International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 07 Issue: 05 | May - 2023 SJIF 2023: 8.176 ISSN: 2582-3930

Design Analysis & Modification of Exhaust Muffler of Agriculture Diesel Engine on the Basis of CAE Modeling and Simulation

Mr. Pravin J. Sawant

Assistant Professor , Department of Mechanical Engineering, Sanjay Ghodawat University, Kolhapur e-mail: sawant.pj@sginstitute.in

Abstract

Vibration and noise reduction methods are essential research area in many industries. Mufflers are widely used in automobile industries to reduce noise and vibration transmission. The noise of agriculture diesel engine under the working condition is becomes a serious problem. Generally, noise level of more than 80 dB is harmful for human being. The intensity of noise can be reduced by using muffler at the exhaust system of I.C. Engine. High noise level can be harmful to people and can lead not only to psychological but also leads the hearing defects. Therefore the demand of low noise level becomes vital and more competitive. The noise level can be reduced by modifying the design of existing silencer on the arrangement of baffles. The baffle creates an obstruction to the flow of exhaust gas which will causes the great amount of backpressure at the exhaust port thus losing engine power, increasing fuel consumption and piston effort to exhale the gases out. Therefore it is required to design the silencer on the arrangement of baffle to minimum back pressure of exhaust gas.

This research paper work deals CAE approach with comparison of newly designed model of chambered exhaust silencer and concludes the best possible design for least pressure drop. Field view is used for post processing the CFD results and reviewing the airflow streamlines. The exhaust gas velocity and pressure graphs are studied across the internal tube and holes. The new design of silencer with less noise and minimum backpressure is suggested.

Keywords: Muffler, noise, simulation, Pressure drop, Back pressure.

Introduction

Noise pollution created by engine becomes a vital concern when used in residential areas or areas where noise creates hazard. Generally, noise level of more than 80 dB is harmful for human being. Fortunately, however, this noise can be reduced sufficiently by means of a well designed muffler which is also known as silencer. Good design of the silencer should give the best noise reduction and offer optimum backpressure for the engine [1].

The silencer is a device for reducing the amount of noise emitted by the Engine. It is manufactured as an acoustic soundproofing device designed to reduce the loudness of the sound pressure created by the engine by the way of Acoustic quieting. Due to increased environmental concerns requiring less noise emissions combined with reduced emission of harmful gases, it is becoming very crucial to carefully optimize the design of exhaust system silencers [2].

It is well known that the acoustic performance of silencing elements decreases with increase in exhaust gas flow through it. If the high pressure exhaust gases were allowed to enter atmosphere directly from the exhaust manifold, a loud unpleasant noise will be heard like firing of a gun. This noise is due to the large difference in pressure between the exhaust gases and the atmosphere [3].

For quite operation of the engine, it is desirable to reduce this noise as much as possible. This is done by using a silencer in the exhaust system of the Agriculture Diesel Engine. So, the silencer is connected at the exhaust of the engine. The function of the silencer is to reduce the pressure of the exhaust gases sufficiently to permit them to be discharged to the atmosphere silently. To reduce the pressure, the exhaust gases are permitted to expand slowly in the silencer. The capacity of the silencer should be sufficiently large to permit the gases to expand to nearly atmospheric pressure before they are discharged into the atmosphere [4].

Also, the silencer should not have any appreciable restriction to flow that will raise back pressure excessively, which affects the engine efficiency. Backpressure usually refers to the pressure exerted on a moving fluid by obstructions against its direction of flow [5].

CAE Analysis

1. CAD modeling

In CAD modeling, the interior fluid flowing geometry through the silencer is developed. As shown in Fig.01, the existing model and of silencers is developed in CATIA as per the dimensions. The model under study can be modeled with the help of CATIA.

Also the modified design of silencer is developed in CATIA as shown in Fig. 02. In this new modified design of silencer some necessary changes are made by trial and error method to improve its performance. After developing the geometry in CATIA, it is saved in '.iges' format to import it in CFD simulation software.

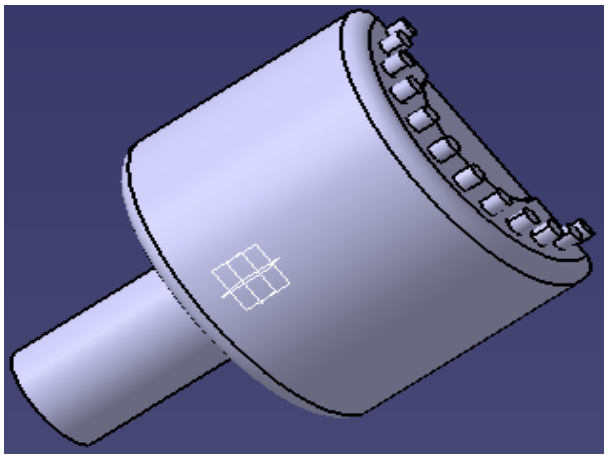

Fig.01. Existing design of silencer.

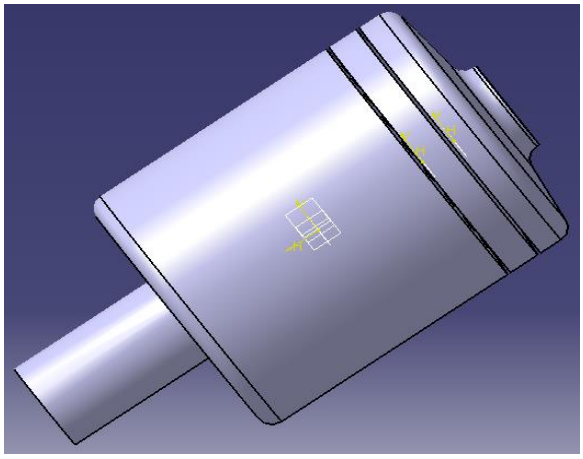

Fig.02. Modified design of silencer.

2. Mesh generation

After developing the geometry of silencer in CATIA, it is imported in ANSYS to carry out the CFD simulation. Meshing of the models is carried out using hyper mesh. Following are the steps carried out on the both silencers cases to get the final meshed models:

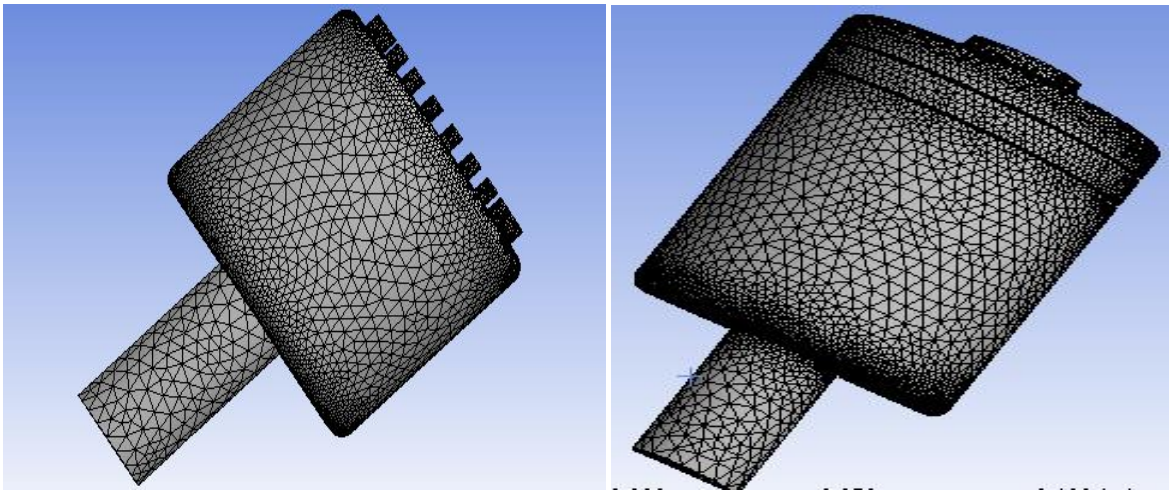

Fig.03. Meshing on existing design of silencer.Fig.04. Meshing on modified design of silencer

3. **Importing and repairing CAD:** In this, the base model modeled using CATIA is imported to hyper mesh. The model file should be imported in '*.iges' format. After importing into hyper mesh, check for the free edges, duplicate surfaces will be carried out. Surface is then be trimmed with proper tolerance so that mesh can be easily put on the surface.
4. **Pre-processing methodology using CFD tool:** Pre-processing for CFD analysis involves the creation of surface mesh, volume mesh and setup of boundary conditions for backpressure study. Meshing is a key step to creating accurate model, correct mesh continuity and mesh density are needed to efficiently compute results and capture the boundary layer effects. The quality of CFD solution is depending on the quality of the underlying volume mesh.
5. **CFD simulation**

CFD is a tool which combines both theoretical and practical approach of problem solving into a single simulation tool. The 3D model of the silencer is imported to the fluent software for further analysis.

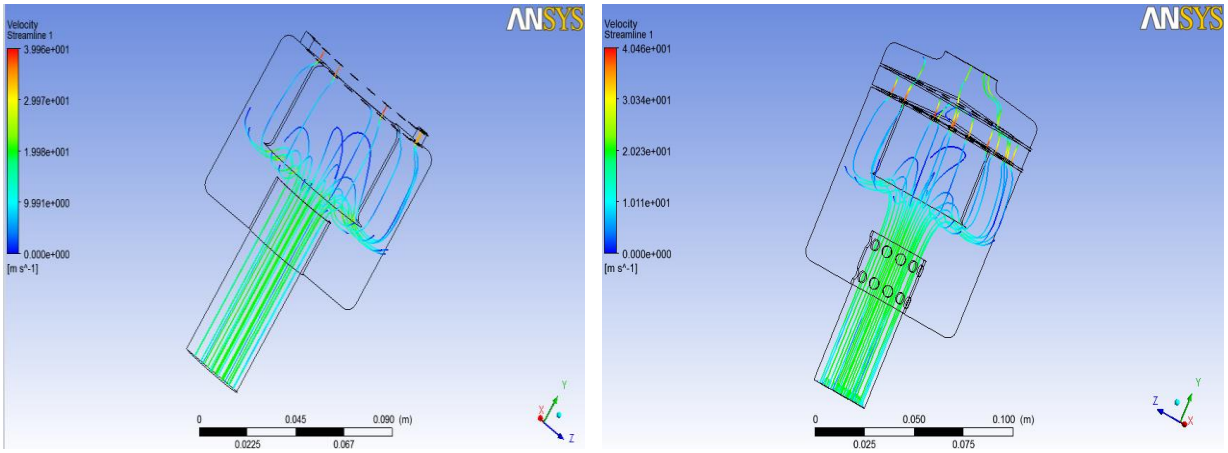

Fig.05. Velocity streamline of existing design Fig.06. Velocity streamline of modified design

The engine runs at various load and speed conditions. The necessary boundary conditions are prescribed by taking in to account the correlative inlet/outlet data at a certain speed for similar type of engine. For the calculation of velocity in CFD simulation, four

different velocities of exhaust gas of agriculture diesel engine are considered. These velocities range from 20 m/s to 80 m/s; the intermediate values are 20 m/s, 40 m/s, 60 m/s and 80 m/s. Specific choices of boundary conditions are as follows: velocity-inlet type, at 20m/s of inlet speed to 80 m/s of inlet speed and pressure outlet type. The walls are considered adiabatic, stationary no slip walls. The initial condition is 1 atmospheric pressure.

The CFD analysis is carried out, for these velocities as inlet to the silencer temperature as 450^0C. Here different velocities are assumption made to get the real velocity of the exhaust manifold also the gas assumed as the exhaust gas properties taken from material library of the ANSYS fluent which is actual condition of the exhaust gas. The outlet pressure is given as atmospheric pressure outside of the silencer. From this FEM analysis the inlet and exhaust gas pressure and velocity profiles are studied from which real exhaust velocity of the engine is calculated.

The CFD simulation is carried out on existing and new modified design of silencer. The simulation results of existing silencer and newly designed silencer are as follows: The CFD simulation is carried out on both silencers. After finishing computation, the convergent results are saved and tabulated as follows

Table No. 01. Simulation results of existing and modified design of silencer

Model No	Existing silencer				Modified silencer			
	Pressure (Pascal)		Velocity (m/s)		Pressure (Pascal)		Velocity (m/s)	
	Inlet	Outlet	Inlet	Outlet	Inlet	Outlet	Inlet	Outlet
1	101839.09	101325	20	30.54	102522.29	101325	20	37.29
2	103350.15	101325	40	61.07	106058.02	101325	40	74.551476
3	105861.98	101325	60	91.59	111757.08	101325	60	111.801
4	109408.8	101325	80	122.15	119843.15	101325	80	149.01479
Mean average	105114.98	101325	50	76.33	110045.13	101325	50	93.16

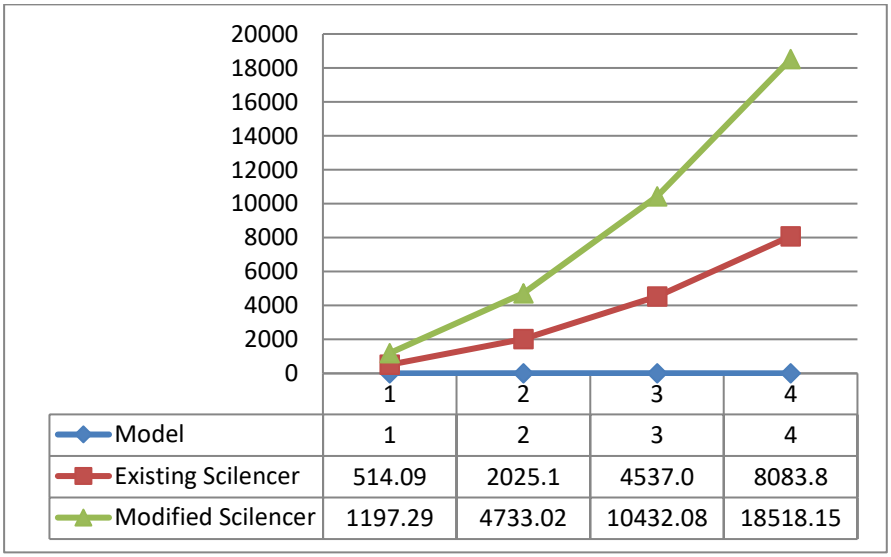

Fig.07. Pressure difference of Existing & Modified silencer

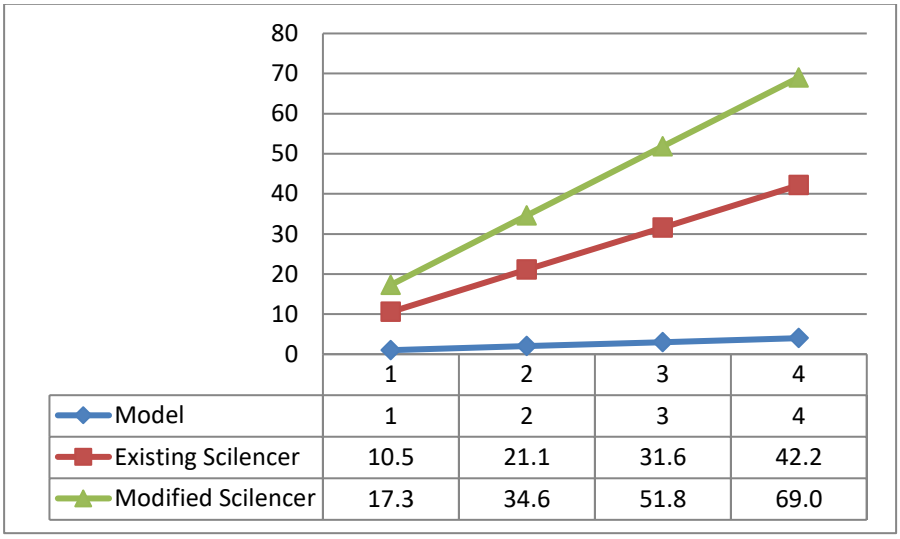

Fig.07. Velocity difference of Existing & Modified silencer

Conclusion

In this work, two different models of silencer have been designed for the engine output of an agriculture diesel engine and the flow has been simulated using ANSYS FLUENT. The flow characteristics obtained through the simulation were promising. The analytical results of existing silencer are compared with newly modified silencers and following conclusions are drawn.

1. On comparing the results and performance of modified silencer with the existing silencer it is observed that, the first modified design of silencer is effective to reduce the pressure drop because of its internal baffle arrangement.
2. Increasing the number of baffle plates, reduce the noise level.
3. The CFD simulation software is a tool to estimate the performance of noise reduction of silencer through modifying its shape. The simulations give valuable information regarding the velocity field and pressure field of the exhaust silencer. The CFD simulation is carried out on both silencers. The velocity is given as inlet boundary condition whereas pressure is at outlet.
4. As compared to existing silencer the velocity of exhaust gas is increased and pressure is reduced at the exhaust of both modified designs of silencer.

References

1. Min-Chie Chiu, Ying-Chun Chang, “Numerical studies on venting system with multi-chamberPerforated mufflers by GA optimization”, Applied Acoustics 69 (2008) 1017–1037.
2. MurariMohon Roy, Mohammad Uzzal Hossain Joardder, Md. Shazib Uddin, “Effect of Engine Backpressure on the Performance and Emissions of a CI Engine”, Jordanian International Mechanical Engineering Conference, 2010.
3. R.Ramganesh, G.Devaradjane, “SIMULATION OF FLOW AND PREDICTION OF BACK PRESSURE OF THE SILENCER USING CFD”, Journal of Chemical and PharmaceuticalSciences, 2015, 7: 2015, 297-300.

4. Mr. S.S. Mane, S.Y.Bhosale, "Back pressure analysis of an engine muffler using cfd and experimental validation", International Engineering Research Journal, 2016, ISSN 2395-1621, Special Issue Page 828-834.
5. Puneetha C G, Manjunath H, Shashidhar M.R, "Backpressure Study in Exhaust Muffler of Single Cylinder DieselEngine using CFD Analysis", Altair Technology conference, 2015, 1-14.