Internal combustion engines are typically equipped with an exhaust muffler to suppress the acoustic pulse generated by combustion process. Backpressure is essential for the performance of muffler. Backpressure is the difference between the mean exhaust pressure and the ambient pressure and is due to drop in stagnation pressure across various perforated element and area discontinuities. All engines have a maximum allowable engine backpressure specified by the manufacturer of the engine. Sometimes it will be high so that may cause negative effect on engine efficiency resulting in a decrease in power output and therefore increasing the fuel consumption. So if the backpressure will be reduced the engine efficiency will be high. The aim of this project is to reduce the backpressure of the exhaust muffler. For this four muffler models are modeled by using CATIA V5 software and the Computational Fluid Dynamics analysis of these muffler models are carried in the ANSYS FLUENT software. By using these software, results of the backpressure of these new modeled mufflers will be compared with existing one. Based on the various boundary conditions used for CFD analysis we have selected the optimum model between four models.

Keywords- Muffler, Back pressure, Modeling, Computational Fluid Dynamics.

I. INTRODUCTION

Mufflers are important part of engine system and used in every exhaust system to minimize sound transmissions caused by exhaust gases. Exhaust muffler's performance is mainly dependent on value of backpressure. If the exhaust pressure increased then there are several adverse effects such as pumping work increases, intake manifold boost pressure reduces, cylinder scavenging and combustion effects and turbocharger problems etc. arises in engine. Therefore if the backpressure is reduced then engine efficiency will be high. This project is mainly for reducing the backpressure of the muffler. For the existing exhaust muffler the backpressure is more and therefore it creates many problems in engine. Therefore for reducing the backpressure four different types of models are modelled by using CATIA software and analysis of these models done by using the Computational Fluid Dynamics software. From the analysis results comparison of backpressure between existing muffler model and newly modelled muffler will takes place. After comparison the muffler model with less backpressure will be choose as an optimum muffler model.

II. LITERATURE REVIEW

The paper presents a practical approach to design, develop and test muffler particularly reactive muffler for exhaust
system, which will give advantages over the conventional method with shorten product development cycle time and validation. This paper also emphasis on how modern CAE tools could be leveraged for optimising the overall system design balancing conflicting requirements like Noise & Back pressure.[1]. In this paper a reactive perforated muffler is investigated numerically and experimentally. The back pressure affect the engine performance directly an acoustic and flow analysis of the present muffler was examined and compared with experimental results.[2]. The paper presents the study of Different types of muffler and designing methods are studied. After studying this methods and procedures for designing a muffler, he conclude that combination type of muffler is more efficient than reactive and absorptive mufflers. New theory for designing muffler by counter-phase counteracts split- gas rushing and methods of designing Active silencer are also preferable for new research work.[3]. The paper conduct design and free free analysis of muffler system in order to determine the resonant frequencies of the system and suggest changes in the system design. For this analysis, Nastran software was used. In order to determine the resonance frequencies, were then compiled to determine which peaks were the most significant for the system[4].

III. MODELLING

Following are the parameters of the Engine for which this exhaust muffler is used.

- Number of Strokes - 2
- Fuel used - Diesel
- Power - 7.5 HP
- Number of Cylinder - 1
- RPM of Engine - 6000
- Volume of Exhausted Gas - 6400 cc/min.

A. Existing Muffler Model

Below figure shows the existing model of muffler which is modelled in the CATIA V5. The material used for the manufacturing of these mufflers is Mild Steel chromium based material.

The components of muffler are,

- Inlet Pipe
- Outlet Pipe
- Chamber
- Front and Back Baffle

- Perforated Holes

B. New Muffler Models

Following are the newly modelled muffler. These are one chamber, two chamber, three chamber muffler are modelled by using software CATIA V5. Below figure shows Various Muffler models 1,2,3,4.

![Fig.2 Muffler Model 1](image)

![Fig.3 Muffler Model 2](image)

![Fig.4 Muffler Model 3](image)

![Fig.5 Muffler Model 4](image)

Fig. No.3 shows the single chamber muffler model in which the inlet and outlet of the muffler are connected as shown.

Fig. No.4 shows two chambers of the muffler with one baffle and inlet and outlet with perforated holes.
Fig. 5 Muffler Model 4

Fig. No.5 shows the one chamber muffler model with inlet and outlet at both ends which are not connected to each other as shown. These four models are modelled by putting same diameter and length of the muffler, only the internal structure of models has changed.

IV. COMPUTATIONAL FLUID DYNAMICS

For CFD analysis of muffler the muffler was drawn through a 3D CAD program. This model was meshed using Ansys Workbench (Fluent) software. The elements used for meshing are quadrilateral and tetrahedral with other default setting in the Ansys fluent. Firstly in the geometry the flow area of muffler will be filled by using the Fill option in workbench. After the meshing the flow analysis will be done by using fluent by putting various boundary conditions. The inside flow is assumed to be turbulence, so k-ε turbulence model used in this study. The air is used as the fluid material in the analysis of muffler. The density is expected to change with temperature so energy equation keep 'ON' at the time of analysis. At the inlet boundary condition Velocity and Temperature will be defined and at the outlet temperature and pressure will be defined as boundary conditions.

A. Boundary Conditions

At Inlet
- Mass Flow Rate - 6400 cc/min
- Temperature - 400°C
- Operating pressure - Default

At Outlet
- Operating Pressure - 101325 pa
- Gauge Pressure - 0 pa
- Temperature - Default

The fluid material used for the muffler is Air at 400°C
The properties of air at 400°C are

- Specific Heat - 1068 J/ kg K
- Viscosity - 3.2e-5 Kg/ms
- Thermal Conductivity - 0.0515 W/ m k

After doing same iterations of all muffler models and by applying same boundary conditions for each model the value of backpressure will be calculated from the pressure contour and the flow through the muffler will see from velocity streamline. Below are the some meshing models,

B. Meshing of Existing Muffler Model

Fig. 6 Meshing of Existing Model

Fig. 7 Meshing of Muffler Model 1

Fig. 8 Meshing of Muffler Model 2
These above figure no. 7,8,9 shows the meshing of the muffler models which takes place in the ansys workbench. After the meshing the values of backpressure will be calculated from the Ansys Fluent by placing all boundary conditions which are given above.

V. CONCLUSION
From the CFD analysis results it is expected to reduce the value of backpressure of four newly modelled muffler than existing muffler. After analysis these results are compared with experimental results. From all the analysis and experimental results we will choose the muffler with optimum backpressure.

ACKNOWLEDGEMENT
I would like to thank to Sagar Auto Parts pvt. Ltd, Dingrajadi, Pune for their support of this project. I would like to show my deepest gratitude to mechanical department head and my guide Prof. Dr. D.R. Panchagade and PG Coordinator Prof. K.M. Narkar without whose guidance and useful inputs this paper would not have been completed. Lastly, I would like to thank all my professors and friends from the M. E. Design course for their encouragement and support.

REFERENCES


