International Journal for Multidisciplinary Research (IJFMR)
E-ISSN: 2582-2160 ● Website: www.ijfmr.com ● Email: editor@ijfmr.com

Flow Induced Noise Prediction in Automobile Exhaust Systems Using Computational Fluid Dynamics (CFD)

R Velmurugan[1], M Sathiyamoorthy[2]

[1]Senior Engineer, Chassis 2, Platform Development, Renault Nissan Technology & Business Centre India
[2]Senior Manager, Chassis 2, Platform Development, Renault Nissan Technology & Business Centre India

Abstract:
Exhaust (tailpipe) noise is an important regulation criterion for automobiles in homologation phase. The exhaust noise must meet the target dB value set by the country where the vehicle will be sold. In a typical Internal Combustion (IC) engine, the exhaust gas is pushed from the engine cylinder at the end of exhaust stroke through the exhaust system along mufflers to tailpipe. The exhaust gas product will carry two types of noise as it reaches the tailpipe, low frequency and high frequency (flow induced) noise. The high frequency noise is generated by turbulence of the gas as it passes down the exhaust all along pipes and mufflers. Computational Fluid Dynamics (CFD) can be used to calculate the high frequency noise sources inside the pipes and muffler system at any Engine Operating Point (EOP). The high frequency noise prediction can be used to optimize the gas flow distribution and noise levels.

Keywords: Automobile muffler systems, CFD, high frequency noise, flow induced noise, Sound Power Level (SPL)

1. Introduction:

J.M Middelberg et.al, conducted CFD study for acoustic and mean flow performance of simple expansion chamber mufflers and concluded that CFD can be used as a tool for acoustic study in mufflers [1].
Jun Chen and Xiong Shi has studied the impact of internal flow field in muffler design using CFD [2]
Exhaust systems acoustic behaviour was studied using CFD in the work [3], [4] & [5]
Flow induced noise in exhaust system is important because it has significant contribution in the overall noise from the exhaust. By predicting the flow induced noise source, design optimization can be made to regulate the gas flow inside the mufflers and pipes.
Commercially available CFD code is used to simulate the gas flow inside the exhaust system by solving the governing equations.
Conservation of Mass (Continuity Equation):

$$\boldsymbol{\nabla}.(\boldsymbol{\rho}\mathbf{u}) = \mathbf{0} \qquad (1)$$

Conservation of Momentum (Navier-Stokes Equation):

$$\boldsymbol{\rho}\left(\frac{Du}{Dt}\right) = -\nabla p + \boldsymbol{\mu}\nabla \mathbf{2u} + \mathbf{f} \qquad (2)$$

Conservation of Energy:

$$\rho C_p \left(\frac{DT}{Dt}\right) = \nabla \cdot (k \nabla T) + Q \quad (3)$$

Where,
ρ-Fluid density (Kg/m^3)
f-body force per unit volume
u-velocity vector
C_p-Specific heat capacity (J/(Kg.°C))
p-pressure (MPa); T-Temperature (K)
μ-dynamic viscosity ((N.s)/m^2)
k-thermal conductivity(W/(m.K))
Q-Heat source

1.1 3D CFD Model:

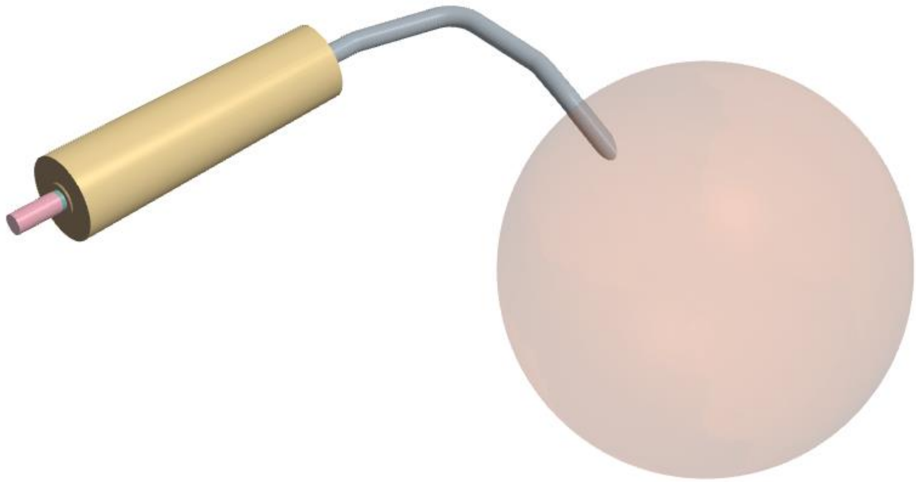

Figure 1: Fluid Volume of Exhaust system

The fluid volume of the exhaust system is extracted from raw CAD data and approximated to generate a good quality mesh.

A sphere with 10 times the diameter of tailpipe is modelled near the tailpipe to conveniently define the outlet boundary condition and capture the gas jet stream exiting the tailpipe. The extracted fluid volume of exhaust system along with the spherical volume is shown in Figure 1

2. Boundary Conditions:

Boundary	Type	Value
Inlet	Mass-flow & Temperature inlet	363 kg/hr; 880 ºC
Pipes and Mufflers	Wall	No-slip
Outlet	Pressure Outlet	0 mbar
Operating pressure= 1013.25 mbar		

Table 1: Boundary conditions for CFD analysis

Table 1 shows the boundary conditions defined for performing CFD analysis.

Thermal boundary conditions for pipes and muffler walls are defined with ambient temperature, external convection co-efficient and emissivity similar to bench test conditions.

3. Model Discretization and Turbulence Modelling:

For this analysis to calculate the high frequency noise from the tailpipe of exhaust systems, a polyhedral mesh with a base size of 2.5 mm was utilized. This mesh provides a good balance between computational efficiency and accuracy. Appropriate mesh refinement was done to precisely capture the perforations in pipes and baffles, Prism layers were incorporated to refine the boundary layers, with their height and number of layers calculated based on the y-plus value to ensure proper resolution of near-wall effects. The realizable k-epsilon turbulence model with all y-plus treatment was employed to accurately capture the turbulent flow characteristics. A grid independency study was performed using base sizes of 2 mm and 3 mm. The study revealed that there was minimal to no impact on back pressure and sound power level results between the different grid sizes, confirming that the chosen mesh resolution is sufficient for capturing the critical flow features and thermal behaviour in the exhaust system.

4. Results and Discussion:

A steady state CFD analysis was carried out for high Engine RPM. At high engine RPM the exhaust gas flow rate and temperature will be high which will create high turbulence inside the system. Accessing noise levels at high turbulent flow condition will give us clear understanding of its sources. High frequency noise source and tailpipe noise are calculated from the sound power level empirical relation. The sound power level (SPL) [dB] is given by,

$$\boldsymbol{SPL\ (dB) = 10 * log10 * \left(\frac{SPL\ Emperical\left[\frac{W}{m3}\right]}{1*10-12}\right)} \qquad (4)$$

SPL at tailpipe of exhaust system at 4500 engine RPM in normalized scale is shown in Figure 2.

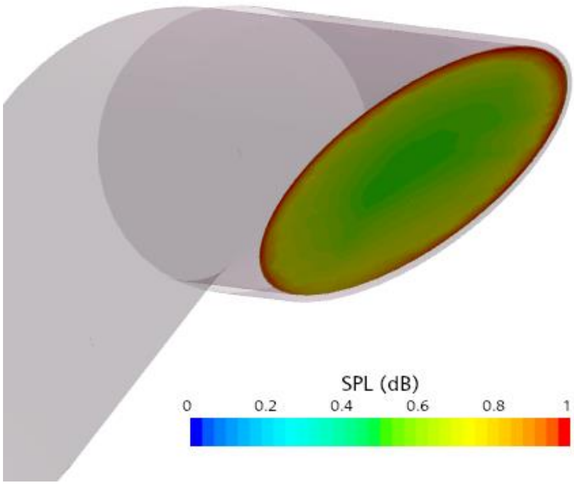

Figure.2 Tailpipe SPL (dB)

The velocity jet stream exiting the tail pipe is shown in Figure 3. The velocity magnitude is show in a normalized scale.

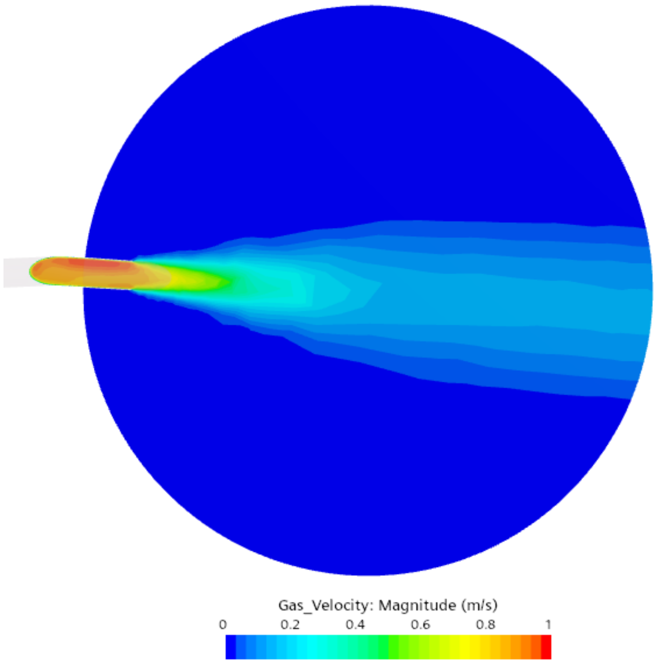

Figure 3: Tailpipe Velocity, normalized scale (m/s)

High Frequency noise sources are mainly from perforations and constricted areas inside the muffler systems where the turbulence will be high. Noise sources at 4500 engine RPM is shown in Figure 4

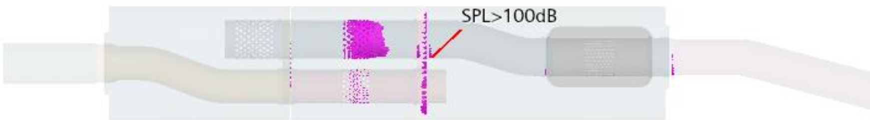

Figure 4: High Frequency Noise sources

The steady state analysis is done for 3 engine RPMs, 3025, 3700 and 4500 to compare it with the test data. The high frequency noise at tailpipe for CFD and test comparison shows, there is maximum of 6.9% deviation from test results at 3700 engine RPM. Figure 5 shows the comparison of CFD and test results of high frequency SPL.

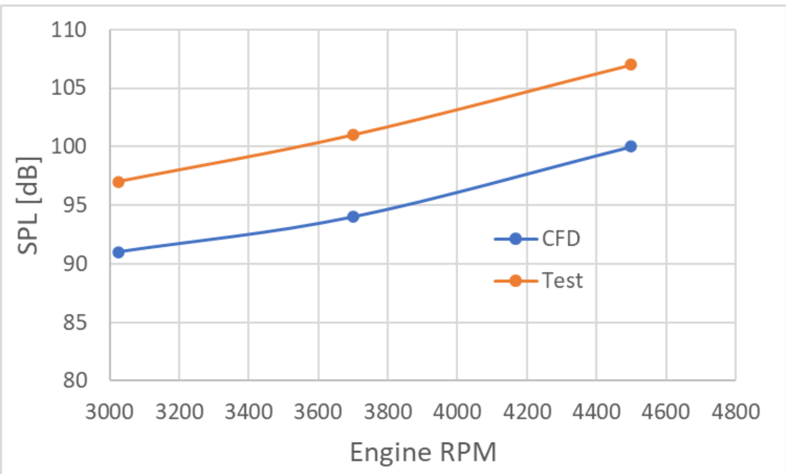

Figure 5: High Frequency Noise @ Tailpipe, CFD vs Test

5. Conclusion:

In this study an attempt was made to calculate high frequency noise of automobile exhaust systems using CFD. A maximum deviation of 6.9% in high frequency noise calculation was seen at 3700 engine RPM. This deviation in results can be due to the assumption of steady state analysis for noise calculation. In reality the exhaust gas flow nature will be highly transient. By considering transient flow analysis the noise results accuracy can be improved. However, the steady state analysis can be used to get an overall idea of high frequency noise in the exhaust system. Future work will be extended with transient analysis to improve the accuracy of CFD results.

6. References:

1. J.M. Middelberg, T.J. Barber, S.S. Leong, K.P. Byrne and E. Leonardi "CFD analysis of the acoustics and mean flow performance of simple expansion chamber mufflers"
2. Jun Chen, Xiong Shi "CFD Numerical Simulation of Exhaust Muffler"
3. Yoshisiro Isshiki, Yuzuru Shimamoto, Tomoyuki, Wakisaka. Simulation Prediction of Pressure Losses and Acoustic Characteristics in Silencers by Numerical Simulation[C] . SAE Paper 960637 1996.
4. Mackey D O, Blair G P, Fleck K. Correlation of Simulated and Measured Noise Emission Using a Combined 1D/3D Computational Technique[C]. SAE Paper 970801 1997.
5. A Selamet, F.D. Denia and A.J. Besa, 2003, "Acoustic behaviour of circular dual-chamber mufflers", Journal of Sound and Vibration, vol.265, pp. 967-85.