

Flow Noise Predictions for Single Cylinder Engine-Mounted Muffler Using a Lattice Boltzmann Based Method

Adrien Mann, Raj Nair, and Jaspreet Singh Gill
Exa Corporation

Brett Birschbach and Patrick Crowley
Briggs & Stratton Corp

ABSTRACT

Exhaust systems including mufflers are commonly mounted on engines to reduce the firing cycle noise originating from the combustion process. However, mufflers also produce flow-induced self-noise, originating from the complex flow path throughout the muffler. As an engine prototype is not available in the early stages of a development program, it is challenging to assess the acoustic performance of the full system when only experiment is available. It is also difficult to pinpoint the design features of a muffler generating noise, as a portion of the noise is generated internally.

Numerical approaches are a possible alternative. However, capturing non-linear dissipation mechanisms and thermal fluctuations of exhaust flows is challenging, while necessary to accurately predict flow noise. Transient and compressible Computational Fluid Dynamics and Computational AeroAcoustics (CFD/CAA) Lattice-Boltzmann based Methods (LBM) have previously been successfully applied to quantify the flow noise generated by mufflers mounted on an experimental cold flow test bench.

In this paper, the accuracy of the method for self-noise predictions of a muffler mounted on a single cylinder engine is demonstrated comparing results with experimental noise measurements obtained in a hemi-anechoic room. The inlet boundary condition of the muffler is calculated using a 1-D system modeling tool, providing oscillating transient temperature and mass flow rate. Measured temperature on the surface of the muffler is prescribed in the simulation on the geometry walls. After validation of the results, the simulation data is further post-processed to identify flow-induced noise sources in the system and propose design changes to reduce self-noise. This study confirms that this numerical approach can be used in a production process to quantify and reduce flow noise in mufflers.

CITATION: Mann, A., Nair, R., Gill, J., Birschbach, B. et al., "Flow Noise Predictions for Single Cylinder Engine-Mounted Muffler Using a Lattice Boltzmann Based Method," *SAE Int. J. Engines* 10(4):2017, doi:10.4271/2017-01-1797.

INTRODUCTION

One of the major drawbacks of combustion engines is the noise originating from the firing cycle, also known as engine orders. To mitigate this issue, exhaust systems including mufflers are commonly mounted on engines. As community noise and work environment regulations become increasingly more stringent, engine and muffler manufacturers must keep improving the acoustic performances of their products. While the main purpose of this system is to reduce the intensity of engine orders, the induced back pressure must be kept at a minimum to guarantee optimal engine operating conditions. Achieving such performances, however, implies the increasing complexity of muffler designs, often leading to the emergence of undesired noise produced by the flow circulating inside a muffler, or muffler self-noise. Addressing those issues early in the development process using an experimental process based on prototyping can be complex, time-consuming and expensive.

To recover the self-induced noise of mufflers at various engine operating conditions, flow and acoustics must be accurately calculated simultaneously. However, typical muffler geometries with characteristic dimensions from millimeters to decimeters are a challenge as numerical codes are often not performing well when working over a broad range of characteristic lengths. Such geometries often include perforate holes, of the order of few millimeters, which tend to drive the maximum resolution of the numerical simulation, and increase the computational cost of simulations. In addition, the temperature field in operating mufflers is not uniform, as the hot flow enters the exhaust system from the engine outlet and exits to atmospheric conditions, with heat loss happening through the muffler shells. Changes in temperature field can have multiple effects: on one hand, a hotter flow will tend to increase flow velocity magnitude, increasing the intensity of flow-induced noise sources. On the other hand, the speed of sound is a function of temperature, which varies depending on the operating conditions, causing changes in resonances and propagation of acoustic waves.

In the early stages of the design, neither prototypes of the muffler nor of the vehicle/engine are available. Consequently, experimental testing is not feasible. However, engine fluctuating outlet conditions can be predicted [1,2], and the Computer Aided Design (CAD) geometry is already available. These inputs enable early numerical simulations capable of predicting the acoustic performance of exhaust systems, when accurate, as well as the investigations of noise generation mechanisms which can be used to guide the acoustic design of the exhaust system [3,4]. The traditional approach of Computational Fluid Dynamics + Computational Aero Acoustics using Navier-Stokes solvers has been more or less successfully explored in the past for this application [5]. But the complexity of the design, the small length scales, and the consequent human and computation time cost remains challenging for industrial applications.

The present study focuses on exploring the numerical simulation of flow-induced noise in muffler in realistic conditions, and the analysis of the associated noise generation and propagation mechanisms. Using Lattice Boltzmann based Methods, previous studies have shown that flow-induced noise levels can be accurately predicted with a steady flow at ambient temperature [3,6,7,8]. Such transient compressible methods extend naturally to simultaneous changes in temperature, and thus can model the hot and fluctuating flow conditions from the real engine firing cycle [4]. After a review of the numerical approach, the study focuses first on the correlation of simulated Sound Pressure Levels (SPL) to the measured noise levels at various microphone locations. Then, the simulation data is analyzed, using advanced post-processing techniques, to identify flow-induced noise sources and the associated noise-generating design features, and acoustic behaviors. Finally, design modifications based on the preceding analysis are implemented, and the resulting noise levels reduction is validated both numerically and experimentally.

EXPERIMENTAL ENVIRONMENT

For this study, the measurements are obtained in a hemi-anechoic room with the muffler system mounted on a single cylinder engine. The engine itself is mounted on a rig grounded to the floor, driving a belt. The engine is operating at Wide Open Throttle (WOT) conditions at 3600 RPM. Skin temperatures at the end of test are stabilized.

Sound Pressure Levels are measured at multiple locations while the engine is operating. The spatial location of the microphones is presented in Figure 1. The first set of microphones corresponds to far-field measurements (Microphones 1 to 5 in Figure 1). This set of microphones helps at quantifying the directivity of the noise propagation away from the muffler. The second set of probes corresponds to the near-field microphones (Microphones 6 and 7 in Figure 1). These near-field probes are close to the muffler flow exit, 30 cm away from the muffler exit at 45° from the jet flow axis, and are ideal to capture the muffler flow-induced noise contribution without contamination from other sub-systems located further than the muffler outlet/jet.

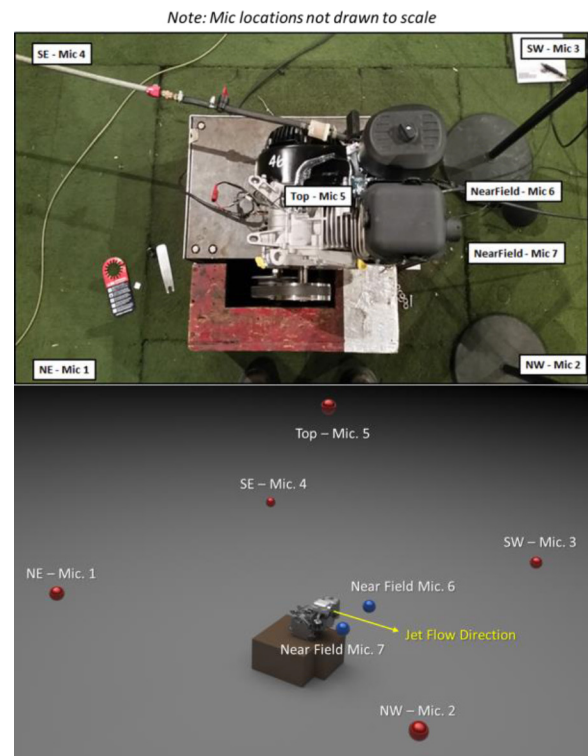


Figure 1. Experimental setup with approximate microphone locations. Engine mounted on rig with driving belt. Experimental (top) and simulation (bottom) models.

However, far-field mics capture a range of other noise sources apart from the muffler noise, such as the running belt, the engine, or the cooling fan. These additional noise sources can significantly contaminate the comparisons between the test and simulation at these microphones. These effects will be considered based on the microphone locations and the comparison of Sound Pressure Levels between experiment and simulation.

NUMERICAL DETAILS

Numerical Approach

Flow noise simulations for the muffler system in this paper were performed using the PowerFLOW 5.3 commercial CFD solver. The CFD/CAA solver PowerFLOW 5.3 based on the Lattice Boltzmann Method is used to calculate unsteady flow physics and the corresponding flow-induced noise generation and radiation. Lattice-based methods are by nature explicit, transient and compressible, and are an alternative to traditional CFD methods based on the discretization of the Navier-Stokes equations and derived variations [9, 10, 11, 12, 13, 14]. The basic idea of LBM is to track the advection and collisions of fluid particles. Since the average number of particles in a representative volume of fluid far exceeds the computing power required to track them individually, the particles are grouped into an integer number of discrete directions with index i . The computation follows the particle distribution function f_i which represents the number of particles per unit of volume, also called

voxel, at a specific time and location moving with velocity c_i . As in statistical physics, the flow variables such as density and velocity are determined by taking the appropriate moments, i.e. summations over the set of discrete directions, of the particle distribution function [15, 16, 17, 18, 19, 20, 21, 22, 23]. Recent improvements in modeling the temperature scalar equation have further enhanced the accuracy of the temperature calculations in the flow solver [24]. LBM is used and validated across many aeroacoustics applications such as automotive wind-noise [25,26], HVAC system noise [27,28], acoustic propagation [29,30], airframe noise [31,32], sunroof buffeting [33] and exhaust systems noise [3,6,7,8]. Further details on LBM can be found in the cited references.

In this presentation, post-processing results of Exa's Flow-Induced Noise Detection (FIND) module in PowerACOUSTICS are presented, which output quantity represents the flow-induced noise sources in the system. This module uses an innovative approach based on the dynamics of the flows structures and Alan Powell theoretical work on vortex sound [34] to calculate the noise sources topology and strength. FIND has already been applied in the literature as a noise sources detection tool for exhaust noise [4], HVAC systems noise [35], wind noise [36] and fan noise [37].

Simulation Model

The simulation model includes the geometry of the muffler mounted on the engine, placed on a mockup of the test rig. The model contains all the complex geometric components of the real muffler system and closely mimics the actual test rig setup. The simulated version of the muffler is presented in Figure 2. As shown in this figure, the simulation model includes the engine and air cleaner geometry to ensure the propagation, reflection and scattering of acoustics are accurately captured. This model will be labeled as baseline for this study.

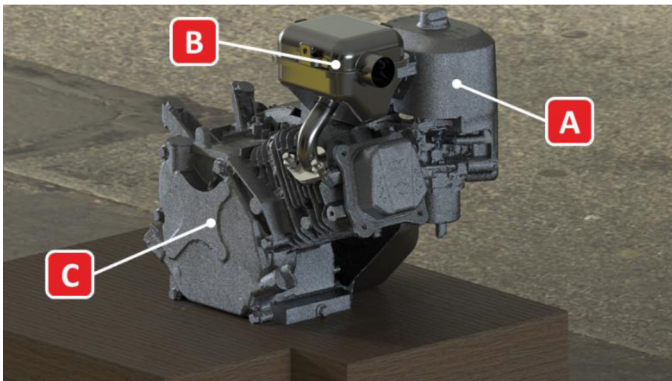


Figure 2. Geometry model used in the numerical simulation. A: Air Cleaner, B: Muffler, C: Engine block.

Since the current study is focused on the prediction of the noise levels originating from the flow-induced noise sources inside the muffler and in the jet flow at the tailpipe exit, other noise sources which could be contaminating the experimental results are not captured in this model. This limitation and its consequences on the predicted levels will be discussed when simulated and measured Sound Pressure Levels are compared.

Simulation Parameters

Measuring the inflow data at the inlet of the installed muffler is challenging. To correctly model the fluctuations in mass flow rate and temperature at the muffler inlet, these quantities are calculated using a 1-D system modeling tool which provides oscillating transient temperature and mass flow at the desired operating conditions. This data for the tested operating point at 3600 RPM WOT is presented in Figure 3. This fluctuating inlet flow conditions are then applied to the simulation model on the inlet pipe cross-section, at the intersection of the elbow pipe with the muffler, as shown in Figure 4. This ability to model the fluctuating boundary condition is critical to accurately represent the test data.

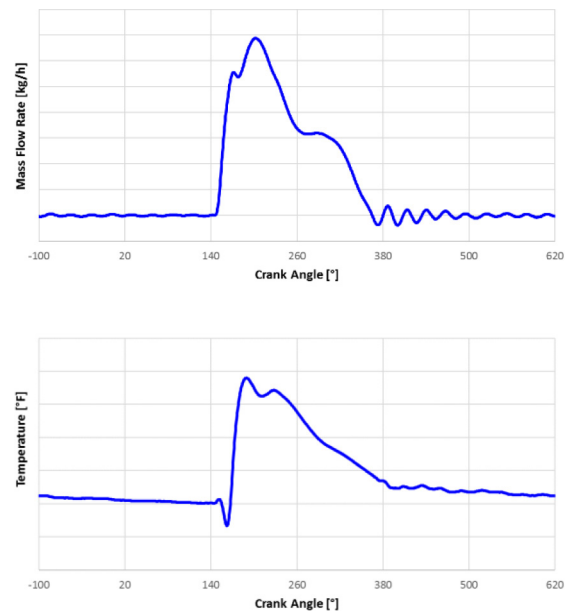


Figure 3. Inlet flow boundary conditions from 1D modeling tool: mass flow rate (top) and temperature (bottom) at 3600 RPM Wide-Open Throttle (WOT).

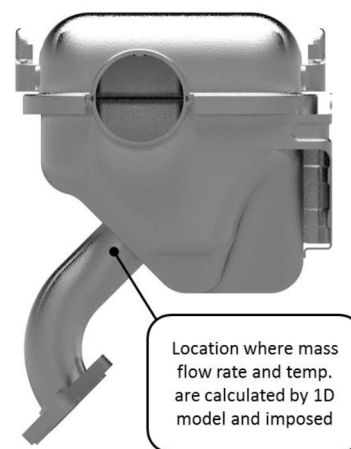


Figure 4. Inlet flow boundary conditions location. The inlet flow is imposed on the inlet pipe cross-section where the inlet pipe intersects the muffler shell.

As shown in previous studies [4], a correct modeling of the thermal conditions is critical for an accurate prediction of noise levels at operating conditions. Temperature changes induce local density and velocity changes as well as local changes in the speed of sound, which are then critical for the correct prediction of noise sources intensity, as