Proceedings of HT-FED04
2004 ASME Heat Transfer/Fluids Engineering Summer Conference
July 11-15, 2004, Charlotte, North Carolina USA

HT-FED04-56288

COMPUTATIONAL AND EXPERIMENTAL STUDY OF EFFECTS OF GUIDE VANES AND TIP CLEARANCES ON PERFORMANCES OF AXIAL FLOW FANS

Sungsu Lee/Chungbuk National Univ., Korea
Dept. of Structural Systems & CAE
Gaesin Dong, Cheongju, Rep. of Korea
e-mail: joshua@chungbuk.ac.kr

Hak-Sun Kim/ Chungbuk National Univ., Korea
Dept. of Structural Systems & CAE
Gaesin Dong, Cheongju, Rep. of Korea
e-mail: learst@bcline.com

Kwang-Hyun Nam/ Chungbuk National Univ.
Dept. of Structural Systems & CAE
Gaesin Dong, Cheongju, Rep. of Korea
e-mail: bakcasd@nate.com

Jae Ik Hong/ Chungbuk National Univ., Korea
Dept. of Structural Systems & CAE
Gaesin Dong, Cheongju, Rep. of Korea
e-mail: obabywow@hotmail.com

Seung Hyun Chun/Hipres-Korea, Co., Ltd., Korea
Jinrye Myun, Kimhae, Rep. of Korea
e-mail: general@hipres.co.kr

ABSTRACT

The effects of guide vanes and tip clearances on the characteristics of axial flow fans are investigated both computationally and experimentally. Performance test of fans carried out in full scale shows considerable effects of tip clearance between rotor tip and duct on the characteristics of fans. The tested results are compared with the computation based on the finite volume method to solve the Navier-Stoke equations with $k-\varepsilon$ turbulence model. The comparison shows good agreements between experimental and computational results. In addition, the effects of shape of guide vanes are numerically studied. The results shows that increased volume of separated region around the guide vane reduces the recovery of tangential component of kinetic energy in the wake, resulting in loss of efficiency

Keywords: Guide Vane, Tip Clearance, Axial Flow Fan

INTRODUCTION

Until recently, axial flow fans were designed using old-fashioned methods which typically involve manual calculations based on 2D airfoil design strategy. It has been well known to the fan industry that the methods requiring many simplifying assumptions limit accuracy as well as availability in practice. These results in the practical designs are mostly depending on trial and error in manufacturing and testing many prototypes. In addition, a prototype with parameters satisfying one working condition, so-called design point does not guarantee the performance at off-design condition, which is very hard to estimate in advance.

Recently, engineers seeking tools for both analysis and design have utilize CFD (computational fluid dynamics), which provides detailed flow field as well as the estimation of performance of fans [1-4]. Although current state of art in CFD application to axial flow fans provides for analysis tool rather than an actual design tool, capability in changing various parameters enables the engineers to make in-direct assessment and modification. This methodology changes the entire strategy of production and development of axial flow fans, in such manners that markets with new requirement are actively developed and modification to existing products is readily tested before building any prototypes.

Using CFD for such purposes, a CFD tool of interest must be numerically tested and validated. This paper presents the computations for flow field around axial flow fans using well-known commercial code, FLUENT. Fully 3D Navier-Stokes equations are solved in finite volume approximations with 2 equations $k-\varepsilon$ turbulence model. Steady flow field is assumed while the interaction between rotor and guide vane (or stator) in the downstream is accommodated by so-called mixing plane model which iteratively considers the effects of different reference frames in averaged sense. The investigations of the effects of guide vanes in the rotor downstream and the tip clearance between rotor tip and inner surface of duct are highlighted. The computational results are compared with those of full-scaled experiments.

COMPUTATIONAL MODEL

A schematic view of the axial flow fan consisting a rotor of 10 blades and 19 guide vanes is shown in Fig. 1. Corresponding configuration of the computational model is shown in Fig. 2. The rotor is operating at 0.5*D* downstream of duct inlet, where *D* is the diameter of the duct. The governing equations are the continuity and the Reynolds-Averaged Navier-Stokes equations for incompressible, viscous and turbulent flow which utilizes 2 equations $k-\varepsilon$ turbulence model. Assuming steady state, the flow domains are disintegrated into those of rotor and guide vane, where the former is in rotationally frame and the latter in stationary one. In addition, only 1/10 of the rotor domain and 1/19 of the guide vane domain are modeled considering periodicity. The interaction between two different reference frames is carried out by mixing plane model which considers the effects on other fluid domain in circumferential average sense. In the mixing plane model, the flow field in the fluid domains surrounding rotor and guide vanes are separately solved in steady state using iterative algorithm for simultaneous equations. During the iteration, the flow field including velocity and pressure in rotor fluid domain is evaluated and circumferentially averaged on the interface. The resulting distributions of the flow variables in the radial direction are imposed on the boundary of the fluid domain for guide vanes. The effects of the flow field in the fluid domain for guide vanes on that of rotor are similarly considered. In Figure 4 depicts schematic arrangement of rotor and guide vane and the grid on the surface of rotor and guide vane is shown in Fig. 4. The principal dimensions of the axial flow fans of interest are listed in Table 1.

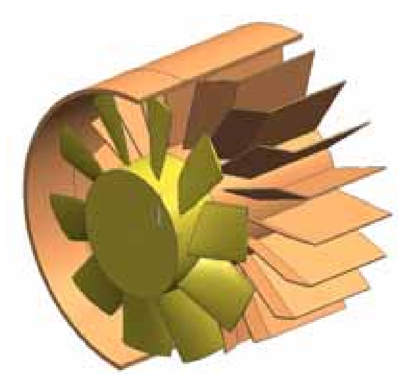

Figure 1. Schematic diagram of axial flow fan

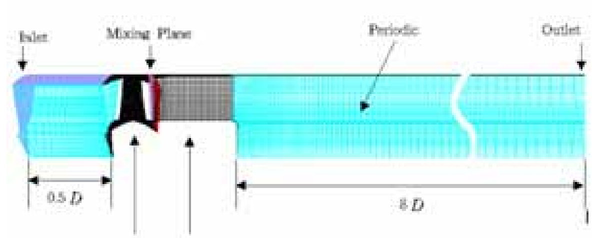

Figure 2. Computational domain

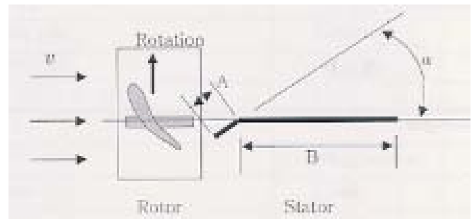

Figure 3. Rotor-Guide Vane arrangement $(\alpha = 25^o)$

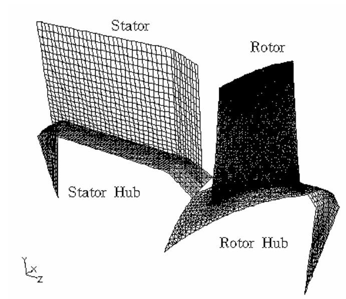

Figure 4. Computational model

Table 1. Principal dimensions of axial flow fan (Length normalized by duct diameter)

Hub Diameter	0.456
Tip chord length	0.189
Hub chord length	0.2
Tip maximum thickness	0.018
Hub maximum thickness	0.009
Tip maximum Camber	0.009
Hub maximum Camber	0.004
Pitch angle difference between Tip and Hub	15 degree
Rotational Speed	1760 rpm

EXPERIMENTAL SETUP

For full-scaled tests using prototypes, the experimental procedure was designed according to corresponding Korean Standards [5]. Figure 5 depicts the schematic arrangement of the experimental facility. The testing condition is controlled by the throttle apparatus installed at the end of the test duct, which sets up different pressure condition for the fan. Once the apparatus is set up, axial flow velocity as well as static pressure in are measured using Pitot tubes across the duct section in downstream. The measured results are quantified as the following non-dimensional coefficients;

$$\Phi = \frac{Q}{ND^3} \text{ : flow rate coefficient}$$

$$\Psi_S = \frac{\Delta p_S}{\rho N^2 D^2} \text{ : static pressure coefficient}$$

$$\Psi_d = \frac{\Delta p_d}{\rho N^2 D^2} \text{ : dynamic pressure coefficient}$$

$$\eta = \frac{Q(\Delta p_s + \Delta p_d)}{P_e} \text{ : efficiency}$$

where Q and N are the flow rate and the rotational speed, respectively. Δp_S and Δp_d are the static and the dynamic pressures measured, respectively, and P_e is the electric power consumed.

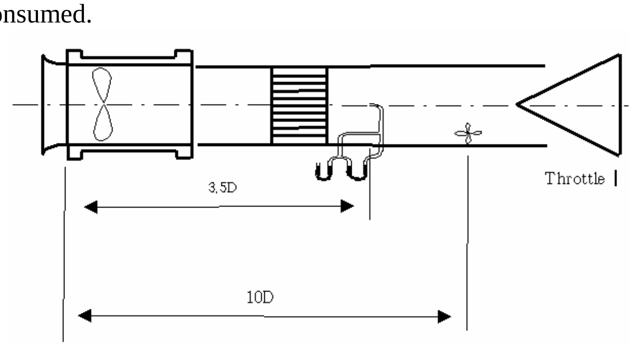

Figure 5. Experimental setup

RESULTS AND DISCUSSION

Figures 6 and 7 show the effects of tip clearance on the static pressure raised by the fan and the efficiency. The pitch angle of rotor blade at hub is 40 degree and the considered cases include tip clearances of 0.1%, 0.3% and 0.5% of the duct diameter. The computational results show about 5% of discrepancy from the measurements for all cases; however, the nearly identical behaviors in the cases of 0.3% and 0.5% tip clearance observed in the experiments are clearly reproduced in the computations. The discrepancy between two results becomes larger as the flow rate decreases and the static pressure increases. It may be attributed to the mixing plane model utilized in the present computations which found that reversed flow occurring near hub at high loading state is not properly considered in average sense. The good agreement between computations and measurements are also found in the efficiency shown in Fig. 7, both of which found about 2% of increase in efficiency by reducing tip clearance from 0.3% of duct diameter to 0.1%; however, it should be noted that reduction of tip clearance can rapidly increase cost of production.

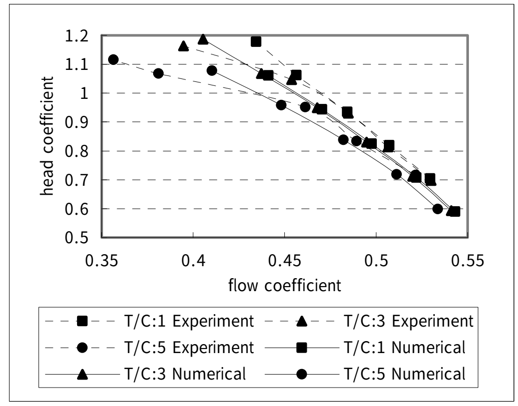

Figure 6. Effects of tip clearance on static pressure

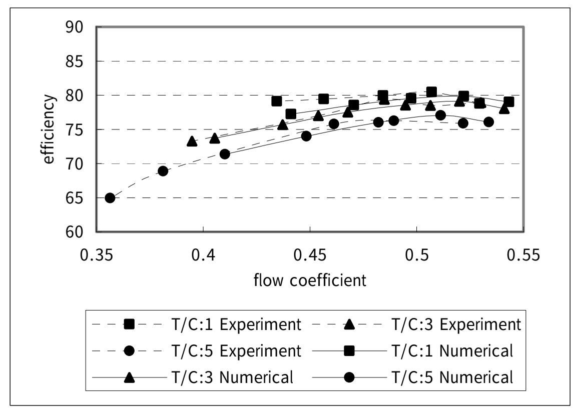

Figure 7. Effects of tip clearance on efficiency

Figure 8 shows the effect of guide vane on the efficiency of the axial fan. The cases considered are listed in Table 2 where the notation is depicted in Fig. 4. The computational results show that reduction of A in Fig. 4 by half increases the efficiency by about 2%, which is attributable to the increase of the static pressure by the fan. The effects of guide vane dimensions on the static pressure are shown in Fig. 9. The diagram illustrates higher static pressure at constant flow rate when the length of A is cut by half. Considering the role of the guide vane as recovery of rotational flow velocity in the rotor downstream, Fig. 10 clearly demonstrates the results of higher efficiency with shorter length of A. In Fig. 10, the circumferentially averaged tangential velocity distributions in radial direction are compared at inlet and outlet of the guide vanes for considered cases. It is found that the recovery of kinetic energy to static head is higher when the length of A is shorter.

CONCLUSIONS

In the present study, the Navier-Stokes equations are solved for flow field around axial flow fan consisting of rotor and guide vane. The performance obtained from the computation is compared with the full-scaled test using prototypes. In particular, the effects of tip clearance and the shape of guide vane are highlighted. From the foregoing results and discussion about computations and measurements, the following conclusion can be drawn;

1. The present computation reproduced main feature of the flow field as well as performance characteristics of the axial flow fan of interest. Axial flow fans with the tip clearance of 0.03% and 0.01% result in efficiency for about 80%, which are commonly found both in computations and experiments.
2. Relatively large discrepancy between computations and experiments is attributable to the iterative nature of the present computation which is employed to consider the effects of different reference frames in averaged sense.
3. The length of guide vane in oblique angle is largely responsible for the recovery of rotational kinetic energy emitted from the rotor. By changing the length of the part, the efficiency varies as much as 2%.

ACKNOWLEDGMENTS

This research was supported by the Program for the Training of Graduate Students in Regional Innovation which was conducted by the Ministry of Commerce, Industry and Energy of the Korean Government

Table 2. Guide vane dimensions

Cases	A/D	B/D
1	0.111	0.394
2	0.111	0.197
3	0.056	0.394
4	0.056	0.197

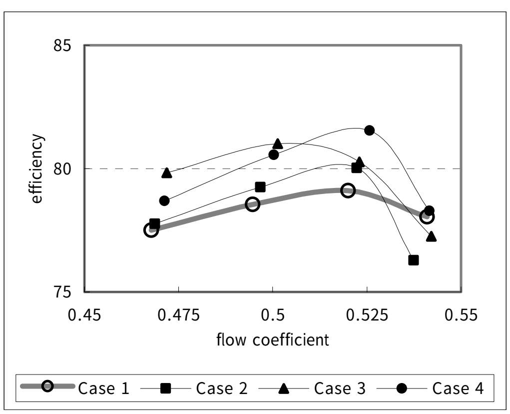

Figure 8. Effects of guide vane on efficiency

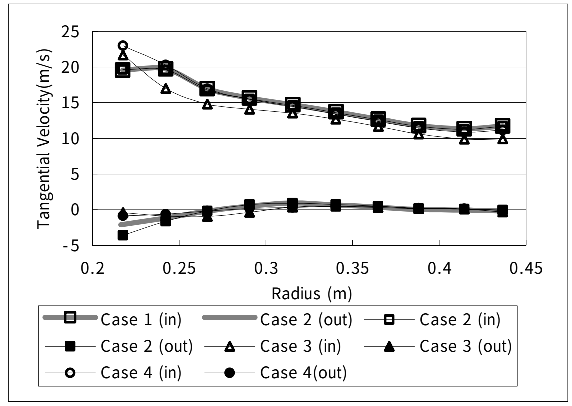

Figure 9. Effects of guide vane on rotational velocities at inlet and outlet of guide vane

REFERENCES

[1] Kim, K. Y., Kim, J. Y., and Jeong D. S., 1997, "Three-Dimensional Analysis of the Flow through an Axial-Flow Fan," J. of KSME(B), Vol. 21, No. 4, pp. 541-550.
[2] Rhie, C. M. et al, 1994, "Advanced Transonic Fan Design Procedure Based on a Navier-Stokes Method," ASME J. of Turbomachinery, Vol. 116, pp. 291-297.
[3] Kim, K. Y., Kim, S.H., Kim, T. J., and Chang, C.M. 1994, "Three-Dimensional Viscous Flows Through An Axial Flow Fan," The 5th International Symposium on Transport Phenomena and Dynamics in Rotating Machinery (ISROMAC-5), Maui, Hawaii, Vol. 2, pp. 190-180.
[4] Oh, K.-J., and Kang, S.-H., 1999, "A numerical investigation of dual performance characteristics of a small propeller fan using viscous flow calculations," Computers & Fluids, Vol. 28, pp. 815-823.
[5] Korean Standard Association, 2001, "KS for Fan Test : KS B 6311."