On Diesel Engine Performance" International Journal of Multidisplenary Research and Advances in Engineering (IJMRAE), ISSN 0975-7074, Vol. 3, No. I, January 2011, pp. 179-186.

[10] 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.

[11] Martínez-Sanz A., Sánchez-Caballero S, Viu A. and Pla-Ferrando R , "Design And Optimization Of Intake Manifold In A Volkswagen Car" ANNALS of the ORADEA UNIVERSITY. Fascicle of Management and Technological Engineering, Vol X (XX) ,(2011).