Engine Cooling Fan Noise Prediction Based on Hybrid Method and Aero-Acoustic Analogy

Masaaki Kabuto  
MSC Software Corporation, Osaka, Japan  
email: masaaki.kabuto@mscsoftware.com

Yuta Tsujimoto  
Kubota Corporation, Osaka, Japan

Maxime Raskin, Yves Detandt, Alexis Talbot, Bastien Ganty  
Free Field Technologies SA, MSC Software Belgium, Mont Saint Guibert, Belgium

Appropriate thermal management greatly contributes to efficiency of internal combustion engines. For large engines used in agricultural or construction vehicles, the engine cooling fan plays an essential role and must be carefully designed for optimal performances. The noise generated by these fans dominates among other noise sources and mainly affects the operators while the noise restrictions may reduce the usage of the vehicles. Computational Fluid Dynamics (CFD) is commonly used to improve the aerodynamic performances of cooling fans but the computation of the acoustics is computationally expensive using only CFD solutions. In this paper a hybrid method based on an aero-acoustic analogy is applied to compute aero-acoustic sources using results from an unsteady incompressible CFD analysis. Aero-acoustic sources are then used in an acoustic propagation finite element simulation to compute the near and far field acoustic pressure around the fan. The method is applied on an industrial engine cooling fan and results obtained through numerical simulations are compared with experimental results for validation. Parameters of the CFD and acoustic analysis are detailed and the ability of the method to capture tonal and broadband contributions to the overall fan noise is assessed.

Keywords: aeroacoustics, fan noise, simulation, hybrid method

1. Introduction

The construction vehicles are powered by large engines which need to be cooled down by fan systems extracting the heat to the exterior. These fans are generating an important noise contribution to the operator and to the people living in the neighbourhood of working area. To prevent a limited usage of the vehicles to comply with noise restrictions rules, the fan needs to be designed to be as quite as possible while delivering the same cooling performances. This optimization is powered by simulation techniques delivering the noise emitted by different fan configurations. While the acoustic pressure fluctuation is included in the flow solution, it is by several order of magnitudes lower than the pressure driving the turbulent flow. Capturing the acoustics in a flow solution requires accurate compressible simulation, including the propagation of acoustic waves travelling at the speed of sound.
2. Case study

2.1 Description

The application case illustrated in Figure 1 consists of a 0.6m diameter fan. The configuration includes the electric driving system and the support (0.4m height) considered as rigid in the present simulations. The fan is made of 7 blades. The blade thickness is equal to 5.5\,mm and is constant along its profile.

![Fan Diagram](image)

Figure 1: (a) Global description of the cooling fan (b) Blades repartition.

The different blades are not regularly spaced in the azimuthal direction as illustrated in Figure 1(b). For an observer located in a reference plane attached to the stator, this increases the period of the signal recording the blade passage, reducing the amplitude at the blade passing frequency (BPF). As some symmetrical patterns are observed, some sub-harmonics of the blade passing frequency are expected in the experimental signal.

2.2 Experimental measurements

The measurements are performed in an anechoic room at Technology Research Institute (Osaka, Japan). The rotation speed of the fan was forced to 2000 rpm (giving a rotation period equals to 0.03s), even if some rotation speed fluctuations are observed during the measurements. The sound pressure level was measured by a class 1 sound level meter (NL32) at 1m in front of the fan on the rotation axis, as illustrated in Figure 2. The overall background noise level in the anechoic chamber was limited to 15dBA.

![Experimental Setup](image)

Figure 2: Illustration of the experimental set-up.

The experimental signal is recorded during 39.6s and converted in the frequency domain for comparison of spectra.
To check the convergence of the experimental results, several Discrete Fourier Transform (DFT) over different time intervals are performed, as illustrated in Figure 3:

1. The total signal is subdivided in intervals overlapping by 50%;
2. Each signal interval is Fourier transformed;
3. The different frequency spectra are averaged.

\[ T_{\text{total}} = 0.0s \quad \text{0.3s} \quad \text{0.6s} \quad \text{0.9s} \quad \text{1.2s} \quad \text{...} \quad T_{\text{total}} = 39.6s \]

Figure 3: Procedure applied on the experimental time signal to convert it into frequency domain results.

In Figure 4, the experimental results are compared for different interval durations: 0.3s (10 fan rotations), 0.9s (30 fan rotations) and 1.8s (60 fan rotations). Longer intervals will lead to spectra with a smaller frequency step. Although the figure shows that the background noise level is relatively similar between all configurations, the peaks levels measured at the blade passing frequency changes. This observation should be recalled in the framework of simulation where the numerical signal is limited to short duration and large frequency step, leading to a possible change of the peak level and spreading of the tonal amplitude.

To be consistent with the numerical simulation regarding the duration of the CFD results (total length available = 0.45s), the experimental curve based on samples duration 0.3s is selected for comparison with numerical results.

![Figure 4: Evolution of the experimental sound pressure level with respect to the duration of the samples.](image_url)

3. Numerical details

Acoustics is generated by the unsteady phenomena related to turbulence. The acoustics and fluid dynamics signals are mixed in the source region. The acoustic part, although being very small compared to the aerodynamics one is very efficient and propagates over large distances without any major dissipation. The aeroacoustic sources region corresponds to the zone where the most active turbulent eddies are located. For fan noise application, the rotor generates a series of harmonics related to the
blade passages associated to rotation, but the blades wake also includes smaller turbulent structures responsible for a broadband part of the noise.

Several methods have been developed to tackle this problem, from analytical methods with fast resolution times to advanced and computationally expensive direct simulation where both fluid dynamics and acoustics perturbations are solved at the same time. In this hybrid method, the aeroacoustic sources are computed from a reliable unsteady CFD solution and then propagated in an acoustic dedicated solver. The input unsteady flow field needs to represent accurately the turbulent physics in order for the aeroacoustic sources to be accurate. This separation between the flow and acoustic parts of the problem saves the computational resources as each part method is optimized and aligned with the physics requirements:

- Amplitude: the fluid pressure fluctuations driving the flow are much larger than the acoustic pressure fluctuations. Capturing acoustics in the flow simulation therefore requires accurate and non-dissipative schemes.
- Propagation speed: The Mach number is the ratio between the fluid velocity and the speed of sound. The maximum propagation speed has an influence on the flow simulation time step.
- Length scale: the turbulent length scales are much shorter than the acoustic wavelength, leading to a very different spatial resolution for turbulence and acoustics modelling.

### 3.1 Theory

The decomposition of the problem where the preliminarily computed unsteady flow is processed to extract the noise sources in a second step is called the hybrid approach. It assumes that acoustics does not provide any feedback effect on the fluid dynamics. The hybrid approach has been proposed originally by Lighthill [1] but several scientists [2] propose similar techniques, differing by the equation solved or by the technique used to solve the mathematical equation.

For the present application, a Finite Element Method (FEM) solver is used to derive a frequency domain solution of the Lighthill equation (1) as detailed in [3]:

\[
\frac{\partial^2 \tilde{\rho}}{\partial t^2} - \frac{\partial}{\partial x_i} \left( \epsilon_0^2 \frac{\partial \tilde{\rho}}{\partial x_i} \right) = \frac{\partial^2 \tilde{T}_{ij}}{\partial x_i \partial x_j} \tag{1}
\]

where the Lighthill equation is solved for density fluctuations \( \tilde{\rho} \), providing that the Lighthill tensor \( \tilde{T}_{ij} \) is computed from the unsteady solution. The equation is solved in the frequency domain. The FEM formulation right hand side includes a volume contribution corresponding to the aeroacoustic sources present in the computational domain and a surface integral accounting for all sources present upstream and convected through a permeable boundary:

\[
\omega^2 \int_N \rho_N dV - \int_N \frac{\partial N_\alpha}{\partial x_i} \epsilon_0^2 \frac{\partial \rho}{\partial x_i} dV = i \omega \int_S N_\alpha \rho_i n_i dS + \int_V \frac{\partial N_\alpha}{\partial x_i} \tilde{T}_{ij} dV \tag{2}
\]

where:
- \( \omega \) is the angular frequency
- \( v_i \) is the \( i \)th component of the velocity
- \( \rho \) is the acoustic density fluctuation
- \( N_\alpha \) is the finite element shape function
- \( \tilde{T}_{ij} \) is the Lighthill tensor

The acoustic domain does not include the rotating blades and the surface contribution needs to be computed on the permeable surface enclosing the rotor. As CFD schemes are mainly designed to capture the turbulent field, they are in general dissipating the acoustic signal rapidly. To capture the signal in a region where the information is still accurate, techniques have been developed in the solver to read the information in the rotating part where the blades wake signals are accurate and reliable for acoustic propagation.
3.2 Simulation setup

The simulation setup has to include a rotating part and a static part connected through an interface. As acoustics deals with small fluctuations, it is mandatory to check the interface connection is as accurate as possible and does not introduce spurious oscillations affecting the quality of the turbulent solution and therefore generating a spurious noise source. For flow part, a finite volume technique solves the incompressible Navier-Stokes with large eddy simulation for the turbulence, while the acoustic solver is based on finite element techniques.

3.2.1 Fluid Dynamics

The CFD model built and run with Cradle SC/Tetra software is made of 24 million cells distributed into two regions: a rotating region around the fan (~11.5 M cells) and a static region (~12.5 M cells) located around the rotating region and the fan support. The mesh size is finer in the rotating region as well as in the static region closed to the rotor/stator interface. The mesh size increases as far as we get away from the rotor/stator interface.

The LES simulation is computed based on a WALE (Wall-Adapting Local Eddy-viscosity) model [4]. The accuracy of the discretization of the equations is ensured by using a second-order central difference scheme applied to the advection and diffusion terms. In SC/Tetra, an effect of a first-order upwind scheme is introduced into the advection term of the Navier-Stokes equation to ensure the stability of calculation [5]. The unsteady simulation is computed with a 1e-4s time step and the turbulent flow is exported each time step during 15 fan rotations (0.45s).

3.2.2 Acoustics

The acoustic model is presented in Figure 5. Starting from the geometry cleaned of any non-representative details as the ones used for the flow simulation, the acoustic mesh is designed to ensure non-dissipation of the acoustic waves in its propagation. In the acoustic simulation, the set-up does not include the rotating blades. For the far field, a surface enclosing the domain enforces the non-reflective condition while keeping the necessary information to compute the solution outside the domain. The turbulent information is collected and injected in the acoustic simulation on a permeable boundary surface enclosing the blades. This surface is defined by the user in the acoustic simulation set-up.

![Figure 5: Cut view of the acoustic mesh and model description.](image)

The struts and surfaces are considered as perfectly rigid for the acoustic propagation. Virtual microphones are recording the acoustic solution for comparison with experimental spectra. The solver
may provide additional acoustic information as the acoustic power radiated by the fan or acoustic maps.

The solver works in the frequency domain, using a mesh designed to model the propagation up to the maximal frequency accurately. In this case, the acoustic set-up involves 137 000 degrees of freedom for a maximal frequency of 1kHz. Mesh refinement would be required if the maximal frequency has to be increased, keeping the same number of finite elements per wavelength and guaranteeing low dispersive and dissipative errors.

The aero-acoustic sources are computed using the total unsteady flow solution available (0.45s). The sources are arranged in two overlapping time intervals each one corresponding to 10 fan rotations (0.3s). As for the experimental results, an overlap of 50% is considered. A Discrete Fourier Transform converts the samples to the frequency domain. In the present case, the DFT has been limited to frequency range requested by the user (1kHz). The signal duration directly influences the frequency step and the minimal frequency (3.333Hz). The acoustic simulation solves the systems considering the aero-acoustic sources for each frequency and each sample separately. The results are averaged over the two samples corresponding to two DFT of the time intervals.

### 3.3 Setup details

The flow solution provides the necessary input for acoustic simulation. The turbulence model is based on Large Eddy Simulation techniques: a model induces the effects of non-resolved turbulent scales on the resolved structures. Small turbulent structures correspond to high frequencies while resolved structures should at least correspond to the maximal frequency requested for aeroacoustic simulation. On the CFD mesh, the maximal frequency corresponding to the resolved scale is mapped to better understand the noise source pattern and check if this frequency is aligned with aeroacoustic perspective of the simulation. The Figure 6 shows the local maximal frequency solved on the CFD mesh in a cut-plane including the fan axis. Close to the blades, the turbulences are very active and induce very high frequency fluctuations. On the rotor/stator interface, high frequency fluctuations are noticed which are spurious numerical oscillations induced by the mesh connection between rotor and stator.

To avoid considering these spurious fluctuations, two Control Surfaces have been designed, collecting the information outside the region affected by the rotor/stator interface. The Control Surfaces are represented on the Figure 6 and collect the information in the rotating region of the CFD simulation.

Figure 6: Map of the characteristic frequency (cut view) and Control Surfaces definition.
The maximal frequency presented in Figure 6 is estimated [6] based on a RANS solution computed on the same mesh as the one for the LES simulation. This RANS solution is used to initialize the flow field in the LES simulation.

4. Analysis

The numerical and experimental results are compared on the virtual microphone recording the acoustic signal 1m in front of the rotor. As detailed in section 2.2, tonal components are expected at the blade passing frequency but also to multiple of rotation frequency due to non-regular azimuthal spacing between the blades. High order azimuthal modes also generate tonal contributions at the multiple of the blade passing frequency.

![Figure 7: Acoustic pressure level on an axial microphone located 1m in front of the fan.](image)

The numerical and experimental results are presented in Figure 7. For the experimental signal, a large series of DFT are computed over different time intervals. The average of these Fourier Transforms is presented with the minimal and maximal amplitudes for each frequency, defining an experimental envelope of all turbulent realisations. For tonal contributions, the envelope width is smaller than for the other frequencies. This is mainly justified by the fact that tonal components are related to regular events as the blade passage, while the remaining part of the spectrum has a larger envelope due to the turbulent realisations, converging statistically but not reproduced at each rotation exactly. In the experiments, we notice the presence of tonal components at the blade passing frequency and its (sub-) harmonics as explained in section 2.2.

Numerical solution corresponds to the one excited by the information collected on the Control Surface 1 which is the closest to the blades. The blade passing frequency emerges in the numerical spectrum, although its amplitude is slightly bigger than the experimental level. The sub-harmonics are not detected and seem hidden by the broadband level which is 5-10dBA higher than the experimental one in the frequency range 0-500Hz. This could be due to the turbulence intensity injected in the numerical flow simulation to a higher level than the experimental value.

Above 500Hz, the numerical results has a higher decay rate than the experimental results. This is currently investigated by checking the resolution of the CFD mesh and check if this part of the spectrum could be improved by a finer mesh resolving better the turbulence in LES simulation.

The convergence of the acoustic simulation is assessed by checking the difference between acoustic results using the input data collected on Control Surface 1 (closest to the blade) and Control Surface 2 (located further downstream, see Figure 6). The comparison presented in Figure 8 does not show significant difference illustrating that the acoustic results are converged.
5. Conclusions

In this paper, we present the numerical analysis of the noise emitted by an axial fan dedicated to the engine cooling. The noise simulation is based on a hybrid method where the noise sources are computed from unsteady flow fields. The noise sources are collected on Control Surfaces; the sources accounting for all perturbations generated upstream the surface. The full acoustic phenomena are generated in the volume and propagated by the flow simulation tool up to the Control Surface.

The method is presented theoretically as well as the numerical parameters involved in this configuration. The fan generates tonal components at multiple of the rotation frequency. The blade passing frequency (number of blades x rotation frequency) is visible, but other sub-harmonics appear due to the non-regular azimuthal positions of the blades along the rotor. The broadband noise level in the numerical results is overestimated compared to the experimental level. This is probably related to a difference in the turbulence in the simulation. To check this hypothesis, further investigations are possible to assess the influence of the turbulence level injected in the simulation inlet section or the CFD mesh refinement which plays a significant role in LES simulations. The present paper illustrates the current level of simulation accuracy. Results and conclusions will be updated once these further investigations are closed.

REFERENCES