PERFORMANCE EVALUATION OF A REACTIVE MUFFLER USING CFD

Sileshi Kore, Abudlkadir Aman and Eddesa Direbsa*
Department of Mechanical Engineering
Addis Ababa Institute of Technology, Addis Ababa University

ABSTRACT

The main objective of this study was to simulate and investigate the performance of a reactive muffler by simulation technique. The dimensions of Alfa Romeo 145 vehicle model 1999 muffler were used to perform the simulation. The simulation was carried out using a commercial software package named FLUENT. In this package, a program GAMBIT was used to create a mesh surface and to define the boundary conditions of the required object, which was read and analyzed by FLUENT. The result shows that CFD can be used to evaluate both the mean flow and acoustic performance of a reactive muffler.

Keywords: *Reactive muffler, Aero-acoustics, CFD, Sound attenuation, FLUENT, Exhaust noise reduction*

INTRODUCTION

A pollutant of concern to mankind is the exhaust noise in the internal combustion engine. However, this noise can be reduced sufficiently by means of a well designed muffler. The suitable design and development will help to reduce the noise level, but at the same time the performance of the engine should not be hampered by the back pressure caused by the muffler [1, 2, and 3].

A **muffler** (or **silencer** or **back box** in British English) is a device fitted to internal combustion engines for reducing the amount of noise. On internal combustion engines, the engine exhaust blows out through the muffler. The internal combustion engine muffler or silencer was originally invented by Milton O. Reeves. Fig. 1 shows the schematic view of gas flow in a piston engine. The specific items are given from 1 to 13.

1. ram air,
2. air filter
3. mass flow sensor,
4. butterfly valve,
5. air box,
6. intake runners
7. intake valve
8. piston
9. exhaust valve
10. extractor pipe
11. collector
12. catalytic converter muffler (expansion chamber)
13. Muffler (expansion chamber)

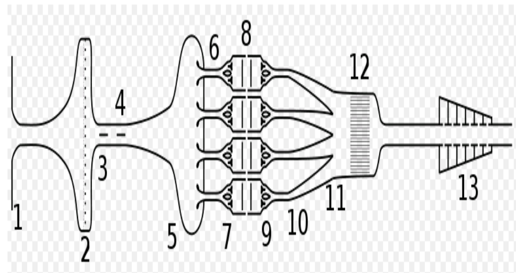

Figure 1 Schematic view of gas flow in a piston engine

In general, sound waves propagating along a pipe can be attenuated using either a dissipative or a reactive muffler [4]. A dissipative muffler uses sound absorbing material to take energy out of the acoustic motion in the wave, as it propagates through the muffler. Reactive silencers, which are commonly used in automotive applications, reflect the sound waves back towards the source and prevent sound from being transmitted along the pipe. Reactive silencer design is based either on the principle of a Helmholtz resonator or an expansion chamber, and requires the use of acoustic transmission line theory

This study focuses on an expansion chamber. An expansion chamber muffler consists of sudden change in cross sectional area that serves to reflect sound wave back to the engine.

The main objective of this work is to simulate and evaluate performance of the muffler of the Alfa Romeo 145 vehicle using FLUENT. The dimensions of the muffler were used to perform the simulation.

AERO-ACOUSTICS

With the ongoing advances in computational resources and algorithms, CFD is being used more and more to study acoustic phenomena. Through detailed simulations of fluid flow, CFD has become a viable means of gaining insight into noise sources and basic sound production mechanisms. FLUENT offers four approaches for simulating aero

*E-mail: Edessa_dribssa@yahoo.com

acoustics [5], in order of decreasing computational effort;

1. Computational aero-acoustics (CAA),
2. The coupling of CFD and a wave-equation-solver,
3. Integral acoustic models, and
4. Broadband noise source models.

Computational Aero-Acoustics

Computational aero-acoustics is the most comprehensive way to simulate aero-acoustics [6]. It does not rely on any model, so is analogous to direct numerical simulation (DNS) for turbulent flow. CAA is a transient simulation of the entire fluid region encompassing the sources, receivers and entire sound transmission path in between [5]. By rigorously calculating time-varying flow structures, pressure disturbances in the source regions can be followed. Sound transmission is simulated by resolving the pressure waves traveling through the fluid. While CAA is the most general and accurate theoretical approach for simulating aero acoustics, it is unrealistic for most engineering problems because of a number of practical limitations, including:

- widely varying length and time scales characteristic of the sound generation and transmission phenomena, and
- widely varying flow and acoustic pressures

While these constraints render CAA unsuitable for most practical situations, there is a small class of engineering problems to which it can be successfully applied. This includes cases where:

- the frequency range of interest is fairly narrow
- the sources and receivers are located close to each other, and
- the sound to be captured is fairly loud

CFD Wave Equation Solver Coupling

The computational aero-acoustics approach is prohibitively expensive for most practical problems due to the large difference in time, length, and pressure scales involved in sound generation and transmission. Computational expense can be greatly reduced by splitting the problem into two parts:

1. Sound generation
2. Sound transmission.

With this approach, sound generation is modeled by a comprehensive transient CFD analysis, while a wave equation solver is used for analyzing sound transmission

Integral Acoustics Methods

The approach of splitting the flow and sound fields from each other and solving for them separately can be simplified further if the receiver has a straight, unobstructed view of each individual point that is a source of noise. Sound transmission from a point source to a receiver can be computed by a simple analytical formulation. The Lighthill acoustic analogy provides the mathematical foundation for such an integral approach [5]. The Ffowcs-Williams and Hawkings (FW-H) method extends the analogy to cases where solid, permeable, or rotating surfaces are sound sources, and is the most complete formulation of the acoustic analogy to date. Both methods are implemented in FLUENT. As an example, the FW-H method has been applied to the prediction of sound radiating from a backward facing elbow (a simplified representation of an automotive A-pillar rain gutter). Using the large eddy simulation (LES) turbulence model, predictions of the sound pressure level for this case were found to be in very good agreement with experimental data taken from the literature.

Broadband Noise Source Models

The three methods described so far require well-resolved transient CFD simulations, since they aim to determine the actual time-varying sound-pressure signal at the receiver, and from that, the sound spectrum. In several practical engineering situations, only the locations and relative strengths of sound sources, rather than the sound spectra at the receivers, need to be determined. If the sound is broadband the source strengths can be evaluated with reasonable accuracy from the time-averaged structure of the turbulent flow in the source regions. Turbulence is the primary cause of sound in aero-acoustics, so in a broad sense, regions of the flow field where turbulence is strong produce louder sources of sound. FLUENT 6.2 includes a number of analytical models referred to as broadband noise source models which synthesize sound at points in the flow field from local flow and turbulence quantities to estimate local sound source strengths. The key advantage of these models is that they require very modest computational resources compared to the methods described in the previous sections. Broadband noise models only need a steady state flow solution, whereas the other methods require well-resolved transient flow solutions. One example recently studied involves the prediction of prominent sound sources around a simplified sedan, using Lilley's acoustic source strength broadband noise model.

In summary, FLUENT offers four ways for simulating aero-acoustics. These range from highly accurate, but expensive methods to quick and approximate approaches. In this work the Ffowcs-Williams and Hawkings (FW-H) method which is less expensive and transient [5] was used.

Empirical Relation

Theoretically, the transmission loss will increase with the increase of the ratio of cross-sectional area of expansion chamber to both inlet and outlet pipe cross-sectional areas.

Transmission Loss of a Muffler

The transmission loss of a muffle is given by the following equation.

$$TL = 10 \log_{10}\left[1 + \frac{1}{4}\left(m - \frac{1}{m}\right)^2 \sin^2 kl\right] \qquad (1)$$

$$k = \frac{2\pi fl}{c}$$

where k *is sound wave number*
l *the length*
f *is the frequancy*
c *is speed of light*

Governing Equation

The computational equation for the muffler is given by the following differential equation.

$$\frac{\partial^2 \phi}{\partial t^2} = u^2 \frac{\partial^2 \phi}{\partial x^2} - \qquad ------- \quad (2)$$

Where ϕ is the pressure

Substituting p in Eq. 2:

$$\frac{\partial^2 p}{\partial t^2} = u^2 \frac{\partial^2 p}{\partial x^2} \qquad ------------(3)$$

BASIC TERMINOLOGIES

Possibly, the simplest form of reactive muffler is the so called expansion chamber muffler shown in Fig. 2.

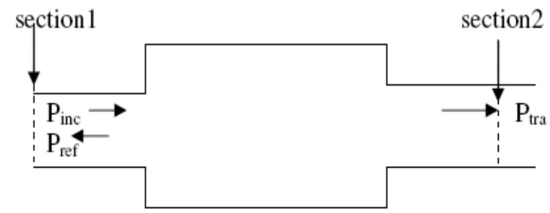

Figure 2 Simple expansion chamber muffler

The acoustic plane wave of amplitude P_{inc} and angular velocity $\omega = 2\pi f$ is propagating in the inlet pipe towards the muffler expansion. The expansion from the inlet pipe result in a reflected plane wave of amplitude P_{ref} propagated away from the muffler as shown in Fig 2. Plane wave will propagate in the expansion section and it is the destructive interference of the wave which makes the muffler effective. A plane wave of amplitude P_{tra} is transmitted along the outlet pipe from the muffler. The outlet pipe is assumed to be infinitely long or anechoic-ally terminated so that there is only one wave, the transmitted wave, in the out let pipe. The ratio of the acoustic power associated with the incident wave W_{inc} to that associated with the transmitted wave W_{tra} can be used to determine the frequency dependent Transmission Loss

TL is given by the following equation.

$$TL = 10 \log_{10} \frac{W_{inc}}{W_{trs}} \text{ -------------------- } (4)$$

If the initial and the outlet pipe of the muffler have equal area, the ratio of the powers as given in Eq. 4 is the same as the ratio of the acoustic intensities. The intensity of a propagating harmonic plane acoustic wave having amplitude of P is $\frac{P^2}{2\rho c^2}$, where ρ is the density of the gas and c is the velocity of sound in the gas. Thus if the values of the ρ and c at the inlet and outlet pipes of the muffler are identical, as are the pipe areas, the formula for the transmission loss given by Eq. 4 becomes:

$$TL = 10 \log_{10} \frac{P_{inc}}{P_{trs}} \text{ ---------------------------- (5)}$$

GAMBIT

GAMBIT is CAD software and is compatible with FLUENT software for running simulations based on fluid flow and heat transfer analysis. GAMBIT allows the user to create a mesh surface and define the boundary conditions of the required constructed object, which will be read and analyzed by FLUENT.

The geometries used in the simulation are as follows according to Fig. 3

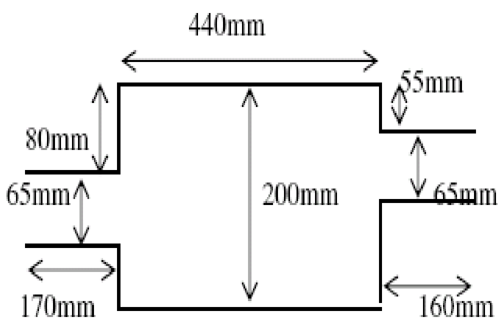

Figure 3 Muffler dimension for simulation

- Inlet pipe length: 0.17 m
- Outlet pipe length: 0.16 m
- Expansion chamber length: 0.44 m
- Inlet pipe diameter: 0.065 m
- Outlet pipe diameter: 0.065 m
- Expansion chamber diameter: 0.2 m

SOLVER AND BOUNDARY CONDITIONS

A commercial CFD package, Fluent 6.3 was used. The solver implemented was a 2-D, segregated implicit solver with 2nd order implicit time stepping [5]. Second order upwind discretization was used for the density, momentum, energy, turbulent kinetic energy and the turbulent dissipation rate equations. PISO pressure velocity coupling is used. The $k-\varepsilon$ turbulence model was used for closure [5].

The working fluid was air with the density modeled assuming an ideal gas and the properties shown in Table1. The boundary conditions consist of a velocity inlet, a pressure outlet and a series of walls.

Table1: Solution variables

Variables	Value
Working fluid	Air
Mean pressure	101.325kPa
Mean temperature	300K
Dynamic viscosity kg/m.s	1.79×10^{-5}

The edges are meshed with spacing of 2 mm in order to have better nodes spacing and gives better results in FLUENT analysis. The inlet is defined as the velocity-inlet boundary and the outlet is defined as the pressure-outlet boundary. The file is saved and exported to FLUENT for analysis.

In this case the inlet velocity is normally 80 m/s. The outlet boundary condition has been set at atmospheric pressure.

Velocity and pressure data are recorded at a point in the inlet pipe and at the pointing the outlet pipe at each time step. The model is marched forward in time step of 60 μ s.

Fourier transforms of the two pressure time histories is taken to evaluate the sound pressure level at receiver-1 and receiver-2.

MATLAB is a high-performance language for technical computing [7]. MATLAB is used to determine the transmission loss of the simplified model muffler of Alfa Romeo.

RESULTS

In Figure 4, the static pressure distribution is indicated by various colored contours and in Fig. 5 the static pressure distribution along the muffler is shown at 4000 rpm. The pressure rise in the expansion chamber is due to stack up of gas flow as the cross-sectional area between the chamber and the outlet pipe is changed. However, at the outlet, the velocity of exhaust gas is increased due to the high pressure in the chamber and pushes the exhaust gas out to the atmosphere.

Figures 6 and 7, show the variation of velocity of exhaust gas flow from the inlet pipe to the outlet pipe. Both figures indicate the inverse relationship between pressure and velocity which obeys the Bernoulli principle.

Figures 8 and 9, show the sound pressure level with frequency for expansion chamber at 4000 rpm at receiver-1 and receiver-2 i.e. at the inlet and outlet respectively.

Theoretically, the transmission loss will increase with the increase of the ratio of cross-sectional area of expansion chamber to both inlet and outlet pipe cross-sectional area. Since the dimensions of the simplified model of Alfa Romeo's muffler are constant, the maximum transmission loss of the muffler is almost constant. Therefore using Eq. 4 the transmission loss is calculated and the result is shown in Fig. 10.

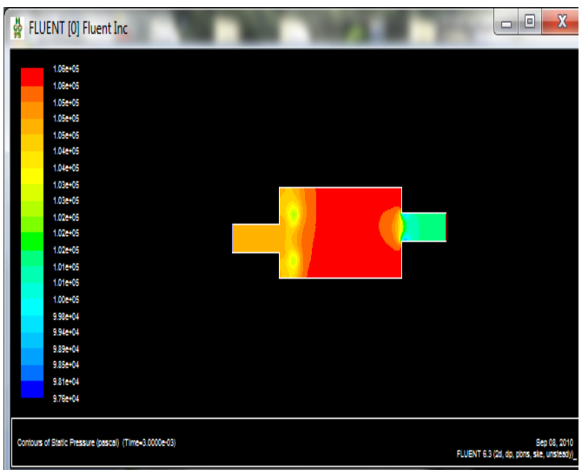

Figure 4 Contours of static pressure

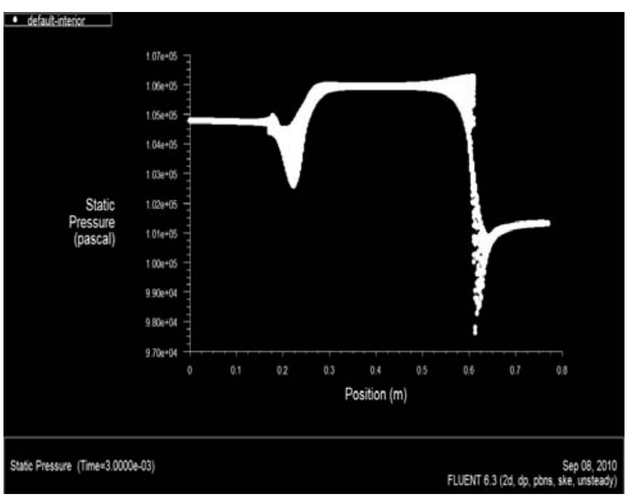

Figure5 Static pressure versus position in muffler

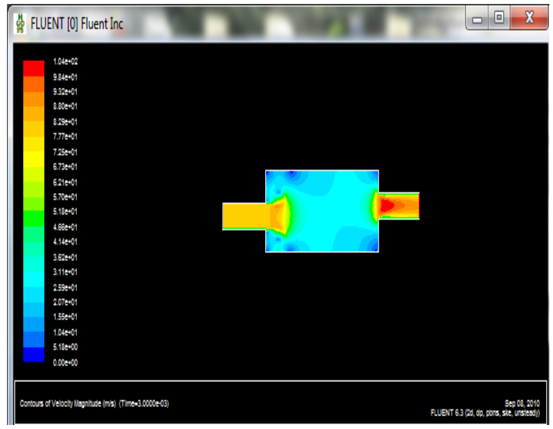

Figure 6 Contours of velocity

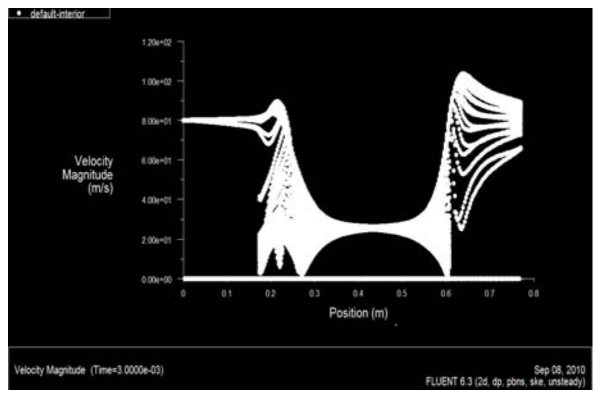

Figure 7 Velocity versus position in muffler

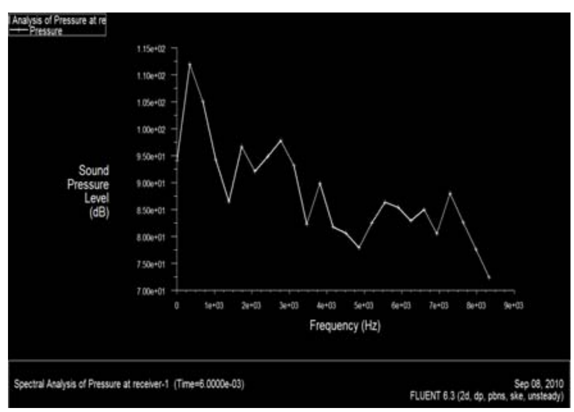

Figure 8 Sound pressure level in dB at the inlet

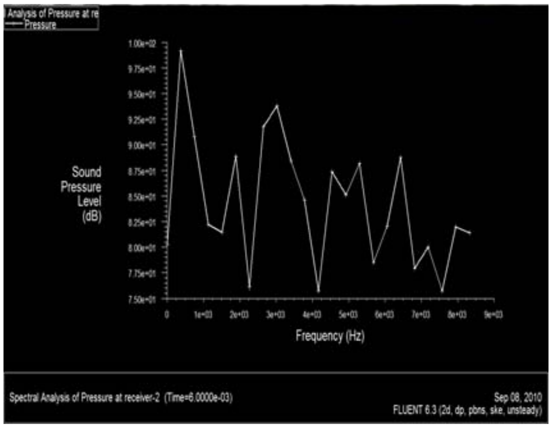

Figure 9 Sound pressure level in dB at the outlet

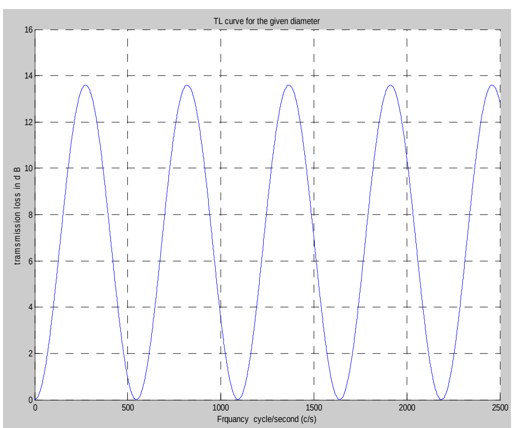

Figure 10 Transmission losses for plane wave equation 1

DISCUSSION

The flow of fluid in ducts and pipes will experience change in pressure and velocity. The fluid pressure decreases if the fluid speed increases (and vice versa). The air at inlet and outlet pipe has a greater velocity compared to the air flow in the expansion chamber which can be seen in Figs. 6 and 7 of simulation. The color contour indicates the velocity of air flowing in and out of the expansion chamber and the velocity of the air in the expansion chamber as well.

The overall simulation result on the air flow in the muffler indicates that the air flow in the expansion chamber creates stagnation pressure that contributes to the back pressure for the muffler performance. The stagnation pressure is the extra static pressure that increases the pressure of air flow inside the muffler, which eventually will affect the engine performance.

Back pressure represents the extra static pressure exerted by the muffler on the engine through restriction in the flow of exhaust gases which for a four-stroke-cycle engine would affect the brake power, volumetric efficiency, and hence the specific fuel consumption rate.

The first five points from Figs. 8 and 9 are considered and tabulated in Table2.The result shows the oscillation of the transmission loss is about 14 dB.

The maximum transmission loss from Eq. 4 is 13.6, as shown in Fig. 10. The deviation from the simulation is very small. Therefore the empirical equation is good to evaluate the transmission loss of such type of muffler, i.e. simple reactive muffler, Alfa Romeo 145 vehicle.

Table2: Sound pressure level and transmission loss

Inlet sound pressure level (SPL_{in}) dB	Outlet sound pressure level (SPL_{out}) dB	Transmission loss TL (SPL_{in} –SPL_{out})
94	80.25	13.75
112.5	99.25	13.25
105	91	14
94	82	12
86.25	81.5	4.75

CONCLUSION

This work indicates that CFD can be successfully used to evaluate both the mean flow and acoustic performance of a reactive muffler. The muffler design requirement is based on the adequate insertion loss, back pressure, size, durability, sound quality, breakout noise from muffler shell and the flow generated noise. The maximum transmission loss is around 14dB for the developed model. Eq. 1 plots using MATLAB supports this result as indicated in Fig. 10.

REFERENCES

[1] Middelberg, J., "*Determining the Acoustic Performance of a Simple Reactive Muffler Using Computational Fluid Dynamics*", Proc. 8th Western Pacific Acoustics Conference, University of New South Wales, Sydney, Australia, 2003.

[2] Selamet,A.,"*Acoustic attenuation performance of circular expansion chambers with extended inlet/outlet*", Journal of Sound and Vibration, vol. 223 no. 2, 1999, pp.197-212.

[3] Munjial, M.L., "*Acoustic of Ducts and Muffler*", Wiley-Interscience, Inc., 1987.

[4] Wilson, C.E., "*Noise Control: Measurement, Analysis, and Control of Sound and Vibration*", Krieger Publishing Company, 1993.

[5] Fluent Inc, "*Fluent 6.3 Users Guide*", 2003.

[6] Mohiuddin, A.K.M., Rahman, A. and Gazali, Y.B.,"*Experimental Investigation and Simulation of Muffler Performance*", Proceedings of the International Conference on Mechanical Engineering, 2007.

[7] http://www.corneluniversity.edu.us. Visited on September7th,2006,from "*FLUENT Tutorials - Cornell University*",