International Journal of Current Engineering and Technology
E-ISSN 2277 – 4106, P-ISSN 2347 – 5161
Available at http://inpressco.com/category/ijcet

Research Article

Geometry optimization of exhaust manifold using CFD

S.B.Girase*, R.J.Yadav, C.K.Patil[3], K.C.More and S.P.Shisode

Department of Mechanical Engineering, MITCOE,Pune

Accepted 12 March 2017, Available online 16 March 2017, **Special Issue-7 (March 2017)**

Abstract

*It is well known that a appropriately designed Exhaust manifold is important for the optimal performance of an IC engine. Though, the intake system is dominant on the cylinder filling process, the exhaust manifold has ability to influence the gas exchange process in various important aspects, like the piston work done during the exhaust stroke, the short-circuit of fresh charge from the intake into the exhaust. We know that if the back pressure of Exhaust gases is higher, it reduces efficiency of the engine. So aim should be to reduce the back pressure of the engine. Also amount of pressure drop is equal to back pressure. So ultimate goal is to reduce its pressure drop by changing its geometry, reducing friction in the manifold . In this paper only Geometry optimization is done. This paper presents 3-D simulation of Exhaust manifold of 4 cylinder, 4 Stroke, turbocharged diesel engine(*4G11 Greaves Engine) *of generator by using ANSYS-CFX code and results are discussed. A Steady state CFD simulations have been accomplished and experimental results are compared for validation. The CFD results gives an idea for changing its geometry.*

Keywords: *Exhaust Manifold, CFD Analysis, Optimization*

1. Introduction

The exhaust system of an internal combustion engine have a significant influence on the engine operation, not only in the field of acoustic emissions, but also in the domain of its performance and pollutant emissions. Among the different components of the system the manifold has a paramount relevance on the gas exchange process. It is imposing the boundary conditions on the cylinder at the exhaust valve and consequently it is able to heavily influence the exhaust flow. Also, it is affecting scavenging process during the valve overlap period, and even into the intake flow conditions at the start of the inlet process.

Though the intake system is dominating in the cylinder filling process, the exhaust manifold has ability to influence the gas exchange process in various important aspects, like the piston work done during the exhaust stroke, the mixing of fresh charge from the intake into the exhaust and even the charging of the cylinder. In this sense, the most influencing boundary condition imposed by the manifold is the back-pressure at the valve. The mean back pressure is determined mainly by the elements, such as the turbine, the silencer, catalytic converter. The instantaneous pressure generation imposed by the manifold at the exhaust valve depends mainly on the layout and dimensions of the pipes, therefore an appropriate design of the manifold geometry will improve the engine power and efficiency, and reduce the emissions of pollutants. Also the amount of pressure drop in exhaust system is equal to the amount of back pressure. So ultimate goal is the reduction of the pressure drop in exhaust system.

The field of engine design and analysis has been changed dramatically over the past few years with the fast growing computational technology. Computational Fluid Dynamics (CFD) has become the most popular design technique for the engine designers. CFD provides illustration of flows, where in the past, requiring significant investment on instrumentation and prototyping to obtain the equivalent results

In this paper, CFD analysis of the exhaust manifold of 4 cylinder, 4 stroke, turbocharged diesel engine (4G11 Greaves Engine) used for generator was done in ANSYS-CFX software. The CFD analysis provides illustration of flows i.e. pressure, velocity & temperature analysis. CFD simulation has been accomplished and experimental results are compared for validation. From this flow analysis, we can change the geometry of it to get optimal performance.[6]

2. Literature review and Problem statement

Governing Equations

CFD is based on fundamental governing equations of fluid dynamics,

*Corresponding author: **S.B.Girase**

- Continuity Equation,
- Momentum Equation
- Energy Equation.

These equations are solved for the analysis of exhaust manifold.

M. Safari *et al*. have worked on Inlet Manifold Optimization. He said in his paper Pressure drop in individual runners can be found by steady state analysis. It also gives internal Pressure variation in the manifold. For steady state condition, constant pressure boundary conditions are applied. Another purpose of this simulation is giving readiness for unsteady simulation of Exhaust manifold.

J. Benajes *et al*. has worked on Dimesions of Exhaust Manifold and its effect on back pressure and engine efficiency. The mean back pressure is determined mainly by the elements, such as the turbine, the silencer, catalytic converter. The instantaneous pressure generation imposed by the manifold at the exhaust valve depends mainly on the layout and dimensions of the pipes, therefore an appropriate design of the manifold geometry will improve the engine power and efficiency, and reduce the emissions of pollutants. Also the amount of pressure drop in exhaust system is equal to the amount of back pressure. So ultimate goal is the reduction of the pressure drop in exhaust system.

The standard k-ϵ model is used in industry which allows turbulent velocity and length scales to be independently determined.

Ansys CFX has given the Tutorials about CFD work has explained CFD Methodology

CFD methodology

CFD can be used to determine the performance of a component at the design stage or it can be used to analyze difficulties with an existing component and which will lead to its improved design .For example, the pressure drop through a component can be excessive:

The first step is to find the region of interest: The geometry of the region of interest is then defined. If the geometry already exists in CAD, it can be imported directly. The other elements of the simulation can be then defined, including the boundary conditions and fluid properties

The flow solver is then run to produce a file of results, which contains the variation of Pressure, velocity and any other variables throughout the region of interest.

The results can be visualized and provide the engineer with a complete understanding of the behavior of the fluid throughout the region of interest.

This can lead to design modifications, which can be tested by changing the geometry of the CFD model and seeing the effect.

Engine Specification of Exhaust manifold:

Engine:	4G11 Greaves Engine.
Application:	Generator.
No. of Cylinders:	Four Cylinder(Inline).
Cycle:	Four Stroke.
Firing Order:	1342.
Crank Position:	0,180,180,0.
Type:	Turbocharged, Diesel Engine.
Speed:	1500 r.p.m.
Power:	68 kw.
Exhaust valve Position:	Exhaust valve opens 33° BBDC & Closes 1° ATDC.

3. Experimental Validation of the Exhaust Manifold

The Internal Volume of the Component Which Is As Shown Below, is Prepared In Modeling Software And Imported.

Experimentation on Exhaust manifold was carried out. Mass Flow rate, Pressure, Velocity, Temperature were measured at four inlets named A, B, C, D and two outlets named E and F of the Exhaust Manifold.

Fig. 1 Internal volume of exhaust manifold

Experimental Results are as under

Table 1: Experimental Results

Location	Inlet A	Inlet B	Inlet C	Inlet D	Outlet E	Outlet F
Pressure (Gauge Pre.) (MPa)	0.141	0.142	0.139	0.137	0.130	0.130
Velocity (m/s)	22.15	21.96	21.08	20.68	62.16	61.06
Temperature (°K)	913	935	934	911	928	921
Mass Flow rate (gm/s)	25.9	26.7	26.2	25.1	52.1	51.7

Boundary conditions were applied to software. Mass flow rates and temperatures were applied to inlets A, B, C, D and Pressures were applied to outlets E and F. Following software results were obtained.

Simulation details

CFD Software used: ANSYS-CFX.
Meshing done in: ICEM CFD.
Mesh: Tetrahedral
Mesh quality:0.25 (industrial requirement is 0.25 to 0.3)
No. of Elements: 685303(Tetrahedral)
Type of CFD simulation: Steady state.
Fluid used: Air Ideal Gas
Turbulence model: k-epsilon model.
Scheme used: High resolution.
Time required for simulation: 1 hr.51 min.
Residual target: 10exp. –4
Reference pressure: 1 atm.
Heat transfer coeff.: 45 w/m2k
Surface roughness: 0.00508 mm.
Boundary conditions: At inlet A,B,C,D Pressures & Temperatures are applied & at outlet E,F Mass flow rates are applied.

Simulation gives following results

Table 2 ANSYS-CFX Results

Location	Location	Inlet A	Inlet B	Inlet C	Inlet D	Outlet E
Pressure (Gauge Pre.) (MPa)	0.131	0.131	0.131	0.131	0.130	0.130
Velocity (m/s)	19.46	19.92	19.39	18.96	56.16	54.42
Temperature (°K)	915	931	928	908	923	924
Mass Flow rate (gm/s)	26.0	26.4	26.1	25.3	52.4	51.4

Above results of Experimentation and CFX results are having variation of 10-15 %. So CFX Results are validated.

4. Post processing Results

For CFD analysis k-ε turbulence model, high-resolution scheme was used. From it contour plot of Pressure, Velocity, Temperature and streamline plot, vector plot were obtained as under.

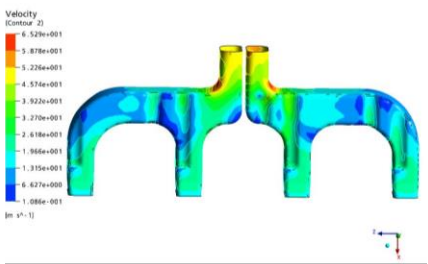

Fig. 2 Contour plot of Velocity

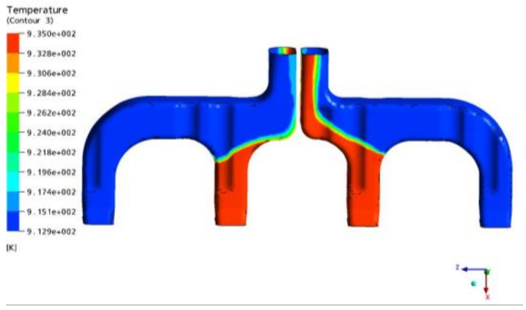

Fig. 3 Contour plot of Temperature

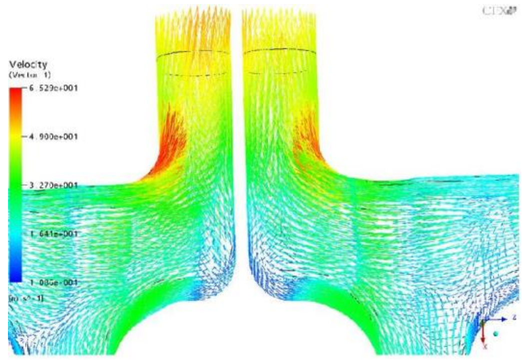

Fig. 4 Vector plot

5. Result and Discussion

3-D CFD analysis were done on exhaust manifold of 4 cylinder, 4 stroke, turbocharged diesel engine. A high resolution (second order) scheme with k - ε turbulence model were used.

Results observed from the software were experimentally validated. From the CFD analysis, we are getting pressure, velocity, and temperature analysis of the flow. From this we are getting an idea about the changes to be made in the manifold to optimize its performance. Exhaust manifold design parameters are

1) Properly design of exhaust manifold geometry to reduce its pressure drop.
2) Minimum possible resistance in the runners.
3) Eliminate the unnecessary turbulence and Eddies in exhaust manifold.
4) For the manifold average pressure values are calculated at four places namely Inlet A, Plane-1, Plane-2 and Outlet E as follows:

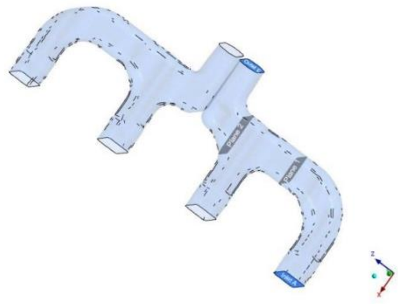

Fig. 5 Exhaust Manifold showing the locations

Results of pressure values obtained by the software are:

Table 3 Results of Pressure at the locations

Location	Average Pressure (Pa)
Inlet A	131510
Plane-1	131508
Plane-2	131172
Outlet E	130104

It is observed that there is highest pressure drop at the outlet. So the geometry near the outlet is to be improved.

So optimization suggestions are

1) To change the manifold geometry near the outlet where there is higher pressure drop.
2) Surface roughness of the internal surface is to be changed to make smooth surface.
3) Sharp changes in the direction of the flow can be changed within the limitations of engine layout, it will reduce the pressure drop.
4) If some changes can be made to guide the flow, performance of the manifold will be improved.

So Modified Shape 1 is as follows:

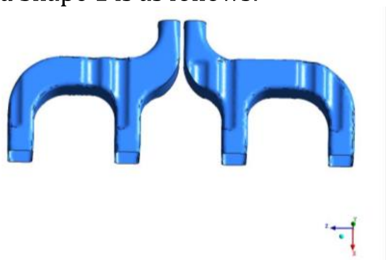

Fig 6: Modified Shape 1

Pressure drop obtained for modified shape1 by applying same simulation Procedure and boundary conditions is =

0.01237 bar = 1.237 kpa.
Net decrease in pressure drop is = 0.04 kpa.

By considering all the above suggestions Modified shape 2 is made as shown below.

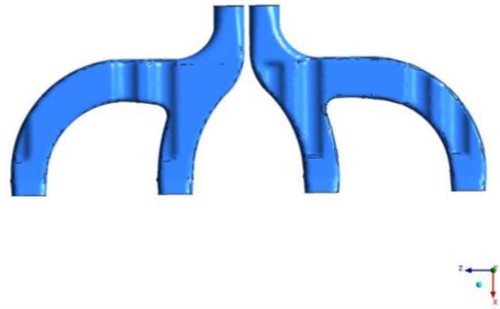

Fig7 Modified Shape 2

Pressure drop obtained for modified shape2 by applying same simulation Procedure and boundary conditions is =
0.01189 bar = 1.189 kpa.
Net decrease in pressure drop is = 0.088 kpa.

Conclusion

This paper has presented a CFD methodology to evaluate an exhaust manifold performance. The Results of CFD simulation have been explained in detail. From these results some changes in geometry are made to improve the performance of this exhaust manifold. Finally, it can be stated that from the experience of this work, 3 – D simulation can be used as a strong and useful tool for design or optimization of exhaust manifold.

References

Desmond E Winterbone & Richard J Pearson, "Design Techniques for Engine Manifolds", Professional Engineering Publishing, 1999, Page no. 213-217.

John D. Anderson, Jr. "Computational fluid dynamics – the basics with application", Mcgraw-Hill international Editions, 1995, page no. 146-239.

M. Safari, M. Ghamari and A. Nasiritosi, "Intake manifold optimization by using 3-D CFD analysis", SAE paper no. 2003-32-0073. page no. 1-5.

J. Benajes, E. Reyes, V. Bermudez and J. R. Serrano, "Pre-design Criteria for Exhaust Manifolds in I.C. Automotive Engines", SAE paper no. 980783. page no. 1-7.

Meda Lakshmikantha and Mathias Keck, "Optimization of Exhaust Systems", SAE paper no. 2002-01-0059. Page no. 1-3.

Website: http://www.ansys.com/cfx.