Awakening to Reality

Available online at www.elixirpublishers.com (Elixir International Journal)

Mechanical Engineering

Elixir Mech. Engg. 95 (2016) 40752-40756

Elixir ISSN: 2229-712X

Steady state investigation of air intake system in automobile engine using CFD

R.Manikandan and E.James Gunasekaran

Department of Mechanical Engg, Annamalai University, Annamalai Nagar- 608002, Tamil Nadu, India.

ARTICLE INFO

Article history:
Received: 23 April 2016;
Received in revised form: 28 May 2016;
Accepted: 3 June 2016;

Keywords
Ansys fluent,
Computational fluid,
Dynamics (CFD),
Air intake system,
Steady flow.

ABSTRACT

The main objective of the present work is to conduct the computational study of steady flow through the air intake system of a multicylinder SI engine. The CAD geometry of the air intake system of the multi cylinder in line engine was created. Two modifications on the induction side one with circular cross section (Path A) and another with rectangular cross section (Path B) were carried out. All the three models are simulated using ANSYS FLUENT, CFD code for the computational study, steady state flow conduction was assumed and the analysis is carried out assuming the intake air flows steady into the system at mean piston speed of the engine. Flow field is predicted by solving the Navier strokes equation and the turbulence is modeled using k-ε RNG model. Good agreement is observed from predicted result.

1.Introduction

At the present the major objective of engine designers are to achieve the goals for best performance and lowest possible emission levels. As the design concept is now a day, down signing, it has become imperative to enhance volumetric efficiency for additional power. To maximize the mass of air induced into the cylinder during suction stroke, the air intake design which plays an important role is to be optimized. The design becomes more complex in case of a multi cylinder engine as air has to be distributed equally to all the cylinders. Hence configuration of air intake system becomes an important criteria for the engine designed. Achieving this by means of experimental methods would cost time and money. There is a need for CFD method (numerical method) which could estimate the volumetric efficiency of the engine during the design stage itself, without undergoing anytime to time and cost consuming experiment. Also mapping the static pressure velocity magnitude distribution of the intake system is an effective method for analyzing the computational prediction of flow. In the present investigation the influence on the steady flow in the intake is being studied through the use of the CFD code ANSYS FLUENT Static pressure and velocity magnitude in the intake have been predicted. The mass flow rates predicted have been compared to the three different volume of intake.

2.Literature Review

Several studies were conducted to analyze the flow through intake system using numerical as well as experimental techniques. A brief review of the literature associated with these area of research has been presented.

Sulaiman etal [7] have studied the flow characteristics of air flowing through various designs of intake manifold of a 200 cc four stroke engine. This study is done by three dimensional simulation of the flow. Simulations are validated by the experimental study. From this study, it was revealed that the variations in the geometry of the air intake system could result in a difference of up to 20% of induced mass of air entering the combustion chamber.

AS pnalpogar et al [6] have used CFD techniques of the flow it air through intake system of an automobile industry to predict the pressure drop. 3D viscous CFD analysis was carried out for an existing model to understand. The flow domains such as intake manifold, filter media and air filter were modeled using a surface modeler and flow analysis was done by a CFD code ANSYS.

3. Geometry Model

Fig (1) shows the solid model geometry existing air intake system with multi cylinder. Fig (2) shows solid model of modified air intake system (circular volume) with multicylinder (path A) Fig (3) shows solid model of modified air intake system (rectangular volume) with multi cylinder (path B)

Fig 1. Solid Model of existing air intake system

Fig 2. Solid model of modified air intake system circular cross section.

Tele:
E-mail address: mechmani1978@gmail.com

Fig 3. Solid model of modified air intake system rectangular cross section.

In order to save the CFD computational time and cost trivial geometrical details that are not important to form the fluid flow as a point of view, such as filets ,bends stiffeners and steps are avoided in the model. All the geometry are obtained from solid model.

Objective

Study of flow through air intake system with multi cylinder engine.

Prediction of mass flow rate using the CFD and comparing it with standard air intake and two modified design of air intake system for a four cylinder SI engine at Full open throttle position.

4. Methodology

The methodology adopted for the present work is as follows.

Flow through the air intake system is simulated to study the cylinder flow field in multi cylinder engine. which included the flowing steps.

i. Solid modeling of the air intake system with multi cylinder engine geometry.

ii. Mesh generation solution of the governing equation with appropriate boundary conditions.

iii. Comparison of simulated results like pressure and flow field are compare between standard intake and two modified system to two modified systems (Path A) and (Path B).

Exhaust side valves were closed. The mean piston velocity was calculated from crank speed. Piston head was considered as outlet with flow rate equivalent calculated piston velocity. Only fluid flow volume was considered. Steady state ideal gas flow analysis was carried. Pressure boundary condition is prescribed at inlet of the filter cone inlet area.

Flow rate and flow velocity was monitored at inlet of filter. The study is expect the explore the potential of using CFD tools to design and optimization of air intake system geometry. The commercial CFD code ANSYS FLUENT is being used to analysis the flow. The CFD includes user interfaces to input parameter and to examine the results.

Analysis mainly involves the creation of basic 3D model, grid generation and fixing the boundary condition Modeling and meshing is done in tetrahedral mesh and is exported to ANSYS FLUENT for completing the mesh.

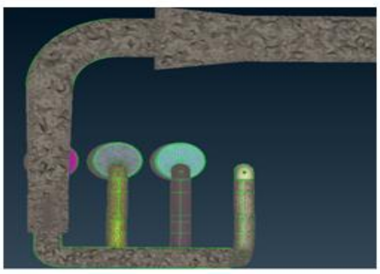

Fig 4. Meshed model of air intake system with multi cylinder engine.

5.Steady State Flow Assumption

The flow pattern in the air intake system with multi cylinder engine region is insensitive to flow and this could be predicted through steady flow test and computational simulation with reasonable accuracy

6. Boundary Condition

The air is at atmospheric pressure and at 32°C at the entry point of air filter. The out let boundary is specified with velocity boundary at every piston surface. In order to realistically simulate the actual flow velocity of air during the intake process, the velocity of air as it goes through the piston (assumed to the open end) is at the velocity of piston. Hence outlet velocity conditions of 1.825 m/s, 3.042 m/s, 4.258 m/s, 5.475 m/s, 6.692 m/s and 7.908 m/s were specified at the outlet boundary.

7. Turbulence Model

Currently the most popular turbulence model which is used in a practical setting is the two equation k-ε model. This model employs two additional transport equations. One for the turbulence kinetic energy (k) and the another one for the dissipation rate (ε). Near wall treatment is handled through generalized wall function. In this study high Reynolds number k-ε model has been used. This model is well established and the most widely validated turbulence model.

8.Result and Discussion

8.1 Comparison of Static Pressure Plot

Figure 5 to 10 compares the static pressure at intake plenum and through the cylinder at the engine speed of 3250 rpm. Compared to the standard intake arrangement the modified intake systems (path-A and path-B) shows lower static pressure as marked in the region by a circle in Figures 5 to 7. Figures 8 to 10 also show the static pressure contour on a sectional plane passing through the cylinder and manifold. It can be observed the modification results in lower static pressure for both Path A and Path B.

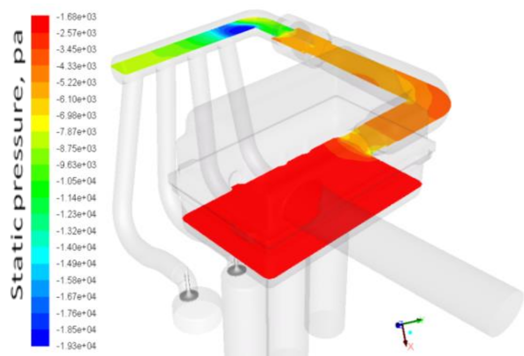

Fig 5. static pressure plot for standard model (3250 rpm) [intake].

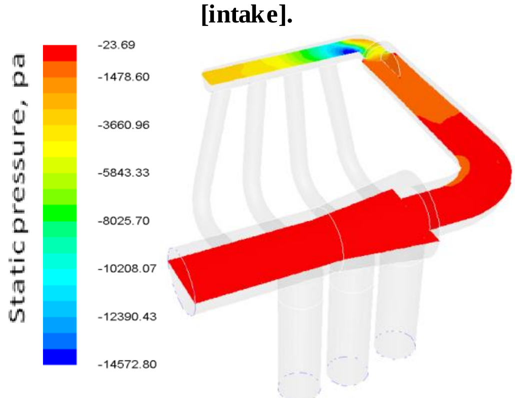

Fig 6. static pressure plot for Path A model (3250 rpm) [intake].

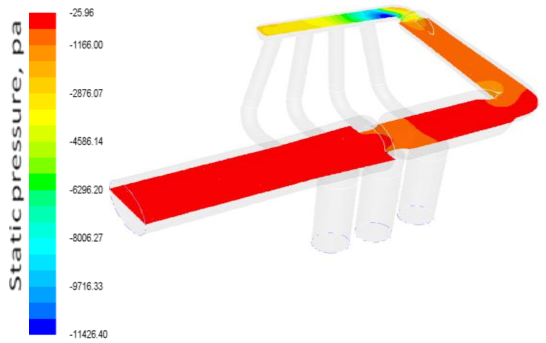

Fig 7. Static pressure plot for Path B model (3250 rpm) [intake].

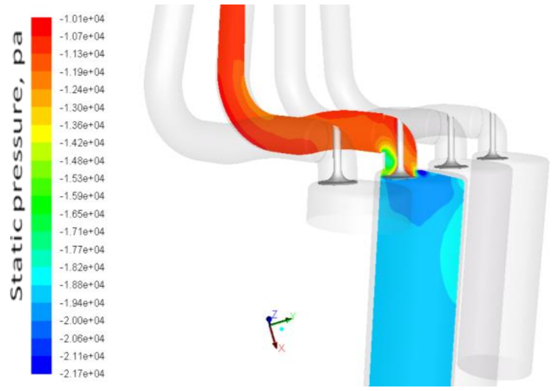

Fig 8. static pressure plot for standard model (3250 rpm) [cylinder 2].

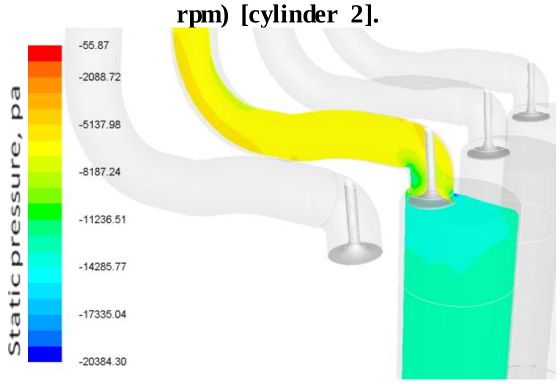

Fig 9. static pressure plot for Path A model (3250 rpm) [cylinder 2].

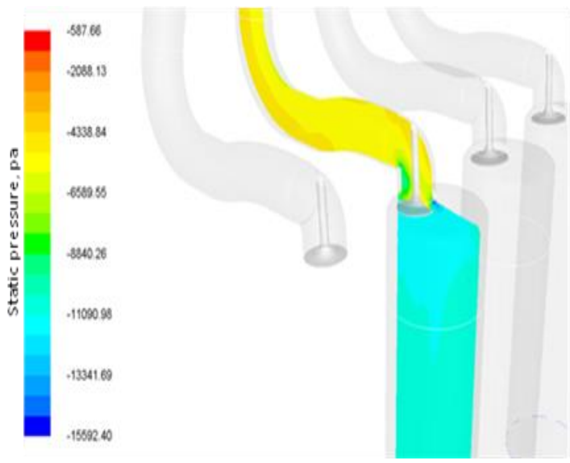

Fig 10. static pressure plot for Path B model (3250 rpm) [cylinder 2].

8.2comparison of Velocity Magnitude Plot

Fig (11 to 16) compares the velocity magnitude at the intake domain at a speed of 3250 rpm. It is evident that from the plot the velocity magnitude inside increase with modification.

The smooth curvature provide after section in the path A is very effectively reduces the recirculation caused by the flow separation compared to that in path B and standard model.

Figure 11 to 13 compares the velocity field near the intake plenum for the three intake systems. Due to smooth flow passage arrangement in the modified system velocity field increases. Figure 14 to 16 show velocity field along a plane passing through the intake manifold for cylinder-2 for the three systems (standard, path-A and path-B). It can be seen that velocity is highest for path-A, path-B also show a high velocity than standard intake systems. But its velocity is lower than path-A.

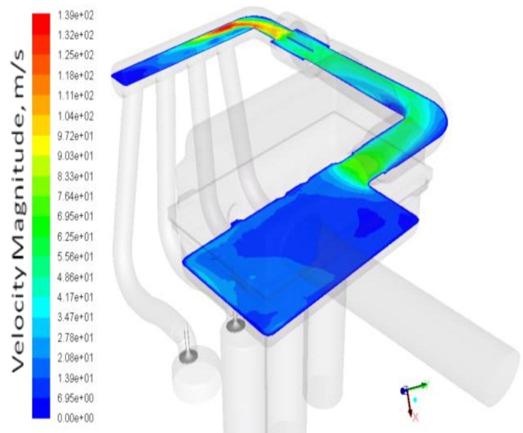

Fig 11. Velocity Magnitude plot for standard model (3250 rpm) [intake].

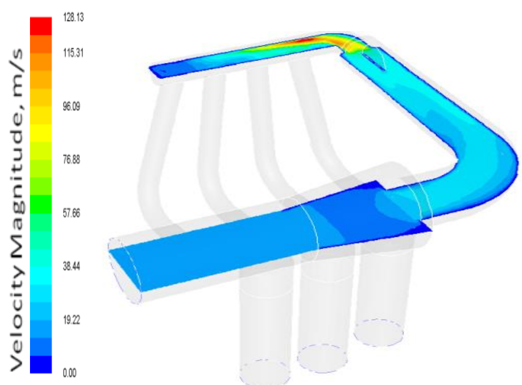

Fig 12. Velocity Magnitude plot for Path A model (3250 rpm) [intake].

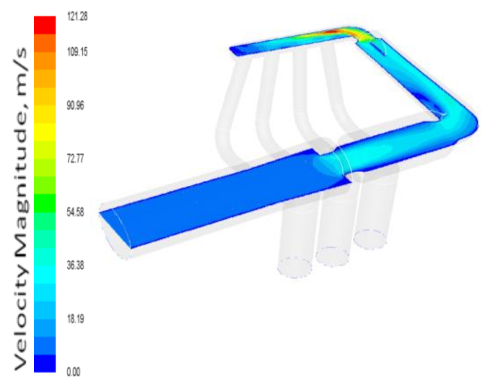

Fig 13. Velocity Magnitude plot for Path B model (3250 rpm) [intake].

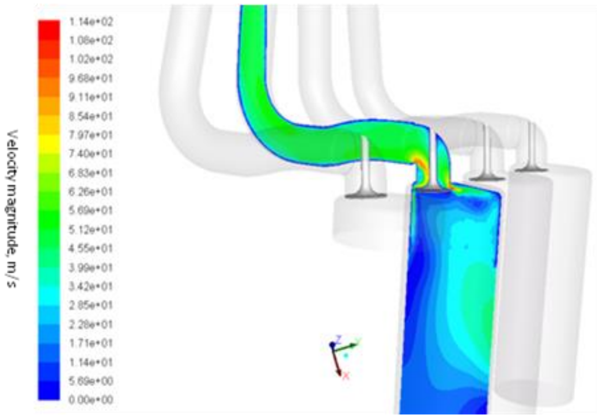

Fig 14. Velocity Magnitude plot for standard model (3250 rpm) [cylinder 2].

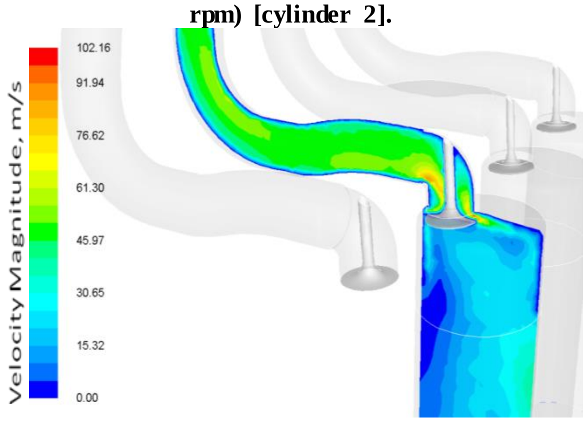

Fig 15. Velocity Magnitude plot for Path A model (3250 rpm) [cylinder 2].

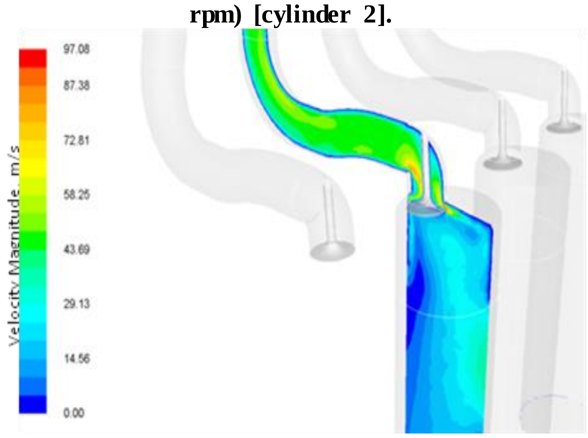

Fig 16. Velocity Magnitude plot for Path B model (3250 rpm) [cylinder 2].

8.3 Comparison of Total Pressure

The maximum total pressure drop between section 1 and 2 found in path A is -1339.04 pa compared to -1938.40 pa found in path B and -3342.0817 pa found in standard model for maximum speed of 3250rpm case.

Nearly more total pressure drop found in standard model compared to path A and path B at inlet section of the intake is reduces the ability to draw more air to the cylinders.

From Fig (17) it can be inferred that the modifications of the intake results in lower pressure drop. For the standard path the pressure drop is 7000 pa compared to 1000 and 3000 pa for modified intake path A and B. At the maximum speed of 3250 rpm the pressure drop for the modified flow paths A and B show a reduction is 71% and 50% respectively. Hence modified Path A is better compared to standard system and path B.

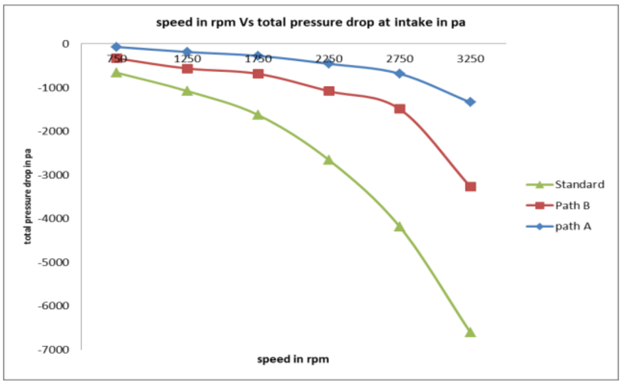

Fig 17. Comparison of total pressure drop at various speed for various models.

8.4 Comparison of Mass Flow Rate

Fig (18) the mass flow rate through the standard intake and modified system A and B. It is clearly seen that path A provides the highest mass flow at all engine speeds. The increase in mass flow at the maximum speed is at 0.1212 kg/sec for path A and 0.1004 kg/sec for path B which are 40% and 50% increase respectively.

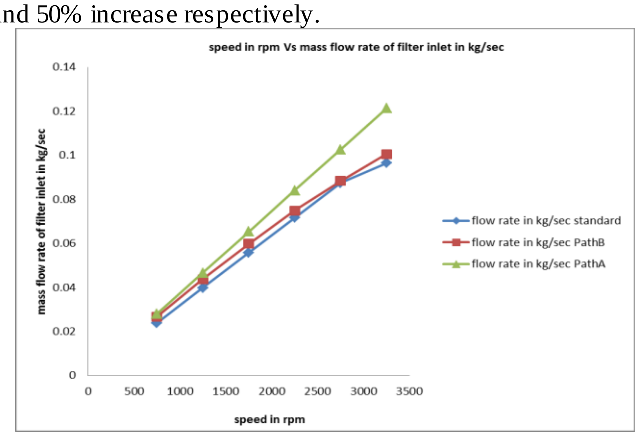

Fig 18. Comparison of mass flow rate at various speeds for various model.

9. Conclusion

CFD analysis was done using commercial CFD solver ANSYS FLUENT to understand the flow phenomenon in an air intake system

Box type air filter attached with the existing air intake system it reduces the mass flow rate and increase the pressure drop in the flow.

CFD results of existing air intake system shows more pressure drop and less mass flow rate of the air inducted to the cylinder. Out of these designs it is concluded that path A will give better performance than other two air intake system.

Box type air filter induces more pressure drop while compared to the cone type air filter. Smooth curved bend reduces the pressure drop in the intake system.

10. References

1. Kale, S.C and Ganesan, V, (2004) "Investigation of the flow field in the various regions of intake manifold of a SI engine", Indian journals of engineering and material Sciences. PP 134-148.

2. M.A. Jemni, G Kantehev, M.S Abid (2012) "Intake manifold design effect on air fuel mixing and flow for an LPG heavy duty engine" International journals of energy and environment PP 61-72.

3. Kale, S.C and Ganesan, V, (2005) "A Study of steady flow through a SI engine intake system using CFD".

4. Suresh Aadepu, I.s.n.v.r. Prasanth, Jarpala. Murali Naile (2014)" Design of intake manifold of I.C engine with improved volumetric efficiency". International journal & magazine of engineering, technology, management research.

5.B.M Angadi, Anadhakuamr S.Malipatil, V.V nayathan, R.S leattimani (2010) “modelling Analysis of Intake manifold of a multi cylinder SI engine” International conference on fluid mechanical and fluid power.

6.A.S Phulpasar N.S.Gabal (2015) “CFD analysis of air intake system” Ird India.

7.S.A Sulaiman, SHM Murad, I Ibrahim ZA Adhul learim (2010) “Study of flow in air intake system for. A Single cylinder Go- kart engine” International Journal of automobile and mechanical engineering.

8.D.Ramasamy, Zamri MS mahendran, S.Vijayan (2010) “ Design optimization of air intake system(AIS) of 1.6L engine by adding fluid vane” IMECS PP 999-103.

9Ansys fluent tutorial.