International Journal of Research Publication and Reviews

Journal homepage: www.ijrpr.com ISSN 2582-7421

A Literature Review on Design of Exhaust Manifold of A Engine

Mr Puspa Datta B

Department of Mechanical Engineering, Vidyavardhaka Polytechnic, Mysuru-570007, India

A B S T R A C T

Exhaust manifolds collect exhaust gases from engine cylinders and release them into the atmosphere via the exhaust system. The efficiency of the engine and the combustion characteristics are largely determined by how the burned gases are evacuated from the cylinder. One of the most important parts of an IC engine is the exhaust manifold. The design of an exhaust manifold is a complicated process that is influenced by a number of factors such as exhaust back pressure, mechanical efficiency, and exhaust gas velocity, among others. Recent studies on exhaust manifold design and performance evaluation using experimental and numerical methods are discussed in this literature review (CFD). The influence of different geometrical forms of exhaust manifolds on performance has been investigated and debated..

Keywords: Exhaust Manifold, Internal combustion (IC) Engine, CFD analysis, Fluid Flow.

1. INTRODUCTION

Exhaust manifolds are a pipe system that gathers and discharges exhaust gases from engine cylinders into the atmosphere. How the exhaust gases were evacuated from the cylinder would affect the engine's efficiency and combustion characteristics. Exhaust manifolds are typically made of cast iron or steel, though stainless steel is now being used in certain newer models.The following are some of the most critical components of the IC engine's exhaust manifold design.The exhaust manifold should be designed to keep the exhaust pipe at a high temperature. This is required because a catalyst near the exhaust pipe's end would absorb more pollution in high-temperature settings.To avoid any damage, the design should guarantee that the natural frequencies of the exhaust manifold do not overlap with the excitation frequency range of engine vibration.The exhaust manifold's bulk should be as low as possible.The summary of recent works on the design of exhaust manifolds and their problems, performance assessment methods and procedures, and essential aspects have been published in this publication.

2. LITERATURE REVIEW

Using CFD simulation of a multi-cylinder engine's exhaust manifold, A.VinayGopal et al.[1] studied optimal geometry for decreasing emissions. The main goal is to design an exhaust manifold and determine pressures and velocities in exhaust manifolds with Long Bend Side Exit (LBSE), Long Bend Middle Exit (LBME), and reducers at various mass flow rates, as well as to determine the performance of the exhaust manifold with various changes in its design by adding a component for the exhaust manifold to increase its effectiveness. The mass flow rates in the exhaust manifolds included in this analysis are 2 kg/s, 4 kg/s, 6 kg/s, 8 kg/s, 10 kg/s, and 12 kg/s for all the different modifications in the exhaust manifolds via CFD analysis. According to the results of the preceding investigations, the Long Bend Middle Exit (LBME) with Reducer provides the best performance.

In the fluid flow assessments for each manifold, merCihan et al.[[2] employed ABAQUS V6.13 to do Computational fluid dynamic calculations for three different fluid materials such as gasoline, alcohol, and LPG. Pressure, velocity, and temperature variations were numerically investigated at various places along the manifolds. Type A exhaust manifolds had higher pressure than type A exhaust manifolds, according to the results. High pressure values of type A were attained for all types of fuels, which improved the engine's performance and efficiency.

The performance of manifolds having a tapered longitudinal section was studied by Jafar M Hassan et al.[[3]. The manifold used in this study has a length of 127 cm and a diameter of 10.16 cm. For this investigation, the authors employed numerical simulations (CFD). The flow conditions for Re = 10x104, 15x104, and 20x104 were taken into account. The uniformity coefficient was used to determine the results. They determined that the tapered header arrangement provides superior flow distribution than the circular cross-section header, and that the flow distribution in the exhaust manifold channels is heavily reliant on the header shape and flow rate, based on their CFD simulation results.

The thermal and structural analysis of a multi-cylinder engine exhaust manifold was examined by Gopaal et al. [[4]. Thermal and coupled field analysis are carried out in this work, and critical frequencies in the operating range are found using Modal analysis. Deflections and stresses at the nearest natural frequencies have been displayed after a harmonic analysis. The acceptability of the exhaust manifold design is based on the results of many analyses. It's an attempt to automate design optimization in order to lessen the technical, scheduling, and economic risks associated with new engine development.

M.Usan et al.[[5] used a highly integrated concurrent engineering software framework to apply a multi-disciplinary optimization method to the exhaust system, exhaust manifold, and catalytic converter. As a starting point, they considered a four-cylinder 1.4-liter engine. Geometry, Structural, Cost, and Fluid Dynamics were the four primary modules in the optimization, and the appropriate software was used for each. The engine torque and catalytic converter inlet temperature were determined over the engine rpm using 1-dimensional transient CFD simulations with AVL BOOST.

For calculating the flow loss coefficient in manifolds, HessamedinNaemi et al.[[6] used numerical simulations (CFD methods). The flow intake and exit were modeled using the boundary conditions'mass-flow-inlet' and 'pressure-outlet,' with the assumption that the flow was compressible. The results of various turbulence models – standard k-, standard k-, SpalartAllmaras model, and RNG k- model – were compared to experimental data in terms of flow loss coefficient. The authors concluded that the RNG k-turbulence model predictions were in close accord with the experimental data based on their findings.

Masahiro Kanazaki et al.[[7] used the Divided Range Multi-objective Genetic Algorithm to design a multi-objective optimization approach for the exhaust manifold.

The transient, Euler flow solver was used to model the three-dimensional fluid dynamics inside the manifold. The optimization's two objective functions were to I maximize exhaust gas temperature at the end of the exhaust pipe and ii) maximize charging efficiency. The authors were able to optimize the manifold for both of these target functions, and the optimized model had more engine power than the baseline model.

For calculating the engine performance of a single cylinder IC engine, Hong Han-Chi et al.[[8] employed GT-Power, a 1-dimensional software. The software's anticipated power output was compared to the experimental data. The authors took into account four criteria in their research: the sphere style plenum diameter, intake runner diameter, exhaust runner lengths, and restrictor location. Helmholtz theory was used to construct the plenum for the intake and exhaust manifold. The Orthogonal Array Testing Strategy was used to conduct the optimization experiment (OATS). The results of the experimental investigation were found to be in good accord with the predictions made by the GT-Power software.

The application of multiple CFD approaches by Ahmed et al. in this study work comprises of identifying the best exhaust manifold for a 4-cylinder, 4-stroke petrol engine. Five header alternatives based on manifold pipe geometry are investigated in the research work: convergent inlet pipe, decreased convergent length, divergent-straight-convergent, reduced, divergent length, and increased convergent length, ideal divergent and convergent, and reduced straight length. Unstructured meshes were used in the simulations, which were run using the simulation and analysis program ANSYS FLUENT. The boundary condition employed in the research was mass flow inflow. Researchers concluded that reducers can be used to achieve the lowest back pressure at the exhaust manifold based on their findings.

Xueyuan Zhang et al.[[9] performed a combined thermo-fluid-solid study of an exhaust system while taking welding stresses into account. The authors examined an operating state of 302 kg/hour exhaust gas flow rate at 870 C. ANSYS FLUENT was used to run the CFD simulations. The FEA solver was given boundary conditions based on the heat profiles produced from the exhaust manifold simulations. The welding residual stresses in the manifold were analyzed using this method, and appropriate design improvements were indicated. By preheating the substrate to the proper temperature, the IC engine's cold start emissions might be decreased. The catalytic converter could be placed near the exhaust manifold to achieve this.

CFD study was used by Simon Martinez-Martinez et al.[10] to predict the performance of the exhaust manifold while placing the catalytic converter close to it (Close-Coupled Catalytic Converter). Cast manifold, 4-2-1 manifold, and L-Shaped manifold were the three options they evaluated. The flow at the exhaust manifold was 90 g/s mass flow with a gas temperature of 900 C. The flow uniformity index and the total pressure drop were used to evaluate the manifold's performance. Although the flow uniformity index (0.96) was nearly same in all three types of manifolds, the flow losses (total pressure drop) were significantly different. The L-shaped manifold had the lowest flow losses, while the 4-2-1 manifold had flow losses that were 50 percent higher.

Thermal analysis for the tubular type ICEngine exhaust manifold was performed by S. N. Ch. Dattu. V et al. [11] for various running situations. Radius 48 mm Exhaust Valve at Extremely Left, Radius 48 mm Exhaust Valve at Center, Radius 100 mm Exhaust Valve at Extremely Left, Radius 100 mm Exhaust Valve at Center were also considered by the authors. These setups were tested on two different types of materials: Cast Iron and Aluminum. The authors recommend aluminum for the exhaust manifold based on the findings of their simulations because of its greater thermal performance. Also included is a 48 mm radius exhaust manifold with the exhaust valve in the center.

CFD calculations on the manifold of a direct injection diesel engine were performed by Benny Paul et al. [12]. For the simulations, they employed the RANS (Reynolds Averaged Navier Stokes) solver with the RNG k-turbulence model. 'pressure-inlet' was used to model the flow inlet for the manifolds. The manifold's wall areas were considered adiabatic, and a No-Slip condition was applied. The authors conducted a grid-independence investigation with three different meshes to ensure that the numerical solutions derived from CFD simulations were independent of the grid size. For this research, STAR-CD was employed, and the meshes were created utilizing Gambit.

MohdSajid Ahmed et al.[[13] used CFD to find the best exhaust manifold for a four-stroke four-cylinder SI engine. Based on the manifold pipe geometry, they considered five different types of exhaust manifolds: convergent inlet pipe, divergent-straight-convergent, reduced convergent length and increased divergent length, reduced divergent length and increased convergent length, identical convergent and divergent, and reduced straight length. The CFD simulations were run with unstructured meshes in ANSYS FLUENT. The flow inlet was modeled using the'mass flow inlet' boundary condition. The authors indicated that reducers might be used to obtain the lowest back-pressure at the exhaust manifold outlet based on their findings. Thermal strains will be induced on the exhaust manifold surfaces due to the high temperature exhaust gases.

.

3. CONCLUSION

The results of the CFD study and the experimental methods for estimating the performance of the exhaust manifold are in good agreement. The flow and temperature fields in the exhaust manifolds are predicted substantially closer to the experimental values with minimum error using the relative k-epsilon turbulence model. The details of thermal stress distribution on the manifold surfaces might be obtained using a combined numerical study of thermo-fluid structural analysis. This could aid in determining the exhaust manifold's potential failure zones. The shape of the runner, as well as its orientation, such as the bend radius, L-shaped, and curved runners, has a considerable impact on the exhaust manifold's performance. Exhaust manifolds with reducers are used to achieve low backpressure and higher exhaust velocity, while also lowering pollutants.

REFERENCES

A.Vinay Gopal, J Bala bhaskar Rao, “CFD Analysis of Exhaust Manifold of Multi Cylinder SI Engine to Determine Optimal Geometry for Reducing Emissions” International Journal & Magazine of engineering, Technology, Management & Research, Volume 4, Issue 5, February 2017

Ömer Cihan , Mehmet Bulut, “CFD Analysis of Exhaust Manifold for Different Designs” European Mechanical Science, December 2019; 3(4): 147-152

Jafar M Hasan, Wahid S Mohammad, Thamer A Mohamed, Wissam H Alawee, “CFD Simulation for Manifold with Tapered Longitudinal Section” International Journal of Emerging Technology and Advanced Engineering, Volume 4, Issue 2, February 2014

Gopaal, M M M Kumara Varma, Dr. L Suresh Kumar, “Thermal And Structural Analysis Of An Exhaust Manifold Of A Multi Cylinder Engine”, IJMET Volume 5, Issue 12, December 2014

M. Usan, O. de Weck, D. Whitney, “Exhaust System Manifold Development Enhancement through MultiAttribute System Design Optimization”, American Institute of Aeronautics and Astronautics

HessamedinNaeimi, DavoodDomiryGanji, MofidGorji, GhasenJavadirad and MojtabaKeshavarz, “A Parametric Design of Compact Exhaust Manifold Junction in Heavy Duty Diesel Engine Using Computational Fluid Dynamics Codes” Thermal Science, Volume-15, No. 4, 2011

Masahiro Kanazaki, Masashi Morikawa, Shigeru Obayashi and Kazuhiro Nakahashi, “Exhaust Manifold Design for a Car Engine Based on Engine Cycle Simulation” International Conference Parallel Computational Fluid Dynamics, Japan, May 2002

Hong Han-Chi, Huang Hong-Wu, Bai Yi-Jie, “Optimization of Intake and Exhaust System for FSAE Car Based on Orthogonal Array Testing” International Journal of Engineering and Technology, Volume 2, No. 3, March 2012

Xueyuan Zhang, Yu Luo, Jianhua Wang, “Coupled Thermo-Fluid-Solid Analysis of Engine Exhaust Manifold Considering Welding Residual Stresses” Transactions of JWRI, 2011L

Simon Martinez-Martinez, Ruben D. Leal-Garza, Fausto A. Sanchez-Cruz, Esteban Baez Villareal, “CFD Analysis of the effect of the Exhaust Manifold Design on the Close-Coupled Catalytic Converter Performance” Journal of KONES Powertrain and Transport, vol-17, No.4, 2010

S. N. Ch. Dattu. V, M. PradeepVarma, B. ShyamSundar, “Thermal Analysis on 4-1 Tubular Type IC-Engine Exhaust Manifold through Analysis” International Journal of Advanced Mechanical Engineering, volume-4, Number 7, 2014

Benny Paul, V. Ganesan, “Flow field Development in a Direct Injection Diesel Engine with Different Manifolds” International Journal of Engineering, Science and Technology, Vol-2, No.1, 2010

Gopaal, M.M.M Kumara Varma, L. Suresh Kumar, “Exhaust Manifold Design – FEA Approach” International Journal of Engineering Trends and Technology, Volume-17, Number 10, November 2014

MohdSajid Ahmed, Kailash B.A., Gowreesh,”Design and Analysis of a Multi-Cylinder Four Stroke SI Engine Exhaust Manifold Using CFD Technique” International Research Journal of Engineering and Technology, volume-2, Issue-9, December 2015