e-ISSN: 2582-5208
International Research Journal of Modernization in Engineering Technology and Science
(Peer-Reviewed, Open Access, Fully Refereed International Journal)
Volume:06/Issue:04/April-2024 Impact Factor- 7.868 www.irjmets.com

REVIEW: CFD ANALYSIS OF EXHAUST MANIFOLD OF MULTI CYLINDER SI ENGINE

Anjani Kumar[*1], Vijaykant Pandey[*2]

[*1]M-Tech Research Scholar, RKDF College of Technology, Bhopal (M.P.), India.

[*2]Assistant Professor, RKDF College of Technology, Bhopal (M.P.), India.

ABSTRACT

Spark ignition engines are employed in many different energy sectors and have become an integral element of civilization in the twenty-first century. Although they support transportation networks, they contribute significantly to air pollution. The main pollutants released from SI engines are CO, CO2, SOX, and NOX. There are essentially three categories of emissions that come from these engines: exhaust emissions, evaporative emissions, and crankcase emissions. The current project's goal is to lower pollution. It is a well-known fact that emissions are reduced via smooth combustion, and the exhaust process plays a major role in achieving this. In the current project, back pressures and exhaust velocities are assessed to improve the design of an exhaust manifold for a multi-cylinder spark ignition engine in order to reduce emissions.

Keywords: Exhaust Manifold, Pollutant, Back Pressure, Computational Fluid Dynamics.

I. INTRODUCTION

In an combustion chamber of internal combustion engine, combustion occur at high temperature and pressure due to which chances of piston seizure , overheating, chances of piston ring, compression ring, oil ring etc can be affected . Excess temperature can also damage the cylinder material. Due to overheating chances of pre-ignition also occurs. In Air cooled motorcycle engines heat release to the atmosphere through forced convection. The rate of heat transfer depends upon the wind velocity, geometry of engine surface, external surface area and the ambient temperature. In this work analysis is done on engine block fins considering temperature inside by means of conduction and convection, air velocity is not consider in this work. Motorbikes engines are normally designed for operating at a particular atmosphere temperature, however cooling beyond optimum limit is also not considered because it can reduce overall efficiency. Thus it may be observed that only sufficient cooling is desirable. Inside the cylinder the temperature of gases will be around 200^0 celsius . This is very high temperature and may result into burning of oil film between the moving parts this temperature must be reduced to about 150-200 at which engine will work more efficiently. The internal combustion engine is a type of engine in which a fuel is burned with an oxidizer (usually air) in a combustion chamber. The expansion of high-temperature and high-pressure gases produced by combustion gives direct force to a few components of an internal combustion engine, such as pistons, turbine blades, or a nozzle. This force propels the aspect forward, generating valuable mechanical electricity. Most modern-day internal combustion engines are cooled using a closed circuit of liquid coolant flowing through channels within the engine block, where the coolant absorbs warmth, to a warmth exchanger or radiator, where the coolant releases warmth into the air.

As a result, even though they are ultimately cooled by air, they are referred regarded as water-cooled due to the liquid-coolant circuit. In comparison, heat created by an air-cooled engine is released directly into the air. Typically, this is helped by metallic fins overlaid on the exterior of the cylinders, which increase the surface area on which air may act. In all combustion engines, a large proportion of the heat generated (approximately 44%) leaves via the exhaust, not via a liquid cooling mechanism or the metallic fins of an air-cooled engine (12%). Approximately 8% of the heat electricity finds its way into the oil, which, while generally intended for lubrication, also plays a role in heat dissipation via a cooler.

There are three types of heat transmission. The first is conduction, which is defined as heat transmission via a medium.

Without bulk motion of the substance, intervening should be counted. A stable has two floors, one at high and one at low temperatures. This type of heat conduction can occur in a jet engine, for example, through a turbine blade. The outside floor, which is exposed to gases from the combustor, is hotter than the inside floor, which has cooling air following it. Convection, or heat switch due to a flowing fluid, is the second heat transmission system.

The fluid can be a gas or a liquid, and both have uses in aircraft generation. The warmth is transferred by bulk transfer of a non-uniform temperature fluid in a convection warmness switch. The 0.33 process involves the transport of electrical through space without the presence of matter. Radiation is the most effective heat switch technique in the area. Even when there is an intervening medium, radiation can be critical; a common example is heat transfer from a gleaming piece of metal or from a fireplace.

Convective heat transfer between surfaces and surrounding fluid can be improved by introducing slender strips of metallic known as fins. Extended surfaces are another name for fins. When available surfaces are insufficient to transmit the needed amount of heat, fins can be employed. Fins are synthetic and come in a variety of sizes and shapes depending on the use. Air cooling for an integrated circuit The engine is a well-known example of an air cooling system in which air serves as a medium. Heat generated in the cylinder can be dissipated into the environment via conduction mode via the fins or extended surfaces used in this device, which can be included around the cylinder.

Figure 1- SI Engine

II. LITERATURE REVIEW

PL. S. Muthaiah et.al [1], He has analyzed the exhaust manifold in order to reduce the backpressure and also to increase the particulate matter filtration. He has modified the different exhaust manifold by varying the size of the conical area of the exhaust manifold and varying the size of the grid wire mesh packed throughout the exhaust manifold. When size of the grid mesh packed decreased the backpressure increases which leads to lower the performance of the engine due to more fuel consumption and hence low volumetric efficiency. When size of the grid mesh packed increased the backpressure decreases the filtration of the particulate matter also reduces which will not satisfy the standards of the pollution control. Computational fluid dynamics is used for the study of the exhaust manifold and best possible design of the exhaust manifold with minimum backpressure and maximum particulate matter filtration efficiency is suggested.

K.S. Umesh, V.K. Pravin and K. Rajagopal et.al. [2] In this work eight different models of exhaust manifold were designed and analyzed to improve the fuel efficiency by lowering the backpressure and also by changing the position of the outlet of the exhaust manifold and varying the bend length. The eight different modified models are short bend centre exit (SBCE), short bend side exit (SBSE), long bend centre exit (LBCE), long bend side exit (LBSE), short bend centre exit with reducer (SBCER), short bend side exit with reducer (SBSER), long bend centre exit with reducer (LBCER), long bend side exit with reducer(LBSER).After analysis they included that the exhaust manifold with long bend centre exit with reducer (LBCER), gives the highest overall performance.

Kulal et al. [3]work comprehensively analyzes eight different models of exhaust manifold and concluded the best possible design for least fuel consumption. CFD is the current trend on automotive field in reducing the cost effect for analysis of various models on the basis of fluid flow. A multi-cylinder Maruti - Suzuki Wagon-R engine with maximum speed of 1500 rpm is taken for the analysis. The load and performance test is conducted. From the experiment back pressure and exhaust temperatures are measured. The mass flow rate and velocities are calculated. Flow through the exhaust manifold is analyzed using commercially available software with mass flow rate and pressure as boundary conditions.

Jafar M Hassan et.al. [4] had analyzed the performance of the manifolds with a tapered longitudinal section. The length of the manifold for this study was 127 cm while the manifold diameter was 10.16 cm. Authors had used the numerical simulations (CFD) for this research work. The flow conditions corresponding to Re = 10x104, 15x104 and 20x104 were considered. The results were analyzed in terms of uniformity coefficient. Based on their CFD simulation results, they had concluded that the tapered header configuration provides better flow distribution as compared to the header with circular cross-section.

M.Usan et.al. [5] had applied a multi-disciplinary optimization approach for the exhaust system, exhaust manifold and catalytic converter, in highly integrated concurrent engineering software framework. They had considered four-cylinder 1.4 litre engine as a baseline model. The optimization contained four major modules – Geometry, Structural, Cost and Fluid Dynamics – and the relevant software for each module was applied. 1-dimensional transient CFD simulations were carried out using AVL BOOST with the engine torque and catalytic converter inlet temperature over the engine rpm were being estimated.

Hessamedin Naemi et.al. [6] had employed numerical simulations (CFD methods) for estimating the flow loss coefficient in manifolds. The flow inlet and exit was model using 'mass-flow-inlet' and 'pressure-outlet' boundary conditions, with the consideration that the flow was compressible. The results from different turbulence models – standard k-ε, standard k-ω, SpalartAllmaras model and RNG k-ε model – were compared in terms of flow loss coefficient against the experimental data. Based on their results, the authors had observed that the RNG k-ε turbulence model predictions were in close agreement with the experimental data.

Sujan Shrestha et.al. [7] The shape of the heating movement of internal combustion engines can be shown by different techniques. These techniques range from original worm systems to multidimensional differential condition representation. Blades are installed on the outside of the chamber to increase heat retention due to convection. Thermal studies of engine compartment blades are increasingly important to understand the heat dissipated in the compartment. An authored study shows that heat transfer improves with a wider surface and the heat transfer coefficient is affected by changes in blade cross-section. This research helps identify a better balance of geometry and materials for greater heat dissipation and engine cooling. We now use common materials like dark cast iron for engine square.

Naman Sahu et. al [8] The engine housing is one of the essential components of the engine and is exposed to various high and high temperatures. Blades are installed on the outside of the chamber to increase heat retention due to convection. Thermal inspection of motor housing blades is increasingly useful for understanding the heat dissipated within the housing. The current survey was conducted to improve information on various recent surveys. This shows that blade heat transfer depends on balance composition, balance pitch, balance design, wind speed, texture and atmospheric environment. Written experiments have shown that the heat transfer is improved by the extended surface and the heat transfer coefficient is affected by changes in the equilibrium cross-section. This research helps identify geometries and materials that balance higher heat dissipation and engine cooling.

Charan et. al. [9] We have broken down a broad surface that is commonly used to promote convective heat transfer in a wide variety of design applications. The holes in the parallel sides of the blades are suitable for improving the speed of heat transfer. As a result of the investigation, it was found that aluminum materials with three triangular holes had the lowest tip temperature and aluminum materials with three triangular holes had the highest heat transfer. As a result of the inquiry, it was found that the Nusselt number in the clamped scale increases when the blade is jammed in front of the blade that is jammed. Therefore, 3 aluminum triangles with horizontal holes are generally considered suitable for balance applications.

3. CFD

Computer primarily based simulation is mentioned during this chapter. procedure simulation is technique for examining fluid flow, heat transfer and connected phenomena like chemical reactions. This project uses CFD for analysis of flow and warmth transfer. CFD analysis accepted go in the various industries is employed in R&D and producing of craft, combustion engines and in powerhouse combustion similarly as in several industrial applications.

Why computational simulation

Three-dimensional (3D) numerical analysis of whorled coil tubes is dispensed by victimization business CFD tool ANSYS 18.2. this can become troublesome and time overwhelming, if this analysis is dispensed by experimentation. Experimental setup is extremely expensive that's why in my work I take facilitate of CFD to create it easier and fewer time overwhelming.

Computational fluid dynamics

Computational fluid dynamics, because the name implies, could be a subject that deals with procedure approach to fluid dynamics by means that of a numerical resolution of the equations that cause the fluid flow and though it's known as procedure fluid dynamics; it doesn't simply wear down the equations of the fluid flow, it's conjointly generic enough to be ready to solve at the same time along the equations that direct the energy transfer and similarly the equations that verify the chemical process rates and the way the chemical process takings and mass transfer takes place; of these things may be tackled along in a regular format. So, this define permits America to wear down a really complicated flow circumstances in fairly quick time, specified for a specific set of conditions, associate degree engineer would be ready to simulate and see however the flow is happening and what quite temperature distribution there's and what quite product area unit created and wherever they're fashioned, in order that {we can|we will|we area unit able to} build changes to the parameters that area unit below his management to switch the approach that these items are happening. So, therein sense procedure fluid dynamics or CFD becomes a good tool for a designer for associate degree engineer. it's conjointly a good tool for associate degree associate degreealysis for associate degree examination of a reactor or an instrumentality that isn't functioning well as a result of in typical industrial applications, several things is also happening associate degreed what a designer has had in mind at the time of fabricating or coming up with the instrumentality won't be really what an operator of the instrumentality introduces into the instrumentality at the time of operation, perhaps once 5 years or 10 years changes might need taken place in between; and in such a case, the presentation of the instrumentality won't be up to the quality and you'd wish to modify it in such some way that you just will restore performance. So, the question is then, what this can managed to the autumn within the performance associate degreed what quite measures we are able to build while not creating an overall adjustment within the finish of apparatus. Is it potential to urge improved performance from the equipment? Is it potential to extend the productivity? If you wish to appear on of these analysis, then procedure fluid dynamics is employed.

III. CONCLUSION

This work is devoted to the evaluation of different models of exhaust manifold for the purpose of reducing exhaust emissions from a four-cylinder SI engine. For this purpose, a set of eight alternatives was chosen, and modelled with the help of CATIA V5 modelling software. In next stage, CFD of different models were carried out based on k-ε model, which finally yield the values of back pressures, and exhaust velocities at different loading conditions. After that performance score was calculated for both the parameters, and as the last step of project overall performance score for different types was calculated.

IV. REFERENCES

[1] Pudiri Madhu and N. Sateesh., An experimental investigation into the Modeling and Simulation of Fins for 150cc Engine, SAE paper R. Caves, Multinational Enterprise and Economic Analysis, Cambridge University Press, Cambridge, 1982. International Journal of Recent Development in Engineering and Technology.

[2] D. Merwin Rajesh and K Suresh Design and thermal analysis of cylinder fins by varying its geometry and material, IJME International Journal, ISSN NO: 2348- 4845 (2014)

[3] Thornhill D. and May A., An Experimental Investigation into the Cooling of Finned Metal Cylinders in a free Air Stream, SAE Paper 1999-01-3307 (1999)

[4] Gibson H., The Air Cooling of Petrol Engines, Proceedings of the Institute of Automobile Engineers, Vol.XIV, 243-275 (1920)

[5] Biermann E. and Pinkel B., Heat Transfer from Finned Metal Cylinders in an Air Stream, NACA Report No. 488 (1935)

[6] Masao Yoshida, Soichi Ishihara, Yoshio Murakami, Kohei Nakashima and Masago Yamamoto, Air-Cooling Effects of Fins on Motorcycle Engine, JSME International Journal, Series B, 49(3), (2006)

[7] Zakhirhusen, Memon K., Sundararajan T., Lakshminarasimhan V., Babu Y.R. and Harne Vinay, Parametric study of finned heat transfer for Air Cooled Motorcycle Engine, SAE Paper, 2005-26-361, (2005)

[8] Zakirhusen, Memon K. and Sundararajan T., Indian Institute of Technology Madras, V. Lakshminarasimhan, Y.R. Babu and Vinay Harne, TVS Motor Company Limited, Simulation and Experimental Evaluation of Air Cooling for Motorcycle Engine, 2006-32-0099 / 20066599 (2006)

[9] Pathak Sunil, Turbo charging and oil techniques inlight motor vehicles, Res.J. Recent Sci, 1(1), 60-65 (2012)

[10] Dev Nikhil, Attri Rajesh, Mittal Vijay, Kumar Sandeep, Mohit, Satyapal, Kumar pardeep, Thermodynamic analysis of a combined heat and power system, Res.J. Recent Sci, 1(3), 76-79 (2012)

[11] G.Raju, et al. (2012). "Optimal Design of an I.C. Engine Cylinder Fin Arrays Using a Binary Coded Genetic Algorithms." International Journal of Modern Engineering Research (IJMER) 2(6): 4516-4520.

[12] U. Magarajan, et al. (2012). "Numerical Study on Heat Transfer of Internal Combustion Engine Cooling by Extended Fins Using CFD." International Science Congress Association 1(6): 32-37.

[13] A. Mishra, et al. (2012). "Heat Transfer Augmentation of Air Cooled Internal Combustion Engine Using Fins through Numerical Techniques." Research Journal of Engineering Sciences ISSN 2278:9472.

[14] J.A. Paul, et al. (2012). "Experimental and Parametric Study of Extended Fins in the Optimization of Internal Combustion Engine Cooling Using CFD." International Journal of Applied Research in Mechanical Engineering (IJARME) 2(1).

[15] G. Babu and M. Lavakumar (2013). "Heat Transfer Analysis and Optimization of Engine Cylinder Fins of Varying Geometry and Material." IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) 7(4): 24-29.

[16] S.S.Chandrakant, et al. (2013). "Numerical and Experimental Analysis of Heat Transfer through Various Types of Fin Profiles by Forced Convection." International Journal of Engineering Research & Technology (IJERT) 2(7).

[17] A.R. Kumar, et al. (2013). "Heat Transfer Analysis in the Cylinder Head of a Four-Stroke SI Engine." International Journal of Engineering Research & Technology (IJERT) 2(5).

[18] R.P. Patil and P. H. M. Dange (2013). "Experimental and Computational Fluid Dynamics Heat Transfer Analysis on Elliptical Fin by Forced Convection." International Journal of Engineering Research & Technology (IJERT) 2(8).

[19] N.P.R. Rao and T. V. Vardhan (2013). "Thermal Analysis of Engine Cylinder Fins By Varying Its Geometry and Material." International Journal of Engineering 2(8).

[20] S. Wange and R. Metkar (2013). "Computational Analysis of Inverted Notched Fin Arrays Dissipating Heat by Natural Convection." International Journal of Engineering and Innovative Technology (IJEIT) 2(11).

[21] V. GANESAN, (2008). I C Engines, McGraw-Hill Education (India) PVT Limited.