

Prediction of the Performance of the Engine Cooling Fan with CFD Simulation

2010-01-0548

Published
04/12/2010

Itsuhei Kohri and Yuji Kobayashi
Tokyo City Univ.

Yukio Matsushima
Mitsubishi FUSO Truck & Bus Corp.

Copyright © 2010 SAE International

ABSTRACT

The technology concerning thermo and fluid dynamics is one of the important fields which have made great progress along with rapid advance in computational resources. Especially, the CFD technology has been proved as successful contribution to the development of the engine cooling system. Therefore, this technology is widely used at early phase of the vehicle development. However, a serious problem has been remained that it does not always give practical precision. Particularly, the cooling fan is one of the primary components in the cooling system to determine the performance, while practical calculation method without depending on large resources has not established.

INTRODUCTION

Recently, computational design of vehicles has become popular to meet the requirement of short-time development. The technology concerning thermo and fluid dynamics is one of the significant fields which have made great progress along with rapid advance in computational resources. Especially, the CFD technology has been proved as successful contribution to the development of the engine cooling system. Therefore, this technology is widely used at early phase of the vehicle development [1][2][3]. However, a serious problem that it does not always give practical precision has been remained. Particularly, the cooling fan is one of the primary components in the cooling system to determine the performance, but practical calculation method without depending on large resources has not established.

The present work is aiming to develop new method which gives practical precision for calculation of the fan

performance with realistic capacity of the computational resources. Then, authors took up a particular cooling fan for small size trucks because its predicted precision was not practical.

To improve the predictive precision, experimental analysis of the flow field around the blade is inevitable. Therefore, experimental technique for rotating blade is required. Then, at first, the flow visualization technique and the pressure distribution measurement technique for rotating blade is developed.

The second challenge of present study is introduction of new idea of “typical vortex structure” around the blade. This idea plays an important role when the relation between the flow field and the fan performance has not been clarified. In present study, the precision of the calculation is evaluated by the behavior of the vortex structure because the vortex structure dominates the flow pattern around the blade. Here, most important issue is to detect the most effective vortex structure. The difference between the calculation and the experiment mainly owes to the difference in the behavior of the vortex structure.

As above mentioned, the major work of this research is development of new method. Generally, the procedure of the improvement of the predictive precision is categorized into five issues, i.e. (i)theoretical equation, (ii)models such as turbulence model, boundary layer model, etc. (iii)discretization method, (iv)calculation scheme, (v)grids structure. However, since present study is conducted by using a commercial code, (ii) and (v) is adopted while (i),(iii) and (iv) are hard to be handled.

Firstly, the effects of the turbulence model are examined. Then the idea for the grid generation is investigated. As a result, the essential treatment for the improvement is found. Furthermore the mechanism which contributes to the improvement is discussed. Finally, new method proposed in present paper is validated to be practical for the realistic condition.

EXPERIMENTAL AND CALCULATION CONDI-TION

EXPERIMENTAL FEATURES AND TEST PIECE

The fan used in present study has eight swept-back and tip-bended type blades shown in Fig. 1. It is the 50% model of the engine cooling fan adopted in small trucks, 250mm of the diameter and 30 mm of the width. The performance is studied by measuring flow rate and pressure gain at 3000 r.p.m. with the test chamber as shown in Fig. 2. Regarding this facility, the repeatability of data varies within 1% of the pressure vs. flow-rate performance for ten times measurements. The flow rate and pressure measurement have been also validated initially. The average of 10000 samples of each output is taken for each measurement. The fan is installed at the center of the circular hole on the separation wall of the chamber and driven by the electric motor. For the initial study, no shroud is equipped.

The performance of the fan is evaluated by $\phi - \phi^*$ characteristics. Here ϕ and ϕ^* are the flow rate coefficient and the pressure gain coefficient defined by following formulas respectively,

$$\phi = \frac{Q}{A \cdot U_t}, \quad (1)$$

$$\phi^* = \frac{\Delta p}{\frac{1}{2} \rho U_t^2}, \quad (2)$$

where Q , A , U_t , ρ and Δp denote the flow rate, the frontal area of the rotating region of the fan, the tip velocity, the air density and the pressure difference between inside and out side of the chamber. The geometrical relationship between the blade and the separation wall is represented in Fig. 3, as the tip clearance $t=10\text{mm}$, the thickness of the separation wall $d=1\text{mm}$ and the over-wrapping ratio $\eta=0.5$, where η is defined by following formula,

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

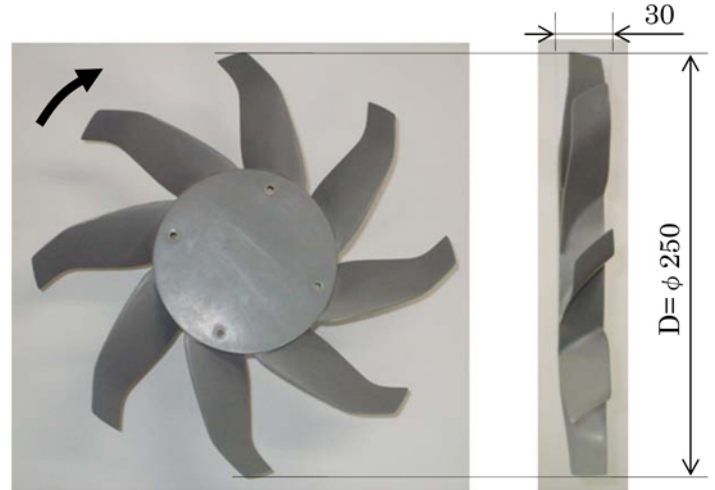


Fig.1. Test piece (1/2 scale model)

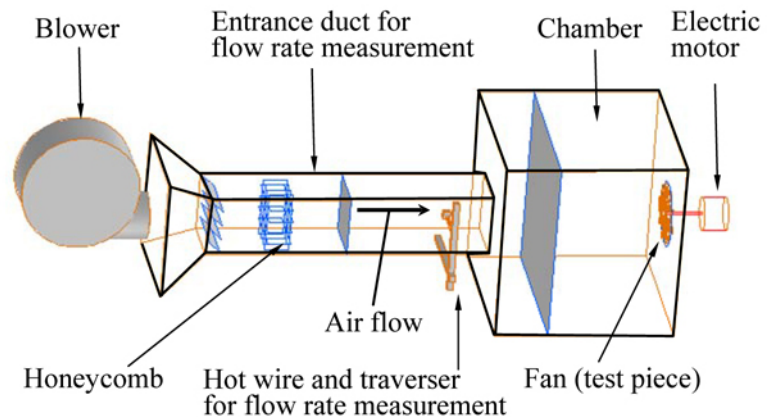


Fig. 2. Experimental features

FLOW VISUALIZATION ON THE BLADE SURFACE

The flow visualization around blades is performed with tufts posted on the surface of the blades. We take a picture of the behavior of the tufts by synchronizing the flash of the strobe scope with the rotation. Fig. 4 shows the set-up for the flow visualization.

The effects of the centrifugal force acting on the tufts are taken into following consideration [4]. The direction of the tuft in the flow is basically determined by the aero- dynamic force F_A acting on the tuft. However, in such the case of rotating blades, the direction is different from the flow

direction due to the centrifugal force F_C . When those forces are in the state of equilibrium, the state of the tuft becomes as shown in Fig. 5 and their normal components to the tuft acting on the small element dl , i.e. dF_{An} [5] and dF_{Cn} , are denoted as following formulas respectively,

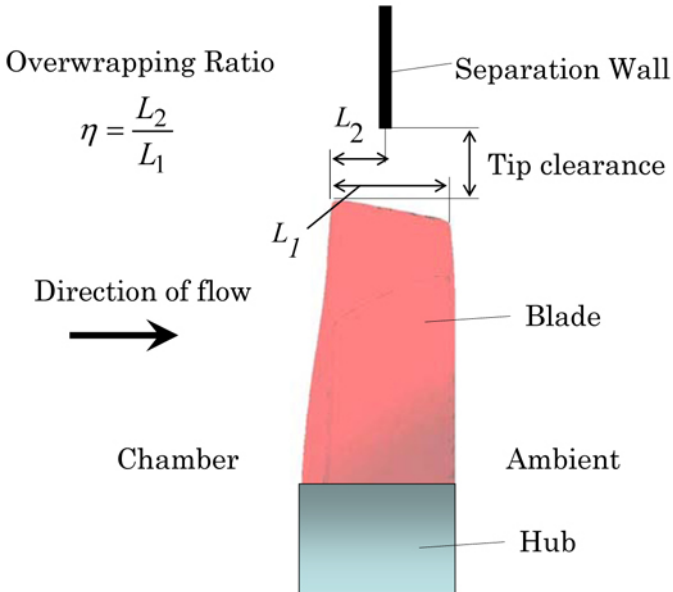


Fig. 3. Blade and separation wall

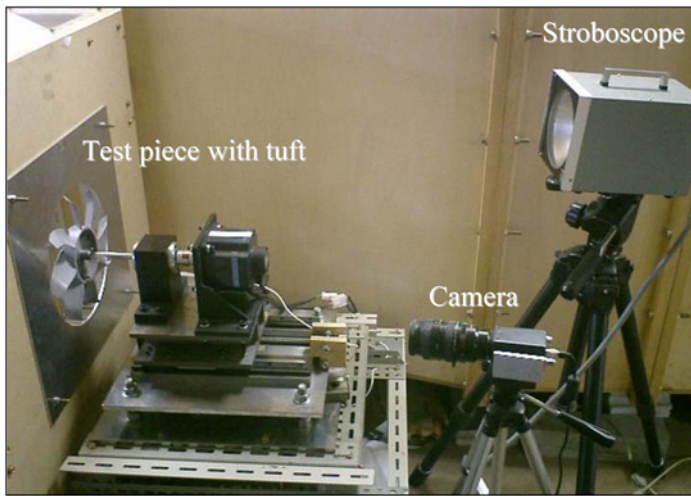


Fig. 4. Set-up for flow visualization

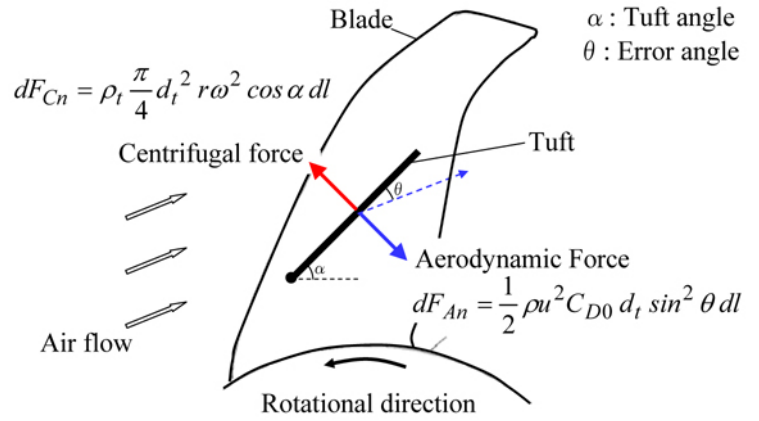


Fig. 5. Equilibrium of forces acting on the tuft

$$dF_{An} = \frac{1}{2} \rho \cdot u^2 \cdot C_{D0} \cdot d_t \cdot \sin^2 \theta \cdot dl \quad (4)$$

$$dF_{Cn} = \rho_t \cdot \frac{\pi}{4} \cdot d_t^2 \cdot r \cdot \omega^2 \cdot \cos \alpha \cdot dl \quad (5)$$

where, ρ , ρ_t , d_t , r , ω , C_{D0} , u , α and θ indicate the density of the air, the density of the tuft, the diameter of the tuft, the radial location of the tuft, the angular velocity, the drag coefficient of the tuft at $\theta = \pi/2$, the relative velocity to the blade, the angle of the tuft to the tangential direction and the angular difference between the direction of the tuft and that of the air flow respectively. Then, supposing the equilibrium of those forces as $dF_{An} = dF_{Cn}$, θ is given by following equation,

$$\theta = \arcsin(\sqrt{\xi \cdot \cos \alpha}), \quad (6)$$

when ξ , namely centrifugal factor, is defined as follows,

$$\xi = \frac{\pi \rho_t \cdot d_t \cdot r \cdot \omega^2}{2 \rho \cdot u^2 \cdot C_{D0}}. \quad (7)$$

Fig. 6 shows relationship between the centrifugal factor ξ and the error angle θ at $\alpha=0$. It is realized that the visualization is practical, i.e. reliable when ξ is smaller than 0.01 to analyze flow structure close to the blade. Then, the material and the size of the tuft are determined to satisfy the threshold of ξ . The results of the visualization in present

paper are evaluated with the effects of the centrifugal force taken account [4].

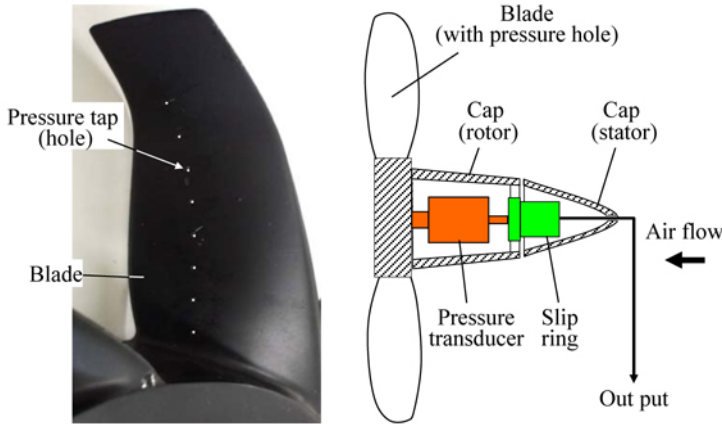
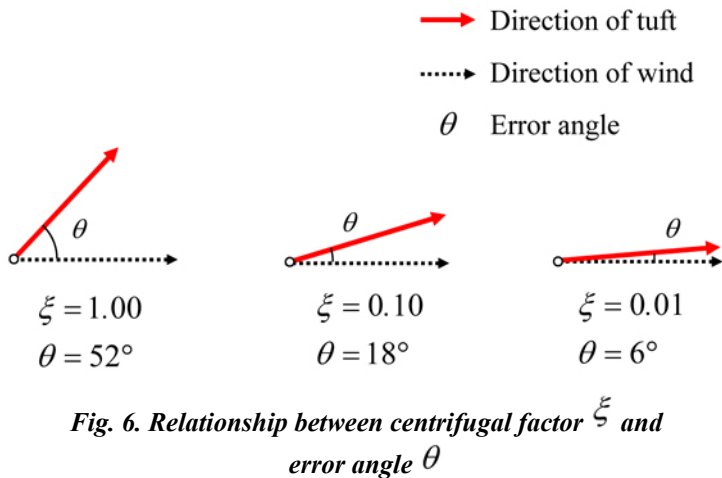


Fig. 7. Pressure measurement on the blade

PRESSURE MEASUREMENT ON THE BLADE

The pressure distribution on the blade in the state of rotation is measured with the blade designed as shown in Fig.7 [6]. The test piece is made with stereo-lithography technology. A spindle shaped cap is equipped on the front hub of the fan, and a pressure transducer and a slip-ring are installed inside it. It has been assured that there are no effects of the cap on the fan performance. The effects of centrifugal force have been corrected.

CALCULATION METHOD WITH CFD

At initial phase of present study, we adopted RANS model coupled with Standard $k-\epsilon$ model which is discretized by finite volume method. SIMPLE method was employed for solution to the velocity-pressure field, second order upwind differential scheme for the convection term, the standard wall

function for the wall BL treatment and MRF method to the space around the rotating blade. The calculation code of FLUENT is employed in present study. Major specification of the calculation is listed in the Table 1.

Fig. 8 shows structure of the calculation grids. The flow field is imitated with tetrahedral unstructured grids, and the grid size near the fan is determined carefully in consideration that accuracy of the flow structure close to the blade surface is sensitive to the precision of the fan characteristics. In present study, 1.5mm in minimum edge length is adopted. As a result, the total number of the elements became order of five million.

Table 1. Specification of CFD calculation

Governing Equation	RANS equations
Fluid	Continuity equation
Sate	Incompressible
Turbulence Model	Steady State
Boundary Layer	Std. $k-\epsilon$ model
Treatment of Rotation	Std. Wall Function
Discretization	MRF
Solution of u and p	Finite Volume Method
Scheme	SIMPLE
Cell Structure	2 nd order up-wind
	All Tetrahedral

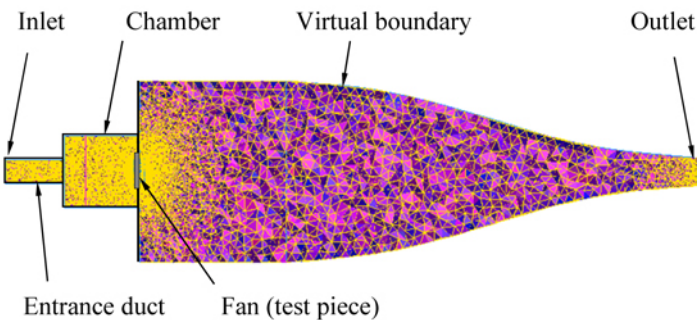


Fig. 8. Calculation grid system

DETERMINATION OF TYPICAL FLOW STRUCTURES RELATED TO PREDICTIVE PRECISION

FAN PERFORMANCE AT INITIAL PHASE OF STUDY

At the beginning of present study, we compare the calculated results obtained by orthodoxy methods listed in the Table 1

with the experimental ones. Fig.9 shows those results. The predicted pressure φ^* indicates constant values lower than the measured ones at low flow rate region where ϕ is lower than 0.12, while φ^* decreases with increase in ϕ by experiment. However, at the region between $\phi=0.12$ and 0.15, both φ^* agree well with each other. On the contrary, at the middle flow rate region between $\phi=0.15$ and 0.18, the predicted φ^* changes gradually with increase in ϕ whereas the measured φ^* decreases rapidly. It is known that the rapid change in φ^* is related to the rapid change in the flow pattern around the blades of the fan. Concretely, centrifugal flow pattern appears at low flow rate region and axial flow pattern at high flow rate region [7]. Furthermore, the predicted φ^* indicates higher value than the measured one at high flow rate region where ϕ is larger than 0.18, and changes parallel to the experiment.

Consequently, it is realized that the calculation does not always predict realistic flow phenomenon as far as basing on the orthodoxy methods. Then the analysis of the difference in the flow structure between both results is conducted.

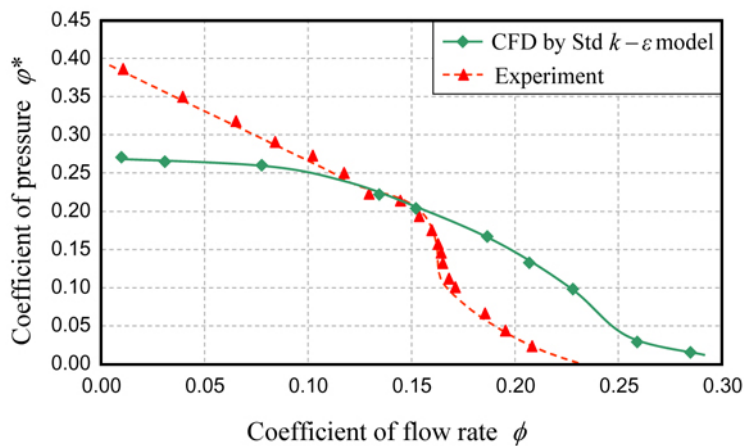


Fig. 9. Fan performance calculated by orthodoxy method

FLOW ANALYSIS WITH VORTEX STRUCTURE

Since it is difficult to grasp the detailed flow around rotating blade experimentally, some contrivances are required. Then we attract typical vortex structures which detach from the blade and develop to the space between blades. Actually, since the main stream passes over those vortex structures, the fan performance is dominated by the nature of them.

As the predicted φ^* agrees well with measured one at the flow rate of $\phi = 0.12$, it is considered that the flow phenomenon around the blade also agree with each other. Fig.10 shows the comparison of the flow direction between the flow visualization with tufts and the calculation of the velocity vector on the blade surface. It is confirmed that calculated flow pattern is approximately identical to the experimental one.

Fig. 11 represents a schematic of the typical vortex structure resulted from the analysis of the calculation. It is realized that three vortex structures, i.e. Vortex-A, Vortex-B and Vortex-C, are generated [8, 9]. Vortex-A is detached from the leading edge toward the tip on the suction side of the blade. Vortex-B is generated at a leading edge near the root of the blade on the pressure side. Vortex-C is generated between root of the blade and lateral side of the hub on the pressure side.

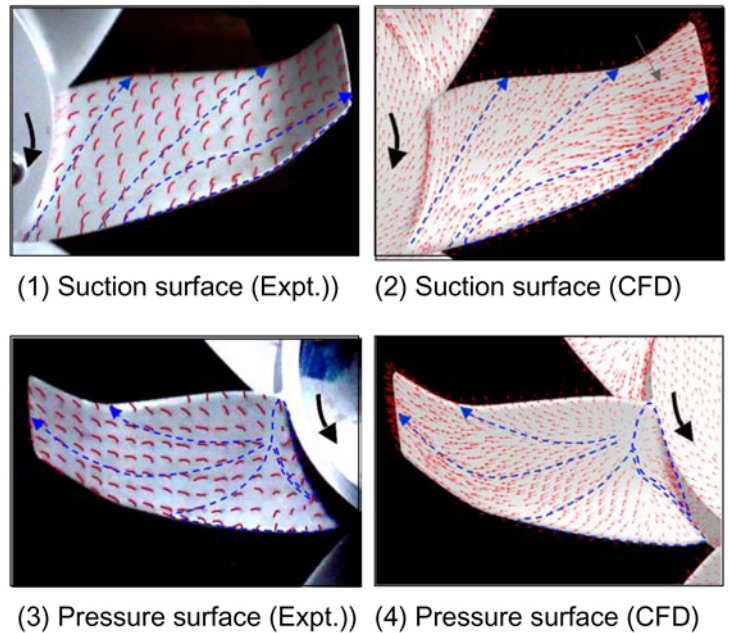


Fig. 10. Comparison of flow direction at $\phi=0.12$