COMPUTATIONAL FLUID DYNAMICS ANALYSIS OF THE ACOUSTIC PERFORMANCE OF VARIOUS SIMPLE EXPANSION CHAMBER MUFFLERS

Middelberg, J.M., Barber, T.J., Leong, S. S., Byrne, K.P and Leonardi, E.

School of Mechanical and Manufacturing Engineering, The University of New South Wales, Sydney 2052, Australia

Abstract

Different configurations of simple expansion chamber mufflers, including extended inlet/outlet pipes and baffles, have been modelled numerically using Computational Fluid Dynamics (CFD) in order to determine their acoustic response. The CFD results are compared with published experimental results. The CFD model consists of an axisymetric grid with a single period sinusoid of suitable amplitude and duration imposed at the inlet boundary. The time history of the acoustic pressure and particle velocity is recorded at two points, one point in the inlet pipe and one point in the outlet pipe. These time histories are Fourier Transformed and the transmission loss of the muffler is calculated. Calculated results show excellent agreement with the published data. The mean flow performance has also been considered. The mean flow model of the muffler uses the same geometry, but has a finer mesh and has a suitable inlet velocity applied at the inlet boundary and the pressure drop across the muffler is found.

Nomenclature

l_e	Length of expansion section
d_i	Inlet pipe diameter
d_o	Outlet pipe diameter
d_e	Expansion section diameter
ρ	Density
c	Speed of sound
ω	Radian frequency
P	Acoustic pressure
W	Acoustic power

Introduction

Virtually all reciprocating internal combustion engines are fitted with mufflers. The muffler fitted to an engine is intended to reduce the pressure pulses associated with the exhaust gas leaving the cylinders of the engine. Generally mufflers fitted to such engines are essentially reactive devices as opposed to being dissipative devices. Reactive mufflers operate by the destructive interference of the acoustic waves propagating within them. Dissipative mufflers operate by the dissipation of acoustic energy, usually within porous fibrous materials. Practical reactive mufflers also have some dissipative function.

An ideal muffler for a reciprocating internal combustion engine should function as a low pass filter. The steady or mean flow should be allowed to pass unimpeded through the muffler while the fluctuating flow which is associated with the acoustic pressure fluctuation is impeded. If the steady flow is not significantly impeded the so-called 'back pressure' will be very low and the engine will function more efficiently.

It is desirable to be able to predict the pressure drop associated with the steady flow through the muffler. Computational Fluid Dynamics (CFD) has developed over the last two decades so that this prediction can be made reliably. It is also desirable to be able to predict the acoustic performance of the muffler. Essentially this means determining as a function of frequency how harmonically varying pressure fluctuations at the inlet of the muffler are largely attenuated before they emerge at the outlet. The present paper is concerned with describing how CFD can be used for this purpose.

This work, like related work of Selamet and Ji [1] is concerned with the important plane wave case. They considered the effects of expansion length on the muffler transmission loss using: (a) two-dimensional axisymmetric analytical solution; (b) three-dimensional computational solution using a boundary element method; (c) experimental results. Selamet and Radivich [2] have published results showing the effects of an extended inlet/outlets on the transmission loss using the same three methods. This paper uses the experimental results from these authors as a basis for comparison for the numerical results presented in this paper.

Basic Terminology

The simplest type of reactive muffler is the simple expansion chamber muffler, shown in Figure 1. The acoustic performance of this device can be found analytically.

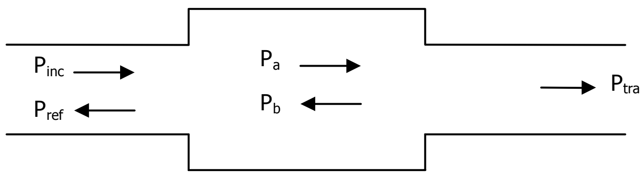

Figure 1. Simple Expansion Muffler

Consider a harmonic plane wave of amplitude P_{inc} and angular frequency ω, which is propagating in the inlet pipe towards the muffler expansion. The expansion from the inlet pipe results in a reflected plane wave of amplitude P_{ref} propagating away from the muffler and a transmitted wave P_a as shown in Fig. 1. Plane waves will propagate in the expansion section and it is the destructive interference of these waves that make 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 anechoically terminated so that there is only one wave, the transmitted wave, in the outlet 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 as given by Eqn. (1).

$$TL = 10log_{10}\frac{W_{inc}}{W_{tra}} \tag{1}$$

If the inlet and outlet pipes of the muffler have equal areas the ratio of the powers as given in Eqn. (1) is the same as the ratio of the acoustic intensities.

The intensity of a propagating harmonic plane acoustic wave having an amplitude of P travelling in the $+x$ direction is $P^2 / 2\rho c^2$. Thus if the values of 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 in Eqn. (1), becomes

$$TL = 20log_{10}\frac{P_{inc}}{P_{tra}} \tag{2}$$

The objective of this work is to use CFD to find the ratio P_{inc}/P_{tra} and hence the transmission loss.

Solver and Boundary Conditions

A commercial CFD package, Fluent 6.1 was used. The solver implemented was an, axisymmetric, segregated implicit solver with 2nd order implicit time stepping [4]. Second order upwind discretisation was used for the density, momentum, energy, turbulent kinetic energy and the turbulent dissipation rate equations. PISO pressure velocity coupling is used. The k-ε turbulence model [4] was used for closure.

The working fluid was air with the density modelled assuming an ideal gas and the properties shown in Table 1. The boundary conditions consist of a velocity inlet, a pressure outlet and a series of walls.

Table 1. Solution Variables

Variable	Value
Working fluid	Air
Mean pressure	101.325 kPa_{abs}
Mean temperature	300 K
Gas constant	287 kJ/kg.K
Dynamic viscosity	$1.79.10^{-5}$ kg/m.s
Ratio of Specific Heats	1.40

The velocity inlet has a time varying velocity imposed at the start of the solution in the form of a full period sinusoid with an amplitude of 0.05 m/s. After the sinusoid is complete, the velocity remains zero at the inlet for the remainder of the solution.

The pressure outlet is set at a constant static pressure, in this case atmospheric pressure.

Viscous shear stress effects are included in the calculations and heat transfer at the walls of the model are neglected.

CFD Analysis – Acoustic

The basic strategy in the CFD analysis to determine the acoustic performance is to apply a perturbation at the entrance of the inlet pipe. The effect of this perturbation at a point in the outlet pipe is established by use of CFD. The time histories of the pressures in the inlet and outlet pipes are Fourier transformed so that the transmission loss can be found. Particular care needs to be exercised so that the pressure time histories to be Fourier transformed are not contaminated by unwanted reflections.

The model has long inlet and outlet pipes, typically fourteen times the length of the expansion section. It has been found that to produce reasonable results, fourteen reflections from inside the expansion section need to be included in the time history of the monitoring point in the outlet pipe. The long inlet and outlet pipes are required to allow the reflected waves from the inlet and outlet boundaries to arrive at the monitoring points at a time after which all the necessary acoustic data has been recorded.

The CFD analysis uses an axisymmetric model. The solution space is discretised with a structured rectangular mesh.

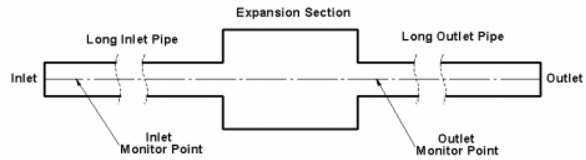

Figure 2. CFD model overview

The time histories that are Fourier transformed must be only of waves travelling from the inlet to the outlet. The reflected waves generated from the first expansion that travel back up the inlet pipe must not be included in the analysis. Likewise the transmitted wave that travels through the model and travels down the outlet pipe that is reflected at the outlet boundary must not be included in the time histories that are Fourier transformed.

The pulse used here for all the mufflers examined, is a single period sinusoid of a frequency of 3200 Hz and an amplitude of 0.05 m/s. The inlet and outlet pipe diameters d_i and d_o are 48.59mm, the expansion chamber diameter is 153.18mm and expansion chamber length varies from 280 to 540 mm. This amplitude equates to a sound pressure level of 117 dB *re* 20 μPa. The inlet boundary is a velocity inlet boundary, where time varying velocity is specified, in this case a single period sinusoid. A plane wave is imposed, so the velocity across the pipe cross-section is constant. Temperature is held constant and the flow domain determines the remaining flow variables, pressure and density. The Fourier transform of this input pulse gives broad distribution of energy in the frequency domain across all the frequencies of interest, typically 0-3 kHz.

The window used for the analysis is a rectangular window. No anti-aliasing filters are used because there is no practical method for applying a filter to the numerical time histories. This is because the CFD solves the flow at discrete points in time, and therefore the time histories are by default already sampled. However, to accurately solve the flow, the CFD solution needs to be run at a time step of 5μs, which equates to a sampling frequency of 200kHz and a Nyquist frequency of 100kHz. The wavelength at the Nyquist frequency is 3.4mm, which is shorter than the mesh spacing of 4mm. This means that the CFD will be unable to resolve frequencies above the Nyquist frequency. Therefore aliasing is not likely to be a problem.

Mesh Independence

The effect of the mesh size can be seen in Figure 3. The model is a simple expansion muffler with the following dimensions, l_e= 540 mm, d_i=d_o=48.59 mm, d_e=153.18 mm.

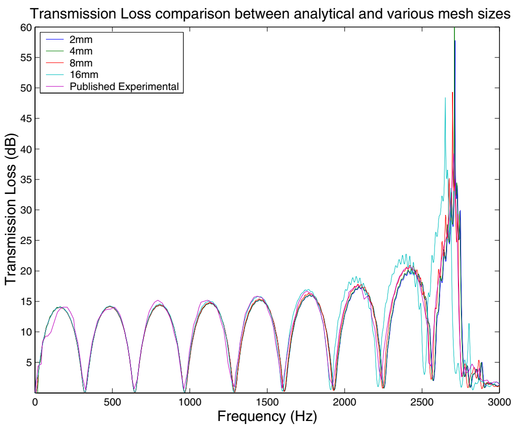

Figure 3. TL comparison between numerical results of different mesh sizes and published experimental results [1]

The lower frequencies below 1.5 kHz show close agreement with each other. For frequencies above this, the mesh influences the solution. This is due to the inability of the coarse mesh to resolve the wavelength with a sufficient number of points and introduces artificial dissipation. For example at the 2.4 kHz attenuation dome in Figure 3, as the mesh get coarser, the attenuation increases. For the remaining CFD acoustic results presented, a mesh size of 4 mm has been used, as this gives a good compromise between solution time and accuracy. For the maximum frequency of interest, 3 kHz, this equates to approximately 28 cells per wavelength for a 4mm square mesh. For comparison, the 16mm mesh has only 7 cells to resolve the same wavelength

Acoustic Results

For validation of the numerical results, published experimental results of Selamet *et al.* [1,2] of transmission loss for the different expansion chamber mufflers are used. The three cases chosen are: (a) a simple expansion chamber muffler (Figure 1); (b) a simple expansion chamber muffler with a baffle (Figure 5); (c) a simple expansion chamber muffler with a baffle and extended inlet and outlet pipes (Figure 7).

All of the CFD derived results have a similar trend of increasing attenuation at higher frequencies. This is most likely to be an artificial dissipation introduced into the model due to the discretisation in space. Dickey, Selamet and Tallio [3] studied the effects of mesh size and Courant number on the transmission loss of an expansion chamber muffler. They suggest that numerical dissipation may be an important reason for the acoustic predictions from automotive simulation codes to be inaccurate at higher frequencies.

Numerical or false diffusion arises as a result of representing a continuous fluid in a discrete form in space, time and the truncation errors associated with the discrete form of the Navier-Stokes equations used to represent the flow.

Figure 4 shows the results of the acoustic analysis of a simple expansion chamber muffler, shown in Figure 1. The muffler dimensions are: l_e= 540 mm, d_i=d_o=48.59 mm, d_e=153.18 mm.

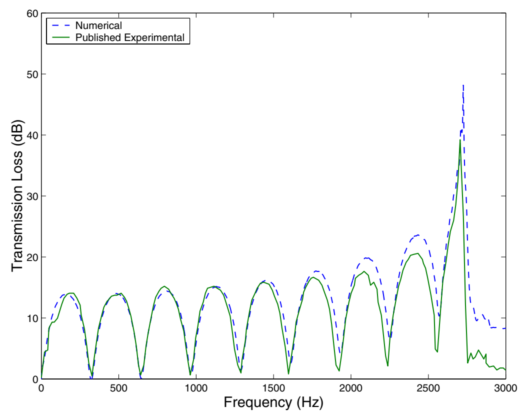

Figure 4. TL comparison between numerical results and published experimental results [1]

The numerical results obtained using CFD match very closely the published experimental results for the first three attenuation domes, or up to around 1 kHz. For frequencies above this, the CFD results are predicting an increasing rate of attenuation as frequency increases. The overall trend is followed closely.

Figure 6 shows the results of the acoustic analysis of an expansion chamber muffler with a baffle in the centre of the expansion section as shown in Figure 5.

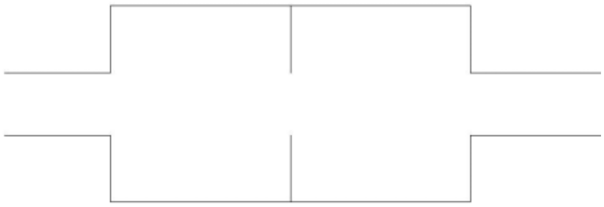

Figure 5. Expansion muffler with a centrally located baffle

The muffler dimensions are: l_e= 280 mm, d_i=d_o=48.6 mm, d_e=153.2 mm. The baffle is located in the centre of the expansion section with a hole diameter of 48.6 mm.

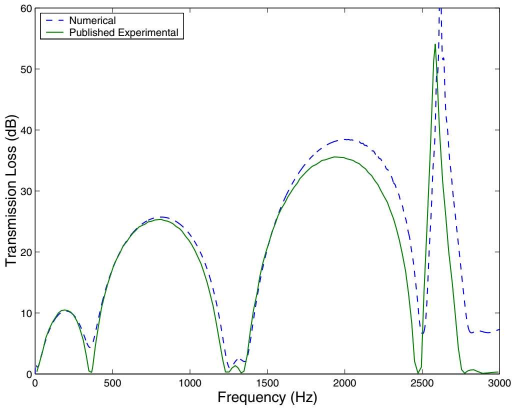

Figure 6. TL comparison between numerical results and published experimental results [5]

Similar to the previous results, the numerical results match closely at the lower frequencies, below 1 kHz. Above this the numerical results are showing a higher attenuation. However, the trend is followed well.

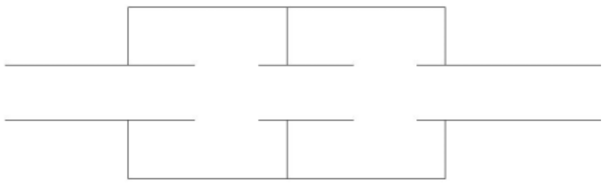

Figure 7. Expansion chamber muffler with a baffle and extended inlet and outlet pipes

Figure 8 shows the results of the acoustic analysis of a simple expansion chamber muffler with extended inlet and outlet pipes and a baffle with a section of pipe, shown in Figure 7.

The muffler dimensions are: l_e= 282.3 mm, d_i=d_o=49.28 mm, d_e=152.9 mm. The baffle is located in the centre of the expansion section with a hole diameter of 49.28 mm. The inlet pipe extends 59 mm into the expansion section, the outlet pipe extends 25 mm into the expansion section. The pipe attached to the baffle extends 25 mm into the upstream chamber and 59 mm into the secondary chamber and had a diameter of 49.28 mm.

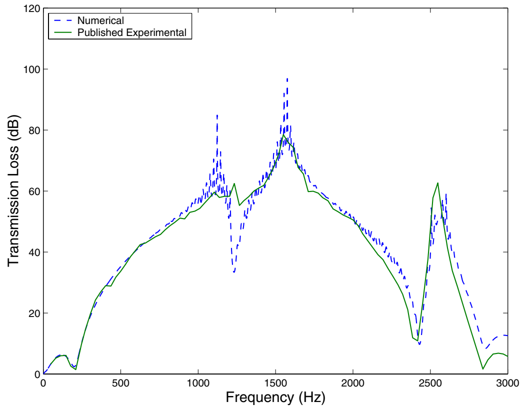

Figure 8. TL comparison between numerical results and published experimental results [5]

CFD Analysis – Mean Flow

The mean flow performance of the three mufflers considered in the acoustic analysis has been assessed. The CFD models used for the mean flow analysis are similar to the acoustic models, with the exception of the mesh and the inlet boundary condition. The mesh used for the mean flow analysis is much finer than the mesh used for the acoustic analysis, in order to resolve the boundary layer and regions of separation with sufficient accuracy. The mesh is graded such that areas of interest and large gradients such as boundary layers and the expansion section have a denser mesh and regions of less interest such as the centre of the pipe and the centre of the expansion section have a less dense mesh. Grading of the mesh reduces the total number of mesh cells required. This reduces the memory requirement and the solution time. The inlet boundary condition is much simpler with a constant velocity applied.

The three cases have been run at a series of flow speeds from 1 to 10 m/s. The pressure drop across the muffler has been calculated using a point upstream and downstream of the muffler and the results are shown in Figure 9.

The muffler with the extended pipes (Figure 7) has the lowest pressure drop due to the flow remaining attached to the pipes as is travels through the expansion section. The simple expansion muffler (Figure 1) has the highest pressure drop of the mufflers considered. This is due to the sudden expansion and sudden contraction, which induce significant losses. The dual chamber muffler (Figure 5) exhibits less pressure drop than the simple expansion chamber muffler. However, the pressure drop is still much higher than the muffler with extended inlet/outlet pipes.

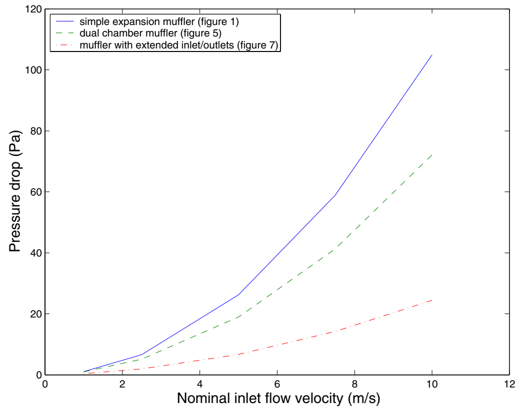

Figure 9. Pressure drop versus flow velocity

Conclusions

The acoustic numerical results obtained using CFD show good agreement with published experimental results. The lower frequencies below approximately 1 kHz are in close agreement. For frequencies above 1 kHz, the predicted attenuation is above the published experimental results, however the trend is followed closely.

CFD can be successfully used to evaluate both the mean flow and acoustic performance of an expansion chamber muffler, with various modifications including baffles and extended inlet/outlet pipes.

Acknowledgments

The correspondence author wishes to thank the Australian Research Council and Colpro Engineering for financial support.

References

1. Selamet A, Ji ZL, "Acoustic attenuation performance of circular expansion chambers with extended inlet/outlet", *Journal of Sound and Vibration*, 223(2):197-212, 1999
2. Selamet A, Radavich PM, "The effect of length on the acoustic attenuation performance of concentric expansion chambers: an analytical, computational and experimental investigation", *Journal of Sound and Vibration*, 201(4):407-426, 1997
3. Dickey NS, Selamet A, Tallio KV, "Effects of numerical dissipation and dispersion on acoustic predictions from a time-domain finite difference technique for non-linear wave dynamics", *Journal of Sound and Vibration*, 259(1):193-208, 2003
4. Fluent Inc, "Fluent 6.1 Users Guide", 2003
5. Selamet A, Denia FD, Besa AJ, "Acoustic behaviour of circular dual-chamber mufflers", *Journal of Sound and Vibration*, 265(5):967-85, 2003