International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 3 Issue 3, March - 2014

CFD Analysis of Exhaust Manifold of Multi-Cylinder Petrol Engine for Optimal Geometry to Reduce Back Pressure

[1]Vivekanand Navadagi, [2]Siddaveer Sangamad
[1]Assistant professor, dept of Mechanical Engineering SVERI'SCE Pandharpur, Maharastra, India
[2]Assistant professor, dept of Mechanical Engineering SVERI'SCE Pandharpur, Maharastra, India

***Abstract*— Exhaust manifold is one of the critical components of IC engine for improving the volumetric efficiency. The volumetric efficiency of the engine can be increased by reducing the backpressure in the exhaust manifold. This work analyzes the flow through two different models of exhaust manifold using CFD. The design of exhaust manifold is modified to get optimal geometry. The analysis results of two models are compared for back pressure. By comparing the results of two models the decrease in back pressure is found which ensure improvement in volumetric efficiency of the engine.**

Keywords*— *Exhaust Manifold, CFD, Multi-Cylinder SI Engine, Back Pressure

I. Introduction

An exhaust manifold collects the exhaust gases from multiple cylinders into one pipe. It is attached downstream of the engine and is major relevance in multi- cylinder engines where there are multiple exhaust streams that have to be collected into a single pipe. When an engine starts its exhaust stroke, the piston moves up the cylinder bore, decreasing the total chamber volume. When the exhaust valve opens, the high pressure exhaust gas escapes into the exhaust manifold or header, creating an exhaust pulse comprising three main parts: The high pressure head is created by the large pressure difference between the exhaust in the combustion chamber and the atmospheric pressure outside of the exhaust system. As the exhaust gases equalize between the combustion chamber and the atmosphere, the difference in pressure decreases and the exhaust velocity decreases. This forms the medium-pressure body component of exhaust pulse. The remaining exhaust gas forms the low pressure tail component. This tail component may initially match ambient atmospheric pressure, but the momentum of the high and medium pressure components reduces the pressure in the combustion chamber to a lower than atmospheric level. This relatively low pressure helps to extract all the combustion products from the cylinder and induct the intake charge during the overlap period when both intake and exhaust valves are partially open. The effect is known as scavenging. Length, cross-sectional area, and shaping of the exhaust ports and pipe works influences the degree of scavenging effect. *Seenikannan et al.* [1] analyzed a Y section exhaust manifold system experimentally to improve engine performance. This paper investigates the effect of using various models of exhaust manifold on CI engine performance and exhaust emission. *Yasar Deger et.al* [2] had done CFD-FE-Analysis for the Exhaust Manifold of a Diesel Engine aiming to determine specific temperature and pressure distributions. The fluid flow and the heat transfer through the exhaust manifold were computed correspondingly by CFD analyses including the conjugate heat transfer. *Dr. Kutaiba et.al* [3] made an approach to estimate of flow characteristic in inlet and exhaust manifolds of internal combustion engines using a four-stroke variable compression ratio single cylinder gasoline engine. In the experimental work, the compression ratio was varied from 7 to 11 at variable speed with constant throttle opening, where engine performance was obtained. *Scheeringa* [4] studied analysis of Liquid cooled exhaust manifold using CFD. Detailed information of flow property distributions and heat transfer were obtained to improve the fundamental understandings of manifold operation. A number of computations were performed to investigate the parametric effects of operating conditions and geometry on the performance of manifolds. *Gopal et al.* [5] has conducted experimental analysis of flow through the exhaust manifold of a multi cylinder Petrol engine of a contessa engine of 20 hp at maximum speed of 2000 rpm and then analyzed using FLUENT [1].

II. Back Pressure

Back pressure usually refers to the pressure exerted on a moving fluid by obstructions against its direction of flow. Back pressure caused by the exhaust system of an automotive engine has a negative effect on engine's performance as it will restrict the flow rate of the exhaust gasses. The result would be the engine not being able to expel the spent exhaust gasses fast enough to prevent spent exhaust gasses from contaminating the fresh air/fuel mixture that is drawn into the engine on the next intake stroke. Ultimately, this will result in reduced engine power.

III. CONSTRUCTION

The exhaust manifold system considered in the present case has 4 inlets connected to the exhaust port of the engines and a single outlet from where the flow is passed on to the exhaust system before ejection into the ambient. Due to lack of experimental data, an industrially available manifold geometry has been considered for the present analysis. It consists of the pipe diameter of 42 mm and total span of the manifold to be 0.6 m. The base model has the outlet placed besides the first port having smaller length of the curved pipe from the individual engine exhausts. The modified geometry has the outlet port placed in the middle with longer curved pipes from the individual engine exhausts, as shown in Figure.

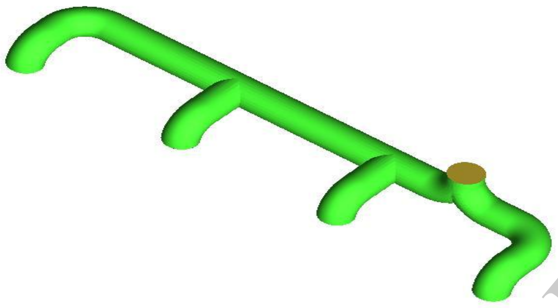


Fig-1: Base geometry of the exhaust manifold

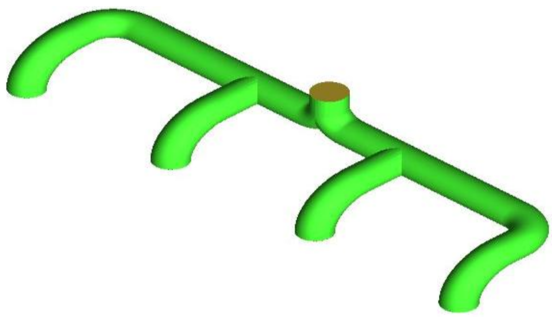


Fig-2: Modified geometry of the exhaust manifold

IV. METHODOLOGY

A steady state single phase single-species simulation with exhaust gas as the working fluid will be carried out for the two geometries at 4 different mass flow rates from each inlet to determine the pressure drop.

The geometry will be created using ANSYS ICEM CFD and a multi-block structured mesh will be created. ANSYS CFX will be used for doing the flow simulation under isothermal conditions. The overall analysis will be performed on ANSYS Workbench. A steady state single-species simulation will be carried out under isothermal conditions for exhaust gas. Turbulence will be modeled by *k*-ε RNG turbulence model appropriate to account for high velocities and strong streamline curvature in the flow domain. The reference pressure will be set at 1 atm and all pressure inputs and outputs will be obtained as gauge values with respect to this.

A. MATERIAL

Air will be the working fluid considered to be operating at 350 ^{0}C and 1.35 bar. The material properties under these conditions are:

Table-1: Properties of material

Property	Air
Density (kg/m^3)	0.7534
Viscosity (Pa.s)	3.0927 x 10^{-5}
Specific heat (J/kg-K)	1056.6434
Thermal conductivity (W/m-K) (not	0.0242

B. BOUNDARY CONDITIONS

The two models will be tested for 4 different engine loads corresponding to different mass-flow rates at each inlet:

Table-2: Boundary conditions

Cases	Engine speed(rpm)	Total exhaust gas flow rate (m^3/s)
i)	1300	3.143
ii)	1700	4.516
iii)	1800	4.981
iv)	1900	5.627

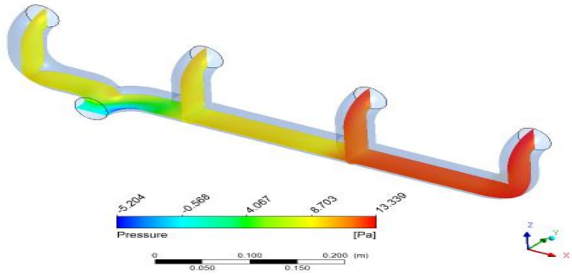
V. RESULTS AND DISCUSSIONS

The models are analyzed using CFX and the results obtained are shown in the color Contours

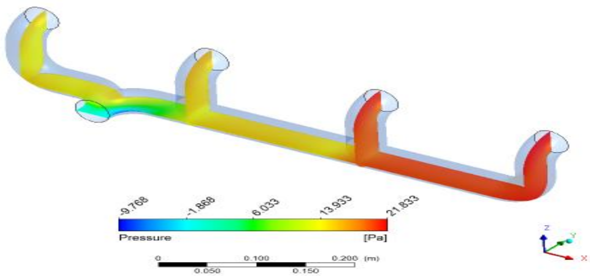
A. Pressure contours for the existing manifold

The high pressure at the head is due to high pressure difference from exhaust in combustion chamber and atmosphere pressure. Fig shows, the pressure was not uniformly distributed in the existing manifold and the effect of non-uniformly distributed pressure gives an impact on the velocity of the flow of exhaust gases. The pressure flow through the outlet of the exhaust manifold should be uniformly distributed and so the new manifold design is drawn.

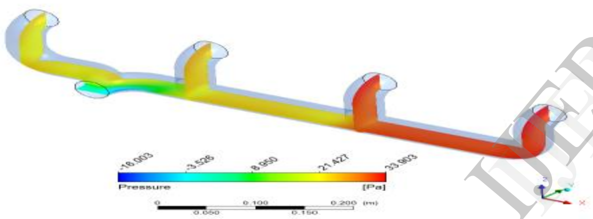
International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 3 Issue 3, March - 2014



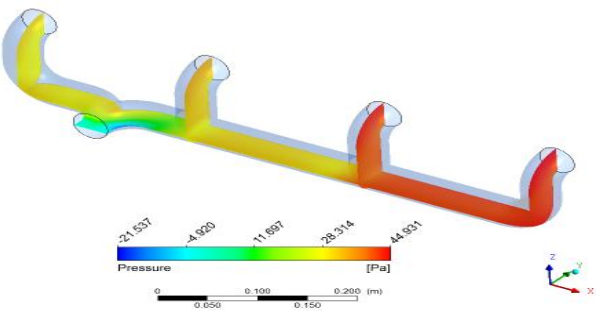
1300rpm



1700rpm

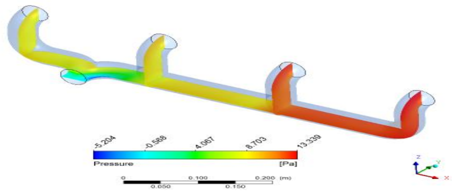


1800rpm

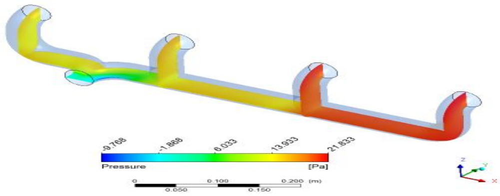


1900rpm

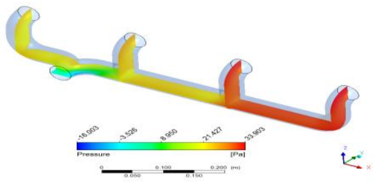
5.2 Pressure contours for the modified manifold



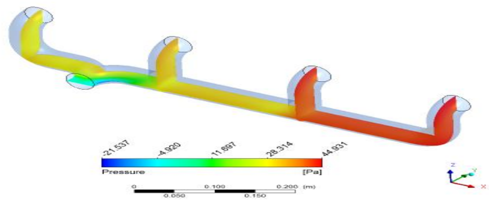
1300rpm



1700rpm

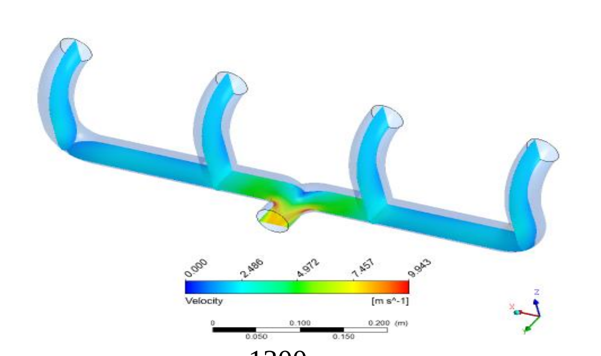


1800rpm

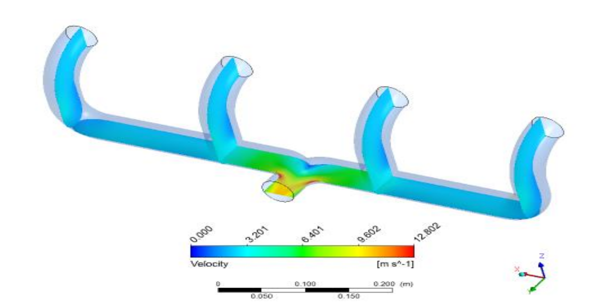


1900rpm

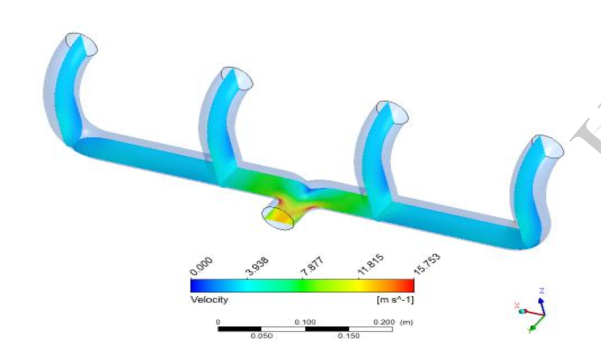
5.4 Velocity contours for the modified manifold



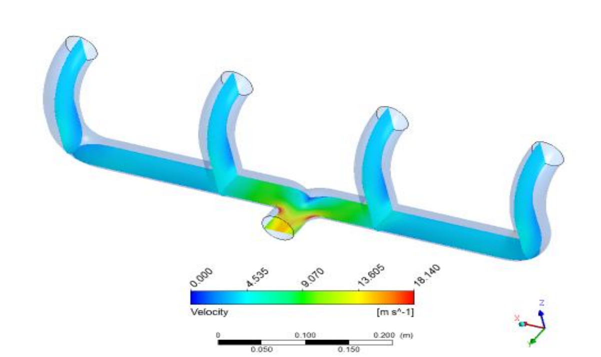
1300rpm



1700rpm

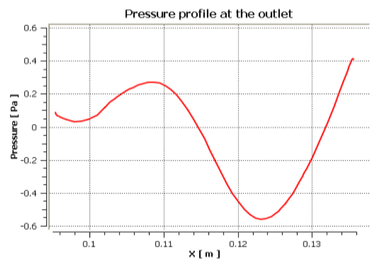


1800rpm

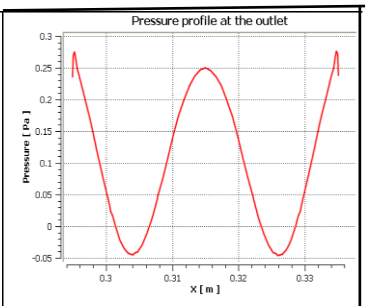


1900rpm

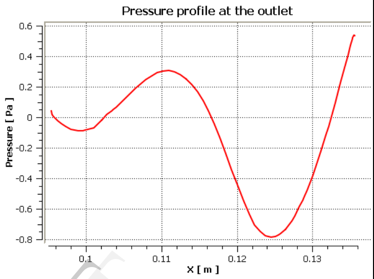
5.5 Pressure graphs



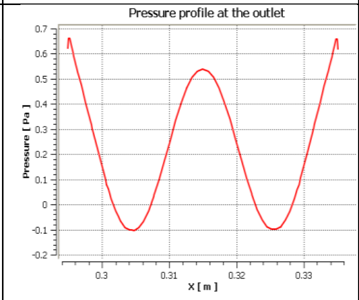
Base Model (1300 rpm)



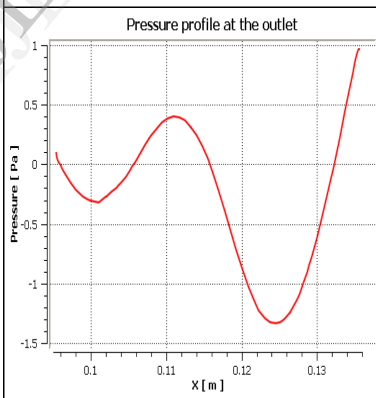
Modified Model (1300 rpm)



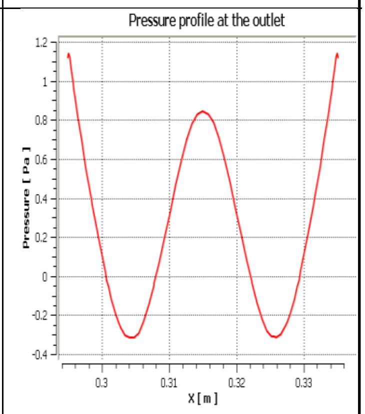
Base Model (1700 rpm)



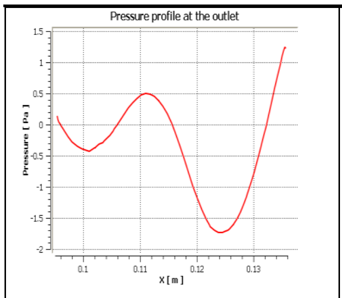
Modified Model (1700 rpm)



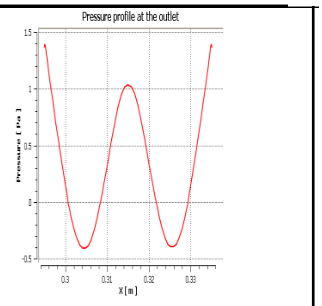
Base Model (1800 rpm)



Modified Model (1800 rpm)

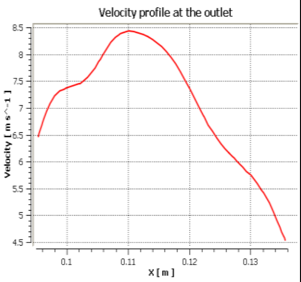


Base Model (1900 rpm)

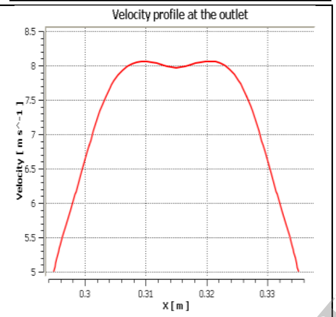


Modified Model (1900 rpm)

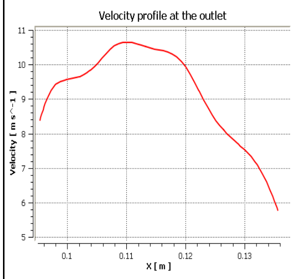
5.6 Velocity graphs



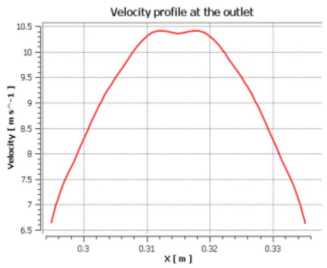
Base Model (1300 rpm)



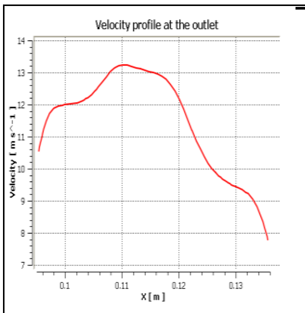
Modified Model (1300 rpm)

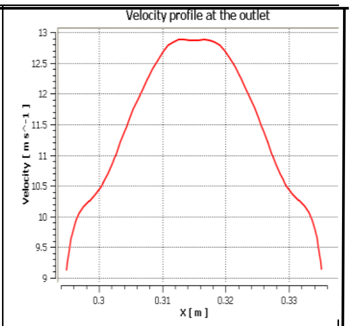


Base Model (1700 rpm)

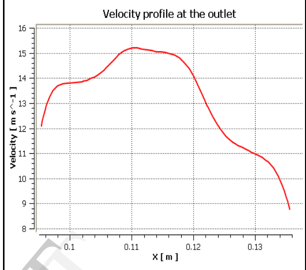


Modified Model (1700 rpm)

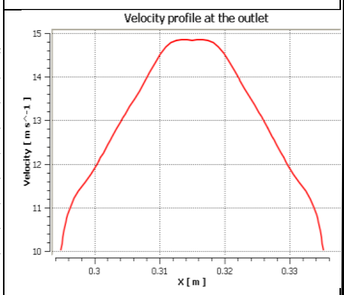




Modified Model (1800 rpm)



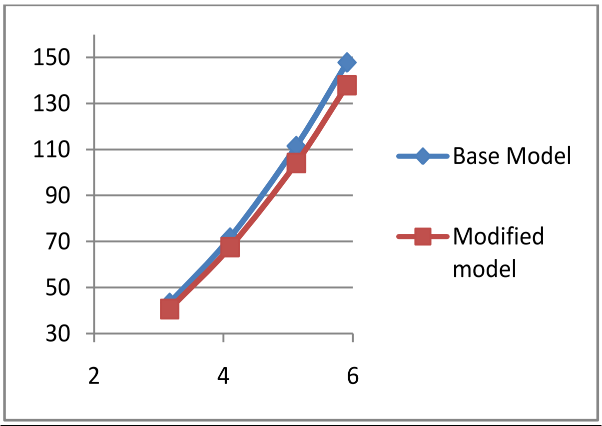
Base Model (1900 rpm)



Modified Model (1900 rpm)

5.7 Back Pressure Comparison

The graph shows that the back pressure is more in base model but it is less in case of modified model.



Mass flow rate

Fig-5.7: Comparison of Back Pressure

VI. CONCLUSION

The flow analysis of exhaust manifold was performed. The existing manifold is modified by changing its geometry to get optimal geometry. Both old and new models are analyzed under same boundary conditions. The results of new model are compared with the existing model. Pressure and velocity graphs are drawn for the new model and are compared with existing model. The decrease in back pressure is shown by using contour and vector diagrams. The flow is made efficient by decreasing the exhaust gas back pressure in the newly modified model thus increasing the volumetric efficiency of the engine.

REFERENCES

1. Seenikannan P., Periyasamy V.M. and Nagaraj P. "An experimental analysis of a Y section exhaust manifold system with improved engine performance".
2. Yasar Deger, Burkhard Simperl, and Luis P. Jimenez "Coupled CFD-FE-Analysis for the Exhaust Manifold of a Diesel Engine" Sulzer Innotec, Sulzer Markets and Technology Ltd, Winterthur, Switzerland.
3. Dr. Kutaiba J.M. AL-Khishali, Dr. Mahmoud A. Mashkour & Ehsan Shamil Omaraa "Analysis of Flow Characteristics In Inlet And Exhaust Manifolds of Experimental Gasoline Combustion In A VCR Engine".
4. Scheeringa K. **"**Analysis of Liquid cooled exhaust manifold using CFD".
5. Gopal P., Senthil Kumar T. and Kumaragurubaran B., "Analysis of Flow Through The Exhaust Manifold of a Multi Cylinder Petrol Engine for Improved Volumetric Efficiency" International Journal of Dynamics of Fluids ISSN 0973-1784 Volume 5 Number 1 (2009)
6. K.S. Umesh, V.K. Pravin and K. Rajagopal "CFD Analysis and Experimental Verification of Effect of Manifold Geometry on Volumetric efficiency and Back Pressure for Multi-cylinder SI Engine" International Journal of Engineering & Science Research IJESR/July 2013/ Vol-3/Issue-7/342-353