

Proper Orthogonal Decomposition Analysis of Flow Structures Generated around Engine Cooling Fan

Yuji Kobayashi
Tokyo City Univ.

Kenji Yoshida
DENSO Corp.

Hideaki Nagano and Itsuhei Kohri
Tokyo City Univ.

ABSTRACT

A cooling fan is one of the primary components affecting the cooling performance of an engine cooling system. In recent years, with the increase in electric vehicles (EVs) and hybrid vehicles (HVs), the cooling performance and noise level of the cooling fan have become very important. Thus, the development of a low-noise fan with the same cooling performance is urgently required. To address this issue, it is critical to find the relation between the performance of the fan and the flow structures generated around it, which is discussed in the present paper.

Specifically, a computational method is employed that uses unsteady Reynolds-averaged Navier-Stokes (URANS) coupling with a sliding mesh (SLM). Measurements of the P-Q (Pressure gain-Flow rate) characteristics are performed to validate the predictive accuracy of the simulation. Additionally, proper orthogonal decomposition (POD) is employed to analyze the unsteady data obtained by URANS calculations to reveal the typical flow structures. This technique makes it possible to classify the phenomena into several typical modes, which help to clarify the phenomena analytically. As a result, the impacts of the dominant flow structures on the fan performance are clarified quantitatively by calculating the contribution ratio obtained by the POD analysis.

CITATION: Kobayashi, Y., Yoshida, K., Nagano, H., and Kohri, I., "Proper Orthogonal Decomposition Analysis of Flow Structures Generated around Engine Cooling Fan," *SAE Int. J. Passeng. Cars - Mech. Syst.* 7(2):2014, doi:10.4271/2014-01-0667.

INTRODUCTION

In recent years, the noise generated by the cooling fan of an electric vehicle (EV) has been relatively loud because the noise level of its motor has been lower than that produced by the engine of a conventional vehicle. Thus, the development of a low-noise fan with the same cooling performance is urgently required. To meet this goal, it is very important to find the relation between the performance of a fan and the flow structures generated around it. CFD technology is commonly used for the prediction of fan performance [1] [2]. However, it does not always have good accuracy. Thus, in the previous study, we focused on the multiple reference frame (MRF) method, which is widely employed to predict the performance of a fan, and the fundamental idea of the applicable limitation of MRF was proposed [3] [4]. However, a countermeasure for correct calculation has not yet been suggested.

Thus, as a continuation of the study reported in the previous paper, we focus on the SLM method in the present paper. The generation of an unreasonable strain-rate on the MRF boundary section, which is a serious problem with MRF calculations, is reduced by using SLM. Additionally, SLM also serves as a countermeasure for the correct calculations.

After confirming the predictive accuracy using the experimental measurement of the P-Q characteristic, this paper focuses on POD, which is commonly used to reduce computational loads [5] [6] [7]. Although applying POD to a flow analysis has not been extensively discussed in the past, it is thought to have the potential to assist in understanding the relation between the fan performance and the flow structures around the blade quantitatively.

To clarify the above issues, this paper conducts three types of investigations. First, from the view point of the calculation loads,

POD is used for the (two-dimensional) 2D cylinder calculation. Secondary, the technique is used for the pressure fluctuation on the suction surface of a fan blade obtained using (three-dimensional) 3D fan calculations. Finally, it is applied to the fluctuation of the vortex magnitude in all the computational grids. From these investigations, the possibilities of using POD for fan calculations are described and the dominant flow structures and their relation to the fan performance are clarified. This paper surveys these issues.

METHODOLOGY

Experimental Method

The fan used in the present study has nine uneven pitch blades, as shown in Fig.1. It is a half scale model of an actual engine cooling fan, which has a diameter of 250 mm and a width of 35 mm. The fan performance is examined by measuring the flow rate and pressure gain at 3000 rpm in a test chamber. Regarding this system, the experimental uncertainty of the measurement of the fan performance has been evaluated to be within $\pm 1\%$.

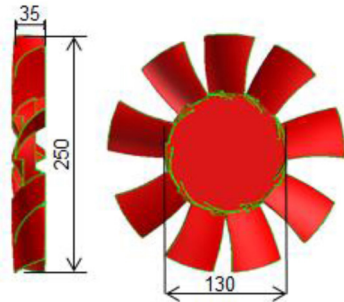


Fig.1. Test piece used in present study

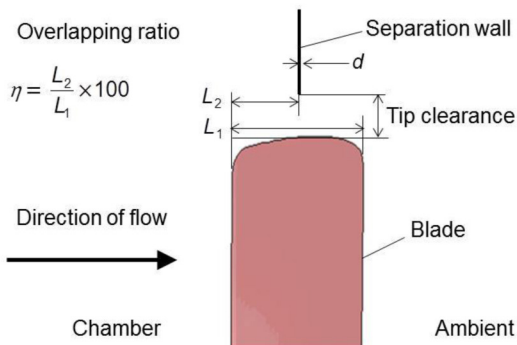


Fig.2. Relation between blades and separation wall

The fan performance is evaluated using the φ - ψ^* characteristics, where φ and ψ^* are the coefficients of the flow rate and pressure gain, respectively, as defined by the following formulas:

$$\varphi = \frac{Q}{u_t \cdot A} \quad (1)$$

$$\psi^* = \frac{2\Delta P}{\rho \cdot u_t^2} \quad (2)$$

where Q , A , u_t , ρ , and ΔP denote the flow rate, frontal area of the rotating region of the fan, tip velocity, air density, and pressure difference between the inside and the outside of the chamber, respectively.

The geometrical relationship between the blades and the separation wall is shown in Fig.2, i.e. the tip clearance is 10 mm, the thickness of the separation wall is 1 mm, and the overlapping ratio is defined as follows:

$$\eta = \frac{L_2}{L_1} \times 100 \quad (3)$$

Computational Method of Fan Calculation

Table 1 presents a summary of the computational method used in this paper for fan calculations. The finite volume method is adopted to solve the unsteady Reynolds-averaged Navier-Stokes (URANS) model coupled with the RNG k - ε model. The Semi-implicit (SIMPLE) method is employed for the solution of the velocity-pressure field. The second-order upwind differential scheme for the convection term, the standard wall function for the wall boundary layer treatment, and the sliding mesh (SLM) method for the space around the rotating blade are also used. The calculation code of ANSYS FLUENT 12.1 is employed in the present study. The computational model is shown in Fig.3, which imitates an experimental system. Grid structures are used, just as in the previous study [3]. Tetrahedral unstructured grids are adopted for the entire computational domain, and a uniform size of 2 mm is generated for the grid in the rotating region of Case1, which will be described later in detail. The identical grid structures are applied to Case2 and Case3, with the only changes occurring in the rotating-boundary section. As a result, the total number of elements is on the order of 13 million.

Table 1. Specification for fan calculations

-	SLM Method	MRF Method
Governing Equation	RANS Equations	RANS Equations
	Continuity Equation	Continuity Equation
Fluid	Incompressible	Incompressible
State	Unsteady	Steady
Turbulence Model	RNG k - ε model	RNG k - ε model
Boundary Layer	Std. Wall Function	Std. Wall Function
Treatment of Rotation	SLM	MRF
Solution of u and p	SIMPLE	SIMPLE
Scheme	2nd Order Up-Wind	2nd Order Up-Wind
Cell Structure	All Tetrahedral	All Tetrahedral
Total Number of Cells	13.0 Million	13.0 Million
Rotating Region	Case1,2,3	Case1,2,3
Time Step [s]	1.1×10^{-4} (2deg/step)	-
Unsteady term	2nd order Implicit	-

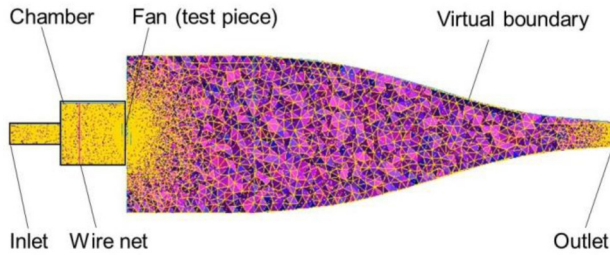


Fig.3. Computational grid used in present study

MRF/Rotating Region Used in Present Study

In the present study, three types of MRF/Rotating-regions are employed, which have conditions identical to those in the previous study [3], as shown in Fig.4. The region used in Case1 is similar to that by Kohri. et al. [8]. As for Case2, the border of the region is defined at the location where the radiator surface exists for an actual vehicle. In Case3, the boundary surface is set to surround the fan, close to the blades. The Table 2 shows concrete size of MRF region used in this paper. Here, γ represents the dimensionless size of MRF-region defined by following formulas;

$$\gamma_u = \frac{L_u}{D} \quad (4)$$

$$\gamma_d = \frac{L_d}{D} \quad (5)$$

where, L_u , L_d and D denote the distance between the central position of the blade and the up-stream of MRF-boundary section, the distance between the one and the down-stream of MRF-boundary section and the diameter of the fan.

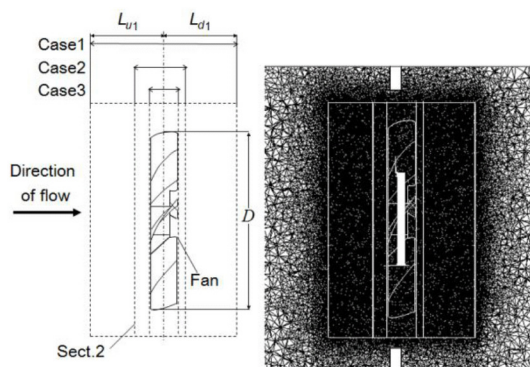


Fig.4. Three types of rotating regions used in present study

Table 2. Size of MRF/Rotating region

Case	Case1	Case2	Case3
γ_u	0.40	0.16	0.08
γ_d	0.40	0.12	0.08

FAN PERFORMANCE

Fig.5 shows the $\phi - \psi^*$ performances obtained by the calculation, compared with the experimental results. It is found that, for Case1 and Case2, the SLM calculations accurately predict the experimental results for all the flow rate regions. Regarding the SLM calculations for Case3, the value of ψ^* at $\phi = 0.02$ is slightly lower than that found in the experiment. However, the precision of the other points are in good agreement with the experimental values. On the other hand, for the condition of $\phi = 0.17$, the SLM prediction agrees well with the results of the experiment, whereas the MRF prediction indicates higher values than the experimental results. Additionally, the SLM results for Case1 and Case2 show identical values in all the flow rate regions, whereas the MRF results are not the same. Based on the previous studies [3] [4], we next focus on the strain-rate generated on the rotating boundary surface to understand why the predictive accuracy is improved by employing the SLM method.

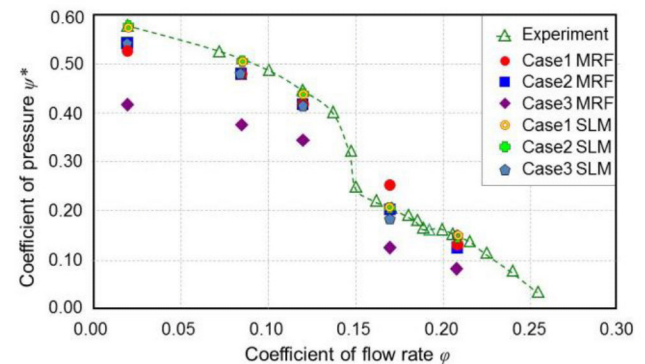


Fig.5. Validation of predictive accuracy obtained from study of fan calculations

INFLUENCE OF DIFFERENCE BETWEEN MRF AND SLM ON PREDICTIVE RESULTS

Discussion of Computational Error on Boundary Surface

In the previous study [3], the generating mechanism of an unreasonable strain-rate that appeared on the boundary surface between the stationary and rotating domains was clarified when the MRF method was adopted. Furthermore, in the case of a condition where $\phi = 0.17$, i.e. the condition that is sensitive to a disturbance in the upstream region, it was also found that the unreasonable strain-rate had an impact on the prediction of the fan performance. This means that, even though the SLM method is used, an unreasonable strain-rate could be generated on the boundary surface. Thus, this paper also focuses on the condition where $\phi = 0.17$ of Case2 and compares the strain-rates generated on the boundary surface when using the SLM and MRF methods.

Fig.6 shows the distribution of the strain-rate calculated on the boundary surface at $\phi = 0.17$. It can be seen that no unreasonable strain-rate is generated on the surface in the SLM case. The

SLM method actually rotates the grids to imitate the revolution of the cooling fan, and linear interpolation is performed for the boundary surface through the interface, as shown in Fig.7. In other words, SLM does not require a coordinate transformation to the rotating frame from the absolute coordinate system when the first grid point inside the rotating domain is calculated. We consider this point to be the main difference between MRF and SLM and the consequent reduction in the generation of the unreasonable strain-rate. For this reason, when using SLM for Case2, the flow field upstream of the fan is also reasonably calculated. On the other hand, when $\varphi = 0.02$, the strain-rate is suppressed as expected, as shown in Fig.8. However, the strain-rate appears on the surface in spots, whereas there is no strain-rate for the condition at $\varphi = 0.17$ as shown in Fig.6. This is thought to be the result of the complex flow structure induced by the strong pre-rotation [3]. Therefore, it is clarified that the unreasonable strain-rate cannot be suppressed completely at the boundary surface where the velocity is remarkably uneven as a result of the strong pre-rotation and/or complex flow structure, even if SLM is used. In the previous study [4], we tried to reduce the unrealistic strain-rate by changing the grid resolution and grid type at the boundary section. In case of grid resolution, the generation of the strain-rate could not be reduced and the predictive accuracy of the fan performance was not improved. On the other hand, in case of grid type, the unrealistic strain-rate was suppressed but the predictive accuracy was not improved than we thought. Therefore, we next focus on the relation between vortex structures and predictive accuracy to clarify the above question.

Relation Between Vortex Structures and Predictive Accuracy

In this section, the relations between the vortex structures, strain-rate generated boundary surface, and predictive accuracy of the fan performance are discussed. Fig.9 and Fig.10 show the vortex structures presented by the three types of iso-surfaces for the vorticity magnitude at $\varphi = 0.02$ and $\varphi = 0.17$. Regarding $\varphi = 0.02$, it is found that SLM produces larger tip and separation vortices than MRF, which are thought to correspond to the prediction of higher ψ^* values. This means, when calculating with SLM, a larger separation is predicted because of the transitional fluctuation induced by the shaking of the vortex structures around the blade. From this consideration, it is deemed that SLM gives accurate predictions at a low flow rate condition as a result of the calculation of the fluctuation of the vortices, even if the grid density of the space is coarse. Thus, it is necessary to calculate the increase in the detachment area to accurately predict the fan performance under a low flow rate condition for the fan used in the present study. In other words, if we predict the performance using MRF, a very small grid size is needed to compute the shear generated by the dissipation of vortices, such as in the case of

Kohri et al. [8]. As a result, it is necessary to use a higher resolution grid density than with SLM. In the case of middle and high flow rate conditions, where the vortices disappear and/or become small, the fan performance can be accurately predicted by using the MRF method.

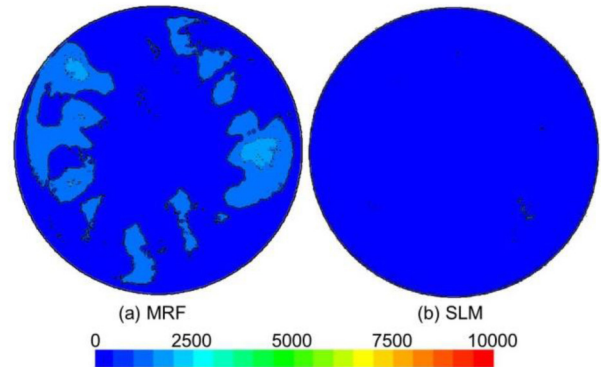


Fig.6. Comparison of strain-rates on Sect.2 as defined in Fig.4 between MRF and SLM (Case2, $\varphi = 0.17$)

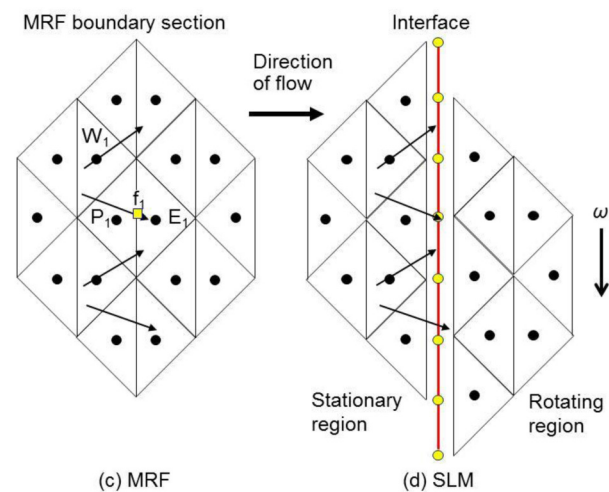


Fig.7. Difference in calculation methods at interface section

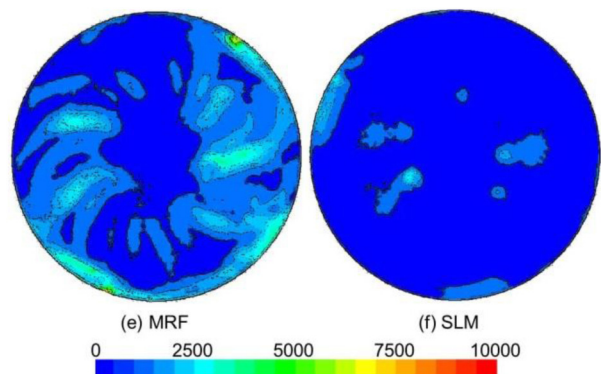


Fig.8. Comparison of strain-rates on Sect.2 as defined in Fig.4 between MRF and SLM (Case2, $\varphi = 0.02$)