

Computational Aero-Acoustics Simulation of Automotive Radiator Fan Noise

Ahsanul Karim, Meisam Mehravaran, Brian Lizotte, Keith Miazgowicz, and Yi Zhang
Ford Motor Co.

ABSTRACT

Flow bench and engine testing can be used to detect flow induced noise, but understanding the fundamental mechanisms of such noise generation is necessary for developing an effective design. This paper describes Computational Aero-Acoustic (CAA) analyses performed to obtain the broad-band and BPF noise sources

A computational aero-acoustics simulation on the aerodynamic noise generation of an automotive radiator fan assembly is carried out. Three-dimensional Computational Fluid Dynamics (CFD) simulation of the unsteady flow field was performed including the entire impeller and shroud to obtain the source of an audible broad-band flow noise between 2 to 4 kHz. Static pressure probes placed around the outer-periphery and at the center of the impeller inlet side and, at the shroud cavities to capture the noise sources. The static pressure at all probe locations were FFT (Fast Fourier Transform) processed and sound pressure level (SPL) was calculated.

The CAA simulations discussed in this paper identifies the SPL from the fan outer-periphery probes show the dominant source of blade passing frequency (BPF) and the broad-band noise. The BPF level is the strongest in fan outer-peripheral region because of large pressure fluctuations as a result of blade-passing. The broad-band noise on the other hand is the results of tip-leakage interactions with shroud surface and, vortex-vortex and vortex-surface interactions.

CITATION: Karim, A., Mehravaran, M., Lizotte, B., Miazgowicz, K. et al., "Computational Aero-Acoustics Simulation of Automotive Radiator Fan Noise," *SAE Int. J. Engines* 8(4):2015, doi:10.4271/2015-01-1657.

INTRODUCTION

With the advent of Eco-Boost engines, the demand for increased power density keeps enhancing continuously. In order to maintain the higher power density demand and avoid overheating problem, the cooling system design becomes an important and challenging task. The cooling fan plays an important role in the thermal system design and, is designed in various sizes and specifications to meet different demands of heat dissipation. In addition, engine room for recent vehicles is getting smaller, so the packaging room for cooling fan system has diminished. These further complicate the under-hood packaging environment, radiator fan and shroud design, and create new Noise, Vibration and Harshness (NVH) challenges. The cooling fan BPF and flow noise that span the audible range can be very objectionable inside the vehicle.

The cooling fan noise is often dominated by blade passing frequency (BPF) and its higher harmonics noise, can also generate audible broad-band flow noise. Many of the cooling fan NVH issues are related to the shroud design or tip clearance. For example, flow interactions with shroud surface and flow leakage through tip clearance of the fan impeller (blades) can create objectionable BPF noises [1]. Tip clearance flow in fans and compressors has been widely studied and documented [2, 3, 4 and 5].

Other noise issues are related to the operating conditions of the fan. For examples, the broad-band noises are the results of aerodynamic turbulences created at the inlet or at the outlet of the fan due to flow collision/interactions.

In order to meet high customer expectation, every NVH error state mentioned above needs to be addressed without sacrificing fan performance. This paper focuses on understanding the flow dynamics around the impeller and interactions with shroud surface, which is needed to develop an effective design. Three-dimensional CFD modeling is used to understand the flow dynamics in and around the fan in the presence of shroud.

BACKGROUND

The acoustic behavior of airfoils at low flow rates and high angle of attack has been studied and is well understood. The aircraft noise signature during the approach maneuver prior to landing is termed airfoil self-noise. This noise produced by the interaction of an airfoil blade and the turbulence in its own boundary layer and wake regions. At airfoil stall, turbulent boundary layer trailing edge noise created by separation can increase by more than 10 dB compared to the noise emitted at low angle of attack [6].

Similarly, the flow-flow and fluid-surface interactions are the main acoustic sources of noise in axial fans and compressors. The presence of tip clearance in axial fans and compressor has significant impact on aerodynamic performance and stall margin. As the blade rotates, the fluid that passes through the blade tip due to blade pressure and suction side pressure difference creates a vortex (tip clearance vortex) in the tip clearance region and act as a blockage to the through-flow. Another flow stream the 'tip leakage' or reverse flow develops due to the presence of tip clearances in axial fans and in compressors is contributed by the pressure difference between suction side and pressure side of the duct. The leakage fluid travels opposite in direction with respect to the main through-flow result in the formation of a shear-layer [7]. The interactions between this reversed flow and the tip clearance vortex causes the vortex to roll-up. This vortex then loses its strength and grows in size. The tip clearance vortex then convects downstream [8].

The pressure pulses caused by the blade passing are responsible for the narrow-band or tonal BPF noise (dipole source $\sim M^3$) and, the flow interactions of the tip clearance vortex and the reverse flow are responsible for broad-band noise (quadrupole sources $\sim M^5$). Since the pressure difference across the blade is not large for this kind of applications, the leakage flows discussed here have low-Mach (M) number. Because of low-Mach number quadrupole sources are dominated by dipoles.

CFD APPROACH

The Computational Aero-Acoustics (CAA) resolves the acoustic pressure fluctuations as part of Computational Fluid Dynamics (CFD) solution. In the present CAA analysis, turbulence is simulated using the Large Eddy Simulation (LES) method with appropriate eddy-viscosity model for the sub-grid stress tensor. Computed data were taken at different locations in the shroud and at the inlet side of the fan. To preserve the acoustic information, emanating from the fan and shroud cavities, data monitors were placed close to the noise source; the fan impeller and shroud cavities. To minimize numerical dissipation and dispersion of acoustic information, certain grid density, time steps and proper boundary conditions (to minimize boundary reflection) were applied.

A general purpose commercial CFD code with finite volume method based Navier-Stokes equation solver was used. The governing equations were solved using coupled implicit method. The Pressure-Velocity coupling is calculated through SIMPLE algorithm. A second order temporal scheme is used and a number of sub-iterations are performed to reduce the residual below an acceptable value at each time step. Therefore, a higher *Courant* number could be maintained without deteriorating the solution accuracy.

The 3D CFD simulation is performed using the entire fan impeller and the shroud. The impeller consisted of eight blades. The radiator is modeled as a porous media and porosity was calculated based on the pressure drop at different flow rates test data.

CFD MODEL

The model includes the radiator at the fan inlet, the impeller, the shroud and the fan inlet/outlet duct as shown in [Figure 1](#).

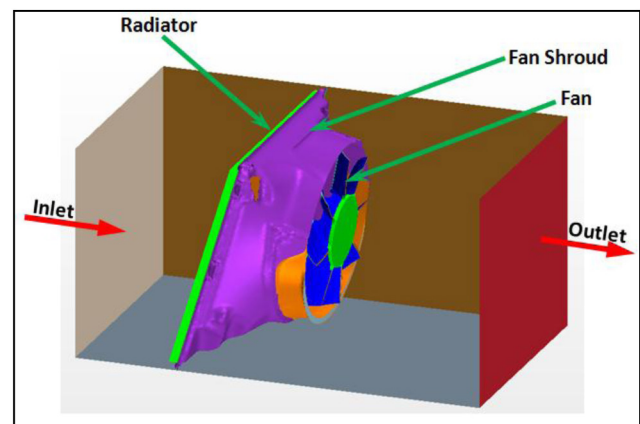


Figure 1. Computational Domain

The impeller consisted of eight blades. The trailing edge of blades was exposed to downstream open flow and the shroud was covering only a part of the blade chord length in the leading edge region. The clearance between shroud and the blade tip was about 1.0 inch to compensate for engine roll in the vehicle.

The computational domain is composed of three zones and connected through arbitrary interfaces. Surfaces between the fan inlet zone and the fan impeller, and the shroud and the outlet define the grid interfaces needed for the relative motion of the fan impeller. The sliding mesh between interfaces around the fan impeller allows that relative motion, as a solid-body rotation of the impeller used in the simulation. The mesh parameters are different for each zone. The geometrical discretization for all three zones is made using polyhedral mesh.

The stagnation pressure boundary condition was specified at the fan inlet and the mass-flow rate was specified at the outlet. At the inlet, there is an unavoidable effect on the final flow solution as a result of the pressure boundary condition. A reasonable length was added to the fan inlet geometry to avoid pressure reflection effect as much as possible. The radiator upstream of the shroud is modeled as a porous media and has the same dimensions as in vehicle.

The shroud shape is rectangular at the inlet and circular at the fan end. At the rectangular end the outside dimensions of the shroud are 800 mm $\times$ 700 mm. To eliminate the influence of inlet and outlet boundaries the fan is placed in a larger domain. The length of the computational domain is 1350 mm. The radiator is included in the computational domain. The overall dimensions of the computational domain, radiator, shroud and fan used in CFD model, are shown in [Figure 2](#).

The static pressure signals, of the CFD solution, were taken using 'point' probes at the outer-periphery and at the center of the impeller inlet side and, at the shroud cavities, as shown in [Figure 3](#), to capture narrow-band BPF and 2 to 4 kHz broad-band noise source. The pressure signals were recorded in time. Pressure signals from 'point' probe measurements were FFT processed and used for spectral distribution and Sound Pressure Level (SPL) calculations, is presented in this paper.

The CFD simulation is performed at a fan speed of 2600 rpm and mass flow rate 4.07 kg/s. The blade passing frequency (BPF) is calculated using [equation 1](#)

$$f_{BPF} = \frac{ZN}{60}$$

(Eq. 1)

Where, Z is the number of fan blades, N is fan rotational speed in RPM.

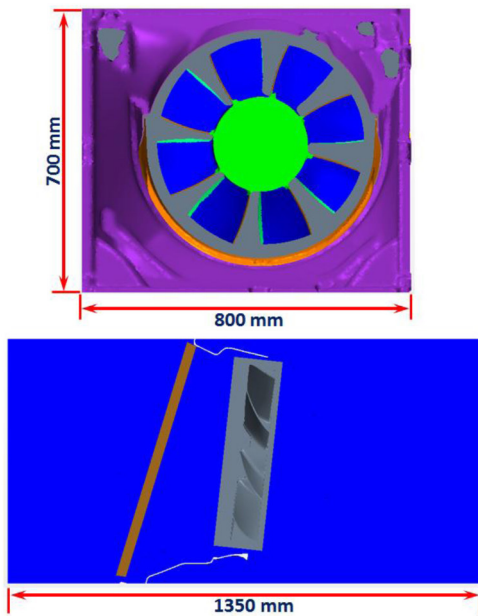


Figure 2. Radiator and Cooling Fan System

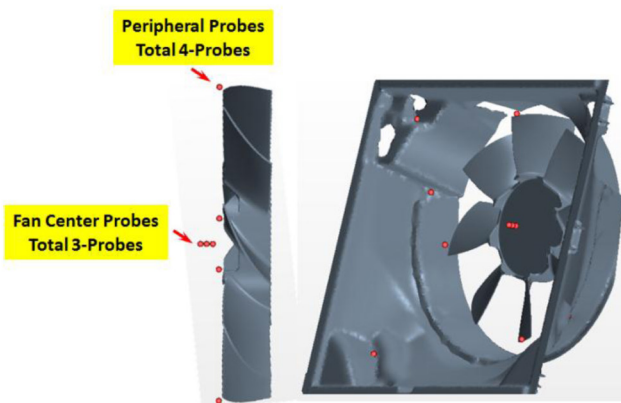


Figure 3. CFD Pressure Measurement Locations

RESULTS AND DISCUSSIONS

The pressure fluctuations at the fan periphery and spectrum of sound pressure level at different locations in the radiator-fan system are analyzed to capture narrow-band BPF and broad-band 2 to 4 kHz noise sources.

The pressure fluctuations at the fan peripheral location is shown the [Figure 4](#). [Figure 4a](#) shows the pressure fluctuations for one revolution of the impeller. As the fan blade passes the probe location it creates a

pressure pulse and there are eight pulses over one revolution period of the impeller as there are eight blades. These pulses are typical of blade passing pulses and causes narrow-band blade passing frequency noise. [Figure 4b](#) is the enlarge view of one pulse showing peak and valley of the pressure signal shown in [Figure 4a](#). The valley of the pressure pulses shows high frequency fluctuations on the top of low frequency blade pass pulses.

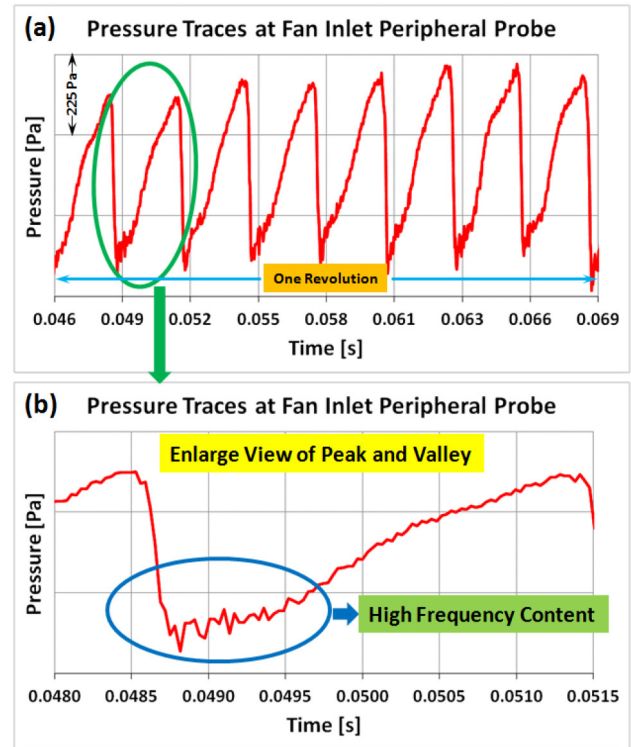


Figure 4. Pressure Fluctuations at Fan Peripheral Location

[Figure 5](#) shows the spectrum of sound pressure level of the pressure signal shown in [Figure 4](#). The green line on the plot is the moving average (or filtered) line showing average SPL. In the SPL spectrum it is apparent that the spectral characteristics of the fan noise are dominated by the BPF and its harmonics. The spectrum shown in [Figure 5](#), also, shows a hump in the frequency range of 2 to 4 kHz. This broad-band noise is the result of high frequency pressure fluctuation content in the pressure signal as shown in [Figure 4b](#).

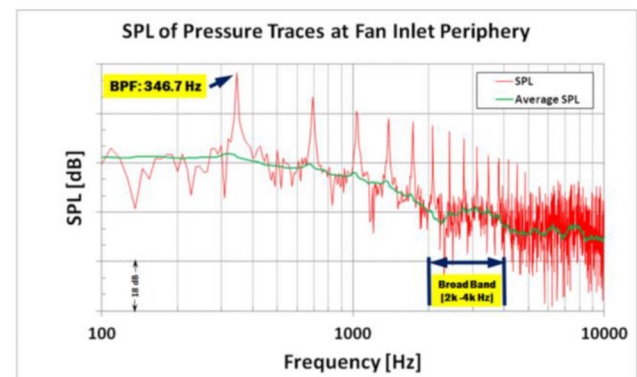


Figure 5. Spectrum of Sound Pressure Level (SPL) at Fan Peripheral Location