COMPUTATIONAL ANALYSIS OF INTAKE MANIFOLD DESIGN OF A FOUR CYLINDER DIESEL ENGINE

Anilkumar.D.B[1], Dr. Anoop Kumar Elia[2]
[1]Asst Professor, SIT Gulbarga, [2]Associate Professor, GNDEC Bidar,,

Abstract

The main objective of the present work is to make a computational study of low pressure steady state through intake manifold. Three-dimensional flow within the manifold runners was simulated using computational fluid dynamics (CFD) and the code Fluent. Flow structures for the both design were analyzed and predicted. The total pressure graph from computation provided comprehensive information on the intake region flow. Analysis was carried out for every runner for both the designs. It is quite familiar that a duly designed intake manifold is essential for the optimal performance of an internal combustion engine. Air flow inside the intake manifold is one of the important factors, which governs the engine performance hence the flow phenomenon inside the intake manifold should be fully optimized to produce more engine power with better combustion In present study, during the new engine development the pressure waves for the intake manifold is simulated using Fluent software. To study the internal air flow characteristic for the 4-cylinder diesel engine during transient conditions. As a result of this 3D CFD analysis, the disproportionate flow of air inside the runners is identified and pressure inside the runner is also experimentally investigated on the engine test bench. The numerical evaluation of the airflow through an intake manifold was carried out using commercial software Fluent. From the steady state analysis the pressure drop for individual runners were determined. It was observed that the pressure drop across runners was non-uniform and higher-pressure drop was observed in runner 1 which is due to the large flow separation region near runner 1.The flow is highly three-dimensional. It is strongly dependent on the valve lift except upstream of the port bend. At higher valve lift flow separation is critical. From the investigation, it is identified that the pressure inside the runners are uniform and smoke level is also reduced for optimized inlet manifold design

Index Terms—Intake Manifold, Gas direct Injection, Compressed Natural Gas, computational fluid dynamics, Specific fuel consumption, Throttle Body Injection Mixer, controlled auto ignition, Reynolds Averaged Navier Stokes, Exhaust Gas Recirculation..

I. INTRODUCTION

An intake manifold is one of the primary components regarding the performance of an internal combustion engine. An intake manifold is usually made up of a plenum inlet duct, connected to the plenum are runners depending on the number of cylinders which leads to the engine cylinder. A typical Intake manifolds have to be designed to improve engine performance by avoiding the phenomena like formation of eddies and non uniform of flow of individual runners. It is quite familiar that a duly designed intake manifold is essential for the optimal performance of an internal combustion engine. Air flow inside the intake manifold is one of the important factors, which governs the engine performance Hence the flow phenomenon inside the intake manifold should be fully optimized to produce more engine power with better combustion In present study, during the new engine development the pressure waves for the intake manifold is simulated using fluent.

The main task of an Inlet Manifold is to distribute air inside the manifold runner uniformly, which is essential for an optimized Inlet Manifold design. The Inlet Manifold design has strong influence on the volumetric efficiency of the engine. An uneven air distribution leads to less volumetric efficiency, power loss and increased fuel consumption

Depending on the amplitude and phase of pressure waves inside the Inlet Manifold, filling of cylinders by air can be affected positively or negatively. The amplitude and phase of these pressure waves depend on Inlet Manifold design, engine speed and valve timing. The unsteady nature of the induction means the effect of the manifold on charging is extremely dependent upon the engine speed. This is because the entry of air inside the Inlet Manifold is a function of varying pulses into it. Therefore these pulses should be fine tuned in engine manifolds to give required power.

II. DESIGN PARAMETERS FOR INTAKE MANIFOLD

To design an optimal intake manifold, following parameters should be taken into consideration:

1. Uniform distribution of air pressure to all the four cylinders, to overcome the uncompleted combustion.
2. Proper designs of intake manifold profile helps to reduce the sudden raise in pressure waves which improve induction process and also eliminate the unnecessary turbulence and eddies inside the intake manifold

Objectives

Past development in the field of engines, there are still some areas where improvement can be done for achieving good performance. Intake manifold plays very crucial role in achieving high efficiency as well as in emission control.

So objectives of research are:

1. To reduce the pressure losses in intake manifold,
2. To achieve uniform pressure flow distribution in all runners of intake manifold, and
3. To propose a good geometry model for intake manifold to improve the performance of engine.

III. METHODOLOGY

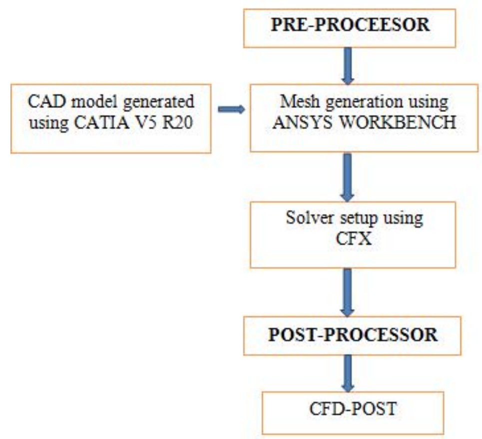
The computational steps in this project consist of three stages as shown in Figure

They are 1.Pre-processing.

2. Numerical Computation.
3. Post-processing.

The project began from preprocessing stage of geometry setup and grid generation. The geometry of the model is drawn using CATIA V5R13. The grid is generated by ICEAM.

The second stage is computational simulation by FLUENT solver using Finite Volume Approach. Finally is the post-processing stage where the Fluid Flow characteristics of the Intake Manifold are found.



3.1 Pre-processing

2.1.1 Geometrical Modeling and Grid Generation

Creation of Geometry

Geometry setup is made using wireframe and surface design to draw 3d model of Manifold as shown in figure.

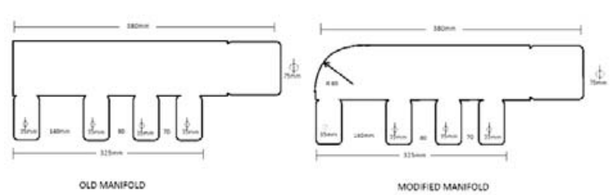


Fig -1: Dimensions of Manifold

The above dimensions are used to create a 3d Intake Manifold of diesel engine. The Intake Manifold merging of geometry is done using design in Catia. The geometry is enclosing in the enclosure whose dimensions almost equal to that of the model. Now the enclosure is freeze to see the model which was hidden during enclosing of the control volume. The 3D model of Intake Manifold is created in the Catia which is shown in below figure. The below figure shows the symmetry model of the Intake Manifold created in Catia.

In order to create a suitable model to run in FLUENT software, pre-design geometry can be created in FLUENT. Therefore, ICEM which is a geometric modeling and grid generation tool is provided along with the FLUENT technology. ICEM allows the user to import geometry from other designing software or computer-aided design (CAD)/CATIA software or create own geometry entirely based on ICEM itself. In addition, ICEM can automatically mesh surfaces and volume while allowing the user to manipulate the mesh through size functions and boundary layer meshing

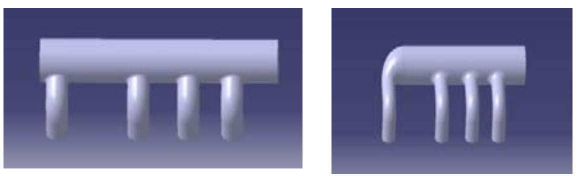


Fig:2 Geometry modeled in Catia

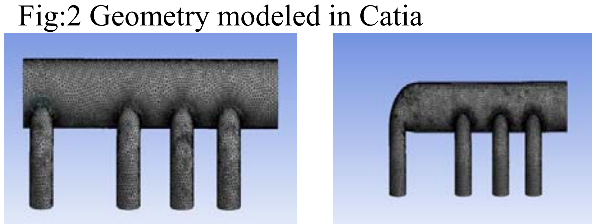


Fig:3 Grid Generated For CFD Analysis (Tetrahedral Elements)

The above mesh contains 184463 nodes and 451963 elements for convectional type intake manifold and 182022 nodes 447222 of elements for optimized intake manifold. so finally this mesh grains are to be reduced in fluent by smooth.

3.2 Numerical computation

The numerical simulation is done by solver thus after the completion of the mesh generation, mesh is checked in Ansys workbench mesh, if the quality of mesh is satisfied then the mesh geometry is sent for numerical simulation. But in this Intake manifold of 3d model created in catia it is to be check by smooth and swap procedure so that the mesh geometry is ready for numerical computation. The solver formulation, standard k- ε, boundary condition, solution control parameters and material properties are defined. After all the parameters are specified, the model is initialized. The initializing and iteration processes stopped after the completion of the computations. The results obtained were examined and analyzed.

• The flow to be considered is incompressible and turbulent.

• The solver used is the Pressure based solver, which gives good results for high speed incompressible flows.

Boundary Conditions

Pressure Outlet: Ambient atmospheric condition is imposed at outlet.

The velocity components are calculated for each angle attack case as follows. The x-component of velocity is calculated by and the y component of velocity is calculated by, where is the angle of attack in degrees.

Operating condition: The inlet velocity taken for 30 m/sec, at different speed Reynolds number at Re=4.8×105, air at STP (Temperature=300k, Pressure=101325pa) as the fluid medium.

Properties of Air:

Density : ρ = 1.225 kg/m3

Viscosity : μ= 1.7894e-05 kg/m-s

Solvers selection : Pressure based solvers.

Mathematical models : K- ε standard wall function

Solution controls : Gauss-Seidel flow turbulence energy.

Momentum : Second Order Upwind Scheme

Initialization : Inlet Values

Reference values : Inlet Values

Convergence Limit : 1x10-6

Assumptions

The flow inside Manifold is treated as steady & incompressible. The CFD computation is now defined. The explicit formulation normally used for cases where the characteristic time scale is of the same order as that of the pressure distance scale, for example the propagation of pressure waves. The implicit formulation is more stable and can be driven much harder to reach a converged solution in less time. Green-Gauss Node Based: This is slightly more computationally expensive than the other methods but is more accurate. Second Order Upwind for flow and turbulence discretization.

To accurately predict pressure, select the 'Second Order Upwind' schemes

The calculation is ready to compute, It is good practice (but not strictly necessary) to run the FMG and then check the coarse FMG solution before starting the main calculation iterations.

The number of requested iterations sets to 100 as it is computed for whole 3D model with enclosure it take much time for simulation of more than 500 iteration. If there are no spurious results from the FMG, so proceed to the main calculation by increasing the number of iteration to 100-500.

3.3 Post-Processing

In post-processing results are obtained, they are, pressure contours, velocity contours and velocity Streamlines for both designs with Reynolds number, and the same results of CFD is compared with the results of experimental analysis. The CFD analysis of intake manifold is done in two stages. First the three dimensional convectional intake manifold profile is analysed and compared with the optimized intake manifold. The comparison of the data arrived at by the CFD approach is given in the tabular form & graphical form for both the designs. The special smooth curved shaped intake manifold and the convectional manifold of same size and volume are numerically tested. CFD analysis was carried out on both designs and pressure & velocity coefficients were evaluated for different speeds of engine 30 m/sec air velocity for air flow. The air velocity of 30m/s is chosen for CFD analysis purpose due to the fact that the present value of air flow velocities of intake manifold. The CFD analysis was carried out using fluent software. The CFD results of both the fuselages were subsequently compared with each other.

IV. GOVERNING EQUATIONS

CFD is based on the fundamental governing equations of fluid dynamics which express the fundamental physical principles of fluid dynamics. These governing equations have four different forms based on how they are derived: integral and partial differential form, conservation and non conservation form. They are not fundamentally different equations but the same equation in four different forms. The conservation of mass (The Continuity Equation) expresses the fact that mass cannot be created or disappear in a fluid system, the net mass transfer to or from a system during a process is equal to the net increase or decrease in the total mass of the system throughout the process .The partial differential equation form of the continuity equation is shown in Equation.

$$\frac{\partial \rho}{\partial t} + \vec{\nabla} \cdot \rho \vec{v} = 0$$

Where ρ is the density of fluid (kg/m3); t is time (seconds); ↖ is velocity vector (m/s).

The conservation of momentum (Newton's second law), as shown in Equation 2-2, describes how the force action on the particle is equal to the mass of the particle times its acceleration. When applied to the fluid element it states that the variation of momentum is caused by the net force acting on a mass element.

The conservation form, partial differential equation, called the Navier-Stokes equation, is shown in Equation.

$$F = ma$$

Where F is body force (N), m is mass (kg), and a is acceleration (m/s2).

$$\frac{\partial (\rho \vec{v})}{\partial t} + \vec{\nabla} \cdot (\rho \vec{v} \vec{v}) = \vec{\nabla} \cdot p + \vec{\nabla} \cdot \vec{\vec{\tau}} + \rho \vec{b}$$

Where in addition to the variables defined in equation 1 and 2,↑ is viscous stress tensor (Newton) given by Equation, ⃗b is body force vector (Newton), and μ is the molecular viscosity coefficient.

$$\vec{\vec{\tau}} = \mu \left(\vec{\nabla} \vec{v} + \left(\vec{\nabla} \vec{v} \right)^T \right) - \frac{2}{3} \mu (\vec{\nabla} \vec{v}) \vec{\vec{I}}$$

The conservation of energy (the first law of thermodynamics) states that energy can neither be created nor destroyed, but can only change forms. Specifically, it states that any changes in time of the total energy inside the volume are caused by the rate of work of forces acting on the volume and by the net heat flux into it. The conservation form, partial differential equation is shown in Equation 2-5.

$$\frac{\partial \rho e}{\partial t} + \vec{\nabla} \cdot (\rho e \vec{v}) = \rho \dot{q} + \vec{\nabla} \cdot (k \vec{\nabla} T) - \vec{\nabla} \cdot (p \vec{v}) + \vec{\nabla} \cdot \left(\vec{\vec{\tau}} \cdot \vec{v} \right) + \rho \vec{b} \cdot \vec{v}$$

Where q˙is the rate of volumetric heat addition per unit mass , *T* is temperature, *e* is internal energy per unit mass.

A. References

Kale and Ganesan [1] investigated steady flow through intake manifold, port, valve and valve seat of a S.I engine for various valve lifts using CFD code STAR-CD. From the studies, the flow field details in different regions of the manifold for various valve lift were predicted. The analysis was carried out for runner 1 and 3 at three different valve lifts for various speeds at wide open throttle condition. Multi-block, trimmed cell were used for meshing the geometry. Grid independence studies were conducted to obtain a grid independent solution. The mesh size selected was between 450,000 and 600,000 cells, varying slightly between the low, medium and high valve lift. Since the velocity variation between 450,000 and 600,000 grids were less than 1% grid distribution of 450,000 was selected. Standard k- _ model was used as turbulence model. The input for boundary conditions was given as manifold pressure for the inlet of the plenum chamber and cylinder pressures (corresponding to the valve lift) for the exit of the valve. Due to the absence of measured data, a turbulence intensity of 5% and a length scale of 10 per cent of port diameter at the inlet plane were specified. The authors concluded that the valve lift has a predominant effect on flow structure of intake manifold. The results were validated with experimental data.

David et al. [2] used the study flow rig experiment to evaluate the swirl of a helical intake port design for different operating conditions. (Optimum swirl ratio is necessary according to the engine operating condition for optimum combustion and emission reduction). The variable swirl plate set up of the W06DTIE2 engine was used to experimentally study the swirl variation for different openings of the wall. The sliding of the swirl plate results in the variation of the area of inlet port entry. Therefore in this study a swirl optimized combustion system varying according to the operating conditions by a variable swirl plate mechanism was studied experimentally and compared with computational fluid dynamics predictions. Based upon the results the author concluded that the CFD model has an application in improving the swirl generating capacity of the port and evolution / distribution of the in-cylinder swirl during intake process according to emission norms. It can provide ways to design high swirl generation capacity intake port which reduces both NOx and particulates the two main pollutants in diesel engines, which is very necessary to attain Euro IV emission norm.

K.Mukund et al [3] conducted a study to predict and analyze the flow around poppet valves in IC Engine using dynamic mesh. The flow field was solved using the commercially available software FLUENT. The governing equations used were the unsteady mass, momentum and energy conservation equations. The standard k-e model was used to model the turbulence in this problem. The mesh was generated using GAMBIT software the mesh was primarily consisted of triangular and quadrilateral elements.

Negin Maftouni , Reza Ebrahimi[4]

It is quite wellknown that a properly designed Intake Manifold is vital for the optimal performance of an IC engine. This paper will present 3-D Simulation of a XU7 Engine Intake Manifold and the results will be discussed. Both steady and unsteady state simulations have been accomplished for this case. Steady state simulation results are compared with flow bench rig data for validation. Boundary condition for unsteady state simulation was obtained from 1-D WAVE code. In the present research the effect of length of runners on the volumetric efficiency has been analyzed by 3-D CFD model at different speeds. Three hypothetical models have been made that all of their runners' length is increased to 110,120 and 130% of initial value. In the model with 20% extended runners, the volumetric efficiency increases at 3500 and 4500 rpm. Finally according to the results of steady and unsteady simulations, some suggestions are recommended to improve the performance of this Intake Manifold.

Radivoje B. PEŠI], Aleksandar Lj. DAVINI], Sne_ana D. PETKOVI],

Dragan S. TARANOVI], and Danijela M. MILORADOVI] [5]

The volumetric efficiency significantly influences engine output. Both design and dimensions of an intake and exhaust system have large impact on volumetric efficiency.

Experimental equipment for measuring of airflow through the engine, which is placed in the intake system, may affect the results of measurements and distort the real picture of the impact of individual structural factors. This paper deals with the problems of experimental determination of intake airflow using orifice plates and the influence of orifice plate diameter on the results of the measurements. The problems of airflow measurements through a multi-process Otto/Diesel engine were analyzed. An original method for determining volumetric efficiency was developed based on in-cylinder pressure measurement during motored operation, and appropriate calibration of the experimental procedure was performed. Good correlation between the results of application of the original method for determination of volumetric efficiency and the results of theoretical model used in research of influence of the intake pipe length on volumetric efficiency was determined.

M. Safari, M. Ghamari, A. Nasiritosi[6]

It is quite well known that a properly designed Intake Manifold is vital for the optimal performance of an IC engine. This paper will present 3-D Simulation of a 1.6L MPFI Engine Intake Manifold by using the FLUENT code and the results will be discussed. Both steady and unsteady state simulations have been accomplished for this case.

Steady state simulation results are compared with flow bench rig data for validation. Boundary condition for unsteady state simulation was obtained from 1- WAVE code. Finally according to the results of steady and unsteady simulations, some suggestions are recommended to improve the performance of this Intake Manifold.

Wagner Roberto da Silva Trindade[7]

During the development of a air intake manifold simulation is necessary to verify the component characteristics in terms of flow considering runner in-balance as focus. Results can also present exhaust gases recirculation (EGR) distribution affecting each of the different runners in order to verify if those recirculated exhaust gases were been equally distributed among all the runner outlets or, at least, presenting a percentual difference of EGR concentration in each runner inside a defined tolerance range. Aiming these results some calculation techniques were used as follow: 1D modeling as well as 3D modeling was used

S.Karthikeyan et al [8] In this paper, pressure waves for the intake manifold is simulated using 1D AVL-Boost software, to study the internal air flow characteristic for the 3-cylinder diesel engine during transient conditions. Based on the 1D simulation results, the intake manifold design is optimized using 3D CFD software under steady state condition.

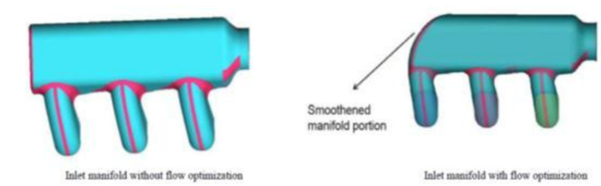


Fig 4.1: Inlet Manifold without and with optimization

From the 1D AVL BOOST software, variation of pressure pulses with various crank angles is obtained for all the three runners at different engine speed conditions and obtaining graph shows that pressure variation is more due to sudden increase in the velocity of incoming air. Therefore intake manifold (IM) geometry near the corners should be made smooth to avoid sudden increase in the pressure waves. Manifold profiles near the corners can be made smooth to avoid the reflecting shock waves with higher velocity as the engine speed increases to 1250 rpm. CFD analysis show that during second cylinder opening, some amount of air is trying to enter into first and third cylinder due to eddies. Flow reversals are present in the plenum, which is the reason for causing an improper distribution of air to all the runners in the initial IM design. But after optimization of the manifold the variations are reduced and almost uniform amount of air is entering the runners. 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 mixing inside the engine using optimized manifold.

Benny Paul et al. [9] 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.

V. RESULTS AND DISCUSSION

The steady state air flow analysis of intake manifold of 4-cylinder diesel engine has been carried out for the both design (i.e. convectional and optimized) in order to know: -

1) Pressure Contour
2) Streamlines analysis

Pressure Contours

Figure 5.1 shows the Pressure contour along plenum chamber with all runners in open Condition. It is observed that Pressure drops at 4th runner through the plenum chamber. This is due to transverse curve at 4th runner within the plenum. There is a drop in Pressure at the inlet of the runner 4 compared to other runners when all runners are in both design condition. This is due to a sharp bend at the region of runner 4 inlet and the plenum wherein the flow does not smoothly enter runner 4. At runner 3 and 2 local regions an increased in the velocity is observed.

Figure 5.2 (b) shows the Pressure contour for 1st and 3rd runner in open condition and 2nd and 4th runner in open conditions respectively. It is observed that when runner 1st and 3rd are open the Pressure distribution within the plenum changes drastically with higher Pressure occurring at inlet region of 1st and 3rd runner. Obviously as no flow occurs at 2nd and 4th runner the Pressure at these runners are very low. When runner 2nd and 4th are open the Pressure increases at the inlet region of these two runners. From fig 6.2 (a) 6.2 (b) it is clearly shows that Pressure near 4th runner is reduced for convectional intake manifold compare to optimized intake manifold

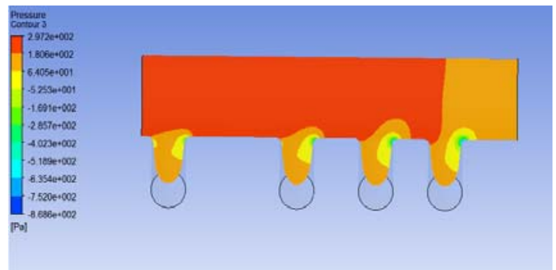


Fig 5.1 (a) Shows Velocity Contour Of Convectional Intake Manifold.

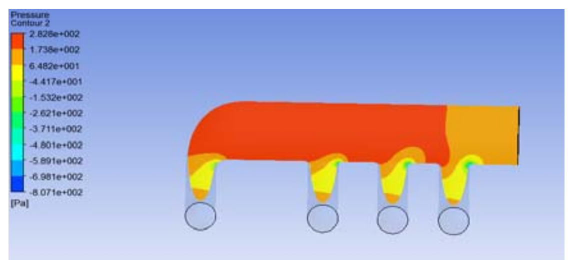


Fig 5.2 (d) Shows Velocity Contour Of Optimized Intake Manifold.

Velocity streamline

The steady state analysis is to find the air flow pattern for both design conditions using Velocity streamline. Further eddies formation during suction stroke can be analyzed. Steady state analysis can provide the loss coefficients. The boundary condition (BC) used in the steady state simulation is constant pressure.

From fig 5.3 (a) for convectional intake manifold eddies formation takes place near 4th runner due to transverse curve near the 4th runner. Even pressure loss takes place while entering ail flow inside runner 4. Due these pressure loss and eddies formation at the 4th runner will effect the other runners, which inturn creates non uniformity of air flow in other runners. Non uniform will create incomplete combustion of fuel which reduces engine performances.

After optimizing intake manifold by smoothening curve at the 4th runner which will reduces eddies formation. Even it has reduced pressure loss nearer entry of the 4th runner. This optimized intake manifold will increases performance by decreasing Eddies formation, increasing uniformity of air flow and reducing pressure loss. Which in turn will burns fuel completely thus increases engine efficiencies.

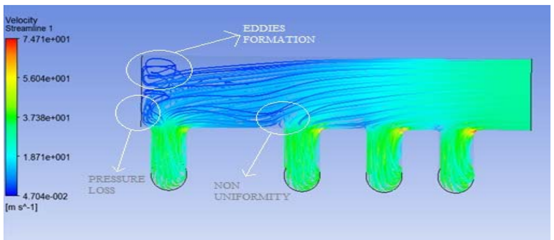


Fig 5.3 (a) shows Velocity streamline Of Convectional Intake Manifold.

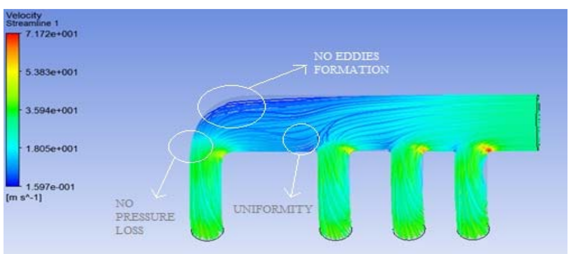


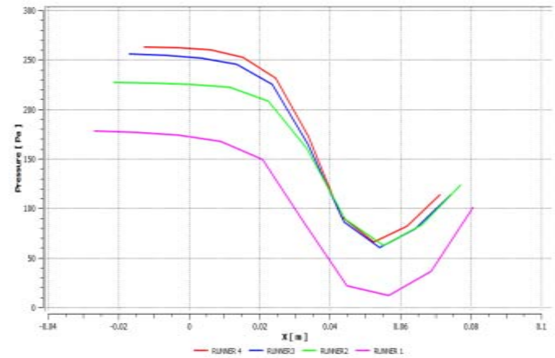
Fig 5.4 (b) Shows Velocity streamline Of Optimized Intake Manifold.

VI. CONCLUSIONS

After plotting graph along plenum for both the designs we get

From fig 6.1 (a) Vertical graph are plotted along runners, it shows pressure variations along runners to the circumferential/ radial plenum distance. From graphs it clearly shows intially for convectional intake manifold pressure variation for all 4 runners are non uniform due to vertical transverse curve near the 4th runner.

After optimizing near the 4th runner (i.e smoothening the transverse curve) pressure variation along all runners are uniformly distributed. Due to this uniformity fuel burns completely and completely combustion of fuel will increase engine performances.



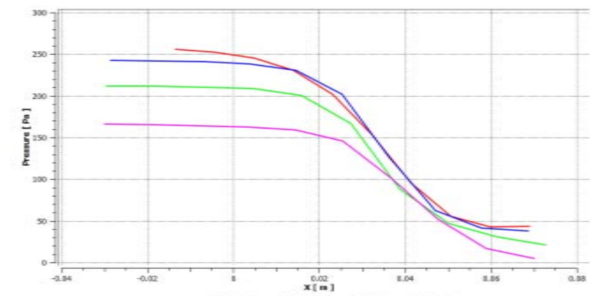


Fig 6.1 (a) Pressure variation along runners to plenum wall for the both convectional and optimized designs.

VII. EDITORIAL POLICY

The satisfaction and elation that accompany the successful completion of any task would be incomplete without the mention of the people who have made it a possibility. It is my profound gratitude that I express my indebtedness to all who have guided me to complete this project successfully.

I am grateful to our Principal Dr. Ashok Vangeri, for his guidance, support and valuable suggestions to complete this project. I sincerely acknowledge the encouragement, suggestions and impetus given to me by my PG Coordinator to complete the project assigned to me.

The valuable guidance, the exemplary support and timely suggestions made available to me by my Guide Dr. Anoopkumar Elia went a long way in the completion of my project I sincerely acknowledge his timely help, guidance and constant support.

I also express my immense gratitude to Dr. Ashok Vanageri, Principal Shetty institute of technology Gulbarga for his help and inspiration during the tenure of the course.

I express my gratitude to my family and friends for their moral support and cooperation which enable me to complete this project work

VIII. REFERENCES

[1] Kale, S.C., and Ganesan, V., (2004) " Investigation of the flow field in the various regions of intake manifold of a S.I. engine," Indian journals of Engineering and Material Sciences

[2] David Rathnaraj Jebamani and Thathapudi Michael Narendra Kumar, “Studies on Variable Swirl Intake System for Di Diesel Engine Using Computational Fluid Dynamics”,Thermal Science Vol 12 (2008), No.1 pp25-32.

[3] K.Mukund, M.Arun, N.Bharath Srinivas, A.S.Appushame, R.Rudramoorthy, “Cold Flow Simulation in Internal Combustion Engines Using Dynamic Mesh” Department of Production Engineering, PSG College of Technology, Peelamedu, Coimbatore, Tamil Nadu, India-641004

[4] Negin M., Reza E., Siamac H., “The Effect of Intake Manifold Runners Length on the Volumetric Efficiency by 3-D CFD Model,” SAE paper 2006-32-0118

[5] Marcelo R. C., Thomas M.M., “Correlation between Numeric Simulation and Experimental Results on Intake Manifold Development,” SAE paper 2009-36-0274

[6] Safari, M., Ghamari, M. and Nasiritosi, A., (2003) "Intake manifold optimization by using 3-D CFD analysis". SAE 2003-32-0073.

[7] Wangner Trindade. “Use of 1D-3D Coupled Simulation to Develop a Intake Manifold System,” SAE paper 2010-01-1534

[8] 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).

[9] 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).