Journal of Chemical and Pharmaceutical Sciences www.jchps.com ISSN: 0974-2115

SIMULATION OF FLOW AND PREDICTION OF BACK PRESSURE OF THE SILENCER USING CFD

***R.Ramganesh, G.Devaradjane**
Department of Automobile Engineering, Anna University –MIT Campus India
***Corresponding author: Email: ramganesh009@gmail.com**

ABSTRACT

The performance of any automotive engine depends not only upon its core engine parts but also on the effectiveness of the sub-systems attached to the engine, like the intake and exhaust systems. The exhaust system being a critical system of any automotive vehicle plays a responsible role of improving the ride quality of the vehicle by attenuating the noise from the engine without deteriorating the engine performance by ensuring an optimum value of back pressure. The crucial component of the exhaust system which is mainly responsible for this function is called as the silencer. This work is mainly concerned with the simulation of flow through the silencer followed by the prediction of the exhaust back pressure of the silencer using computational fluid dynamics (CFD). Exhaust back pressure is one of the crucial parameters that is always scrutinized by the automotive manufactures to ensure that their engine delivers a superior performance. Despite contradicting relationship between the noise attenuating capability and back pressure of the silencer, an overall low back pressure for the whole exhaust system is achieved with a compensating design of the exhaust system, hence the silencer. The technological development of the high speed computing machines has led to widespread use of simulation tools to predict the behavior of the flow in an exhaust silencer, thus helping in virtual simulation of the system as a first step, which can further be advanced to the optimization stage thus reducing the overall product development cycle time. This project which is about the study of the silencer utilizes the Hypermesh tool as a pre-processor for element generation followed by the simulation of flow and subsequent analysis using computational fluid dynamics, Fluent software.

INTRODUCTION

The stringent environmental laws demand automotive systems to be produced with superior performance with reduced noise, emissions, maintaining good fuel economy at the same time. The performance of any vehicle highly depends upon the performance of the engine. Engine being the power generating source continuously generates pressure waves during its working process. These pressure waves are nothing but the pulsating behavior of the exhaust gases. These exhaust gas pulses move due to a differential pressure, from the engine to the ambient atmosphere. These pulses when they interact with the atmosphere which is at a low pressure than the combustion gases, it imparts energy to the acoustic field creating self-sustained oscillations with some average amplitude that finally gets exhibited as noise. Silencer also called as muffler is one of the crucial component of the exhaust system responsible for reducing the noise of the exhaust gases or attenuating the sound waves generated by the automotive engines. Silencers reduce these sound wave amplitude by a concept called as destructive interference, wherein a wave of equal amplitude and opposite in phase is introduced against the intended wave, thereby these waves cancel each other, which is the principle behind silencers for sound reduction. Silencers achieve this by creating turbulence in an expansion chamber with obstructions. On the other hand these obstructions create a hindrance for the flow of exhaust gases from the engine to the atmosphere due to which a static pressure is imposed on the engine which is commonly called as exhaust back pressure. This exhaust noise and exhaust back pressure always have an inverse contradicting relationship between them. Thus the exhaust back pressure can be defined as the resistance pressure imposed on the engine due to the resistance of the exhaust system in facilitating the flow of the exhaust gases from the engine to the outside ambient. More clearly back pressure can be defined as the resistance to positive flow of the exhaust stream. Increased back pressure causes the engine to run hotter, reduces the brake power of the engine, increases the pumping work of the engine and above all Increases the fuel consumption. Hence the silencer of an exhaust system should have a low back pressure for optimum engine performance.

Modelling and Element Generation

Study and Modelling: The schematic of the silencer described below is a simple reflective type of silencer consisting of an inlet tail pipe and outlet tail pipe which are connected together by means of the expansion chamber. The expansion chamber is again considered to be partitioned into two zones or chambers by means of a central plate. The left part of the chamber has a connection from the inlet tail pipe for a suitable distance before the central plate while the right side chamber has a pipe connection from the central part of the plate to the outlet tail pipe. The inlet and the outlet tail pipe extending into their respective chambers have suitable porosity as shown in the model. The final 3D model of the silencer was created using creo 2.0 as shown below in fig. 2.

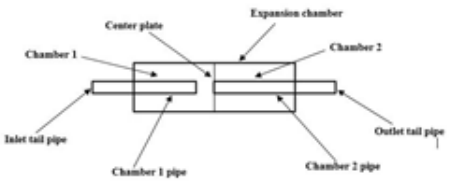

Fig. 1. Silencer schematic

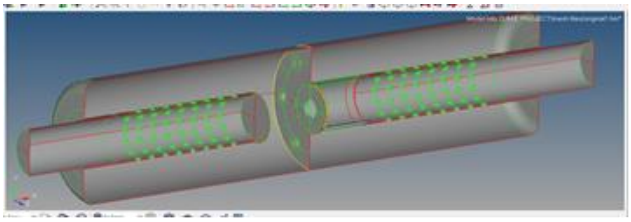

Fig.2 Silencer section model (3D)

Element Generation

After the study of the constructional features of the silencer in the above section, 3D silencer model is then converted into suitable finite element model using Hypermesh tool as the pre-processor. For the 2D element generation three noded trias element has been preferred instead of four noded quadratic element on account of it's less computational time, appropriate accuracy and good surface coverage. To study the effect of turbulence more accurately hybrid mesh concept has been used for element generation wherein, smaller element size is used near the holes provided in the center plate and in the chamber pipes, so as to capture the exact geometry of the shape which is nothing but creating a fine mesh in these areas. For the areas which are devoided of porosity (ie) the outer wall of the expansion chamber, the inlet and the outlet tail pipe- a larger element size is used so as to make the mesh relatively coarse ensuring it still captures the surface geometry effectively. This technique ensures that the solving time gets reduced in the solver (Fluent) and at the same time the required flow is studied more accurately in the intended areas. Finally the 2D element model is converted to 3D element model after the calculating the first layer thickness (given in next section) and specifying it in the CFD tetramesh panel of the hypermesh. The summary of the element size and element model of the silencer is shown in below table and figures respectively.

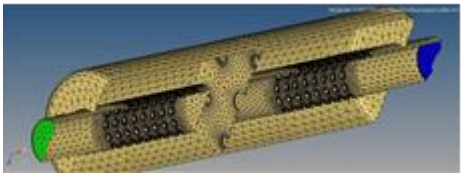

Fig. 3. FE section model (2D)

Fig. 4. FE section model (3D)

Parameters	Values
Element size (near holes)	3mm
Element size (near wall)	10mm

CFD Analysis

Computational fluid dynamics provides a new approach in the philosophical study and development of the whole discipline of fluid dynamics. The advent of high speed digital computer combined with the development of accurate numerical algorithm for solving problems on these computers have revolutionized the way we solve fluid dynamics problems today, which is what computational fluid dynamics is about. In simple words CFD is a tool which combines both theoretical 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. However certain assumptions and few pre-calculations as given below have been performed to determine the suitable input conditions for solving the problem and predicting the back pressure. The assumptions are

- The flow is assumed to be steady and non-uniform.
- The flow is assumed to be incompressible (Due to low Mach number).
- The exhaust gas is assumed to have the properties of air.
- The pre-calculations summarized below are carried out using properties of air at temperature of the exhaust.

- The exhaust mass flow rate is given by, m_{ex} = (engine capacity in lts * $N*\eta_{vol}*\rho)*(T_{ex}(^0F) + 460)/(540*60*2000)$ kg/s
- The velocity of the flow at the inlet is, $V_f = m_{ex} / (\rho*A)$
- The Reynolds number for the flow is, $R_e = (\rho* V_f*D) / \mu$
- The flow Mach number, $M = V_f / c$; where $c = (\gamma*R* T_{ex}[^0K])^{1/2}$
- Wall shear stress, $\tau_w = (0.5*C_f* \rho* V_f^2)$; where $C_f = 0.0791*R_e^{-0.25}$
- The friction velocity, $u_\tau = (\tau_w / \rho)^{1/2}$
- The first layer thickness for $y^+ = 30$[1], is $y = (y^+ * \mu) / (\rho* u_\tau)$

Based on the above calculations, the required input conditions generated for flow analysis have been summarized in the table given below.

Solver type	density solver (Incompressible)
Turbulent model	Realizable k-e turbulent model
Wall function	Standard wall function
Fluid	Air
Fluid properties	Properties of air at exhaust temperature
Inlet condition	Mass flow inlet(value from calculation) [5]
Outlet	Pressure outlet
Wall	Stationary wall

RESULTS AND DISCUSSION

The results obtained from the streamline flow clearly indicates that as the chamber 1 pipe is closed at its end, the only way for escaping of the gases into the expansion chamber 1 is through the pores provided on the chamber 1 pipe. Further these gases from the expansion chamber 1 move to chamber 2 through the pores provided on the center plate.

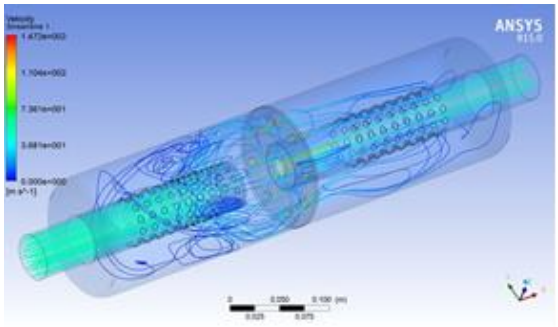

Fig. 6. Silencer pressure contour

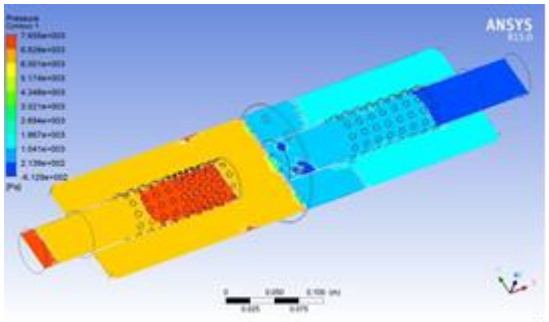

Fig. 7. Silencer pressure drop

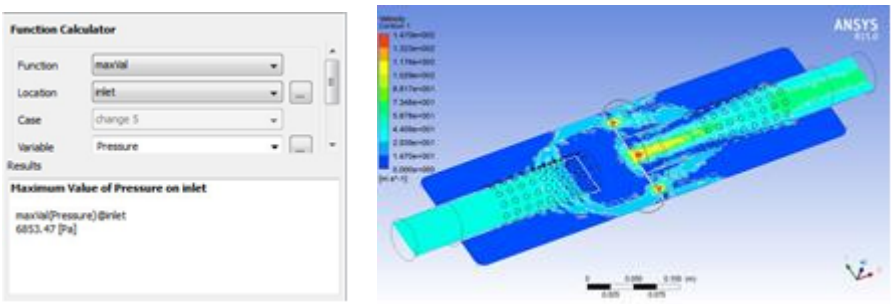

Fig.8. Silencer velocity contour

The clarity from the pressure contour plot of the silencer conspicuously shows that the region of pressure build is the chamber 1 and more specifically it is the chamber 1 pipe and its associated porosity. Furthermore from the velocity contour it is clear that the peak flow velocity occurs at the center of every pores in the center plate with

the center hole on the plate dominating the flow (ie) the majority of the particles escape from chamber 1 to chamber 2 through the central hole of the center plate. Finally for this silencer the pressure drop calculated from the analyses was found to be 68 mbar.

CONCLUSION

Finally the simulation of flow through the silencer and prediction of the back pressure for the same has been successfully carried out using computational fluid dynamics. The above investigation and analysis has put forth a collective idea that the back pressure of any silencer mainly depends on the porosity of the pipes in the expansion chamber. However they are not the only factor determining the value of back pressure because the porosity of the central plate also plays an equal dominating role. This work can further be extended in the future in the direction of optimization of the silencer for reduced back pressure and analyzing the same for effectiveness of the sound attenuation quality.

REFERENCES

Jiawei Liu, Simulation of whistle noise using computational fluid dynamics and acoustic finite element simulation-Theses and Dissertations-nMechanical engineering paper 9.

Rakesh Verma et al. FEA Approach for Calculating Back pressure in Automotive Muffler. SAE Technical Paper 2013-26-0046.

Textbook- “Fluid Mechanics and Hydraulic Machines – R.K. Bansal”

Ansys-workbench Fluent help

Shital Shah et al. A Practical Approach Towards Muffler Design, Development and Prototype Validation. SAE Technical Paper 2010-032-0021.