Springer

Journal of Mechanical Science and Technology

Journal of Mechanical Science and Technology 23 (2009) 1840~1845

www.springerlink.com/content/1738-494x
DOI 10.1007/s12206-009-0612-0

The effect of blade angle on the flow and pressure distributions in the vicinity of the diffuser blades of a room air conditioner †

Hyeon-Seok Seo[1], Jin-Beak Kim[2] and Youn-Jea Kim[3,*]

[1]*Plant Research Division, Korea Institute of Construction Technology,Gyeonggi-Do, 411-712, Korea*
[2]*Samsung Electronics Co., Meatan-dong, Suwon 443-742, Korea*
[3]*School of Mechanical Engineering, Sungkyunkwan University, 300 Cheoncheon-dong, Suwon 440-746, Korea*

(Manuscript Received March 16, 2009; Revised April 29, 2009; Accepted April 29, 2009)

Abstract

Many studies on air-conditioning systems are more focused on the individual thermal comfort rather than the thermal efficiency, due to an increase in health concerns. There are several factors influencing the thermal comfort, such as temperature, humidity, convection and air movement, etc. Numerical analyses were performed to investigate the effect of blade angle on the flow characteristics in the vicinity of diffuser blades of a room air conditioner (RAC), with three different blade discharge angles of 45.1°, 58.6° and 116°. We used the commercial code FLUENT to calculate the two-dimensional steady thermal flow fields with different impeller rotational velocities. The angular velocities were located within the range from 900 rpm to 1200 rpm. Turbulence closure was achieved using a standard k-ε model. A moving reference frame (MRF) approach was adopted to simulate the flow field generated by the impeller in an RAC. The results were graphically depicted with various geometrical configurations and operating conditions.

Keywords: CFD (Computational Fluid Dynamics); Discharge angles; RAC (Room Air Conditioner); Rotational velocity

1. Introduction

RAC (Room Air Conditioner) is composed of a heat exchanger, a cross-flow fan, a stabilizer, a rear guider and a blade with a diffuser region, etc (see Fig. 1.). The cross-flow fan is a notable recent addition to the indoor units of an air conditioner. Fluids pass through an impeller blade twice and the impeller has a large absolute flow velocity because it is a forward curved blade. This can reduce the rotating speed to achieve an equal pressure difference at the same flow rate in comparison to other types of blades. There are two different types of vortices in the flow fields in an RAC unit. One is an eccentric vortex, i.e., forced vortex, induced by recirculation from the stabilizer to the impeller, and the other is the free vortex in the rear guider [1]. The impeller has two regions that work as a turbine and a pump as a result of these vortices. The control of the location and intensity of the eccentric vortex is important and it is directly connected to the fan performance in the RAC unit.

Several empirical and numerical studies have been conducted on the shape parameters of the components

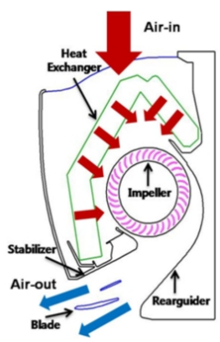


Fig. 1. Schematic diagram of the RAC unit.

† This paper was presented at the 7th JSME-KSME Thermal and Fluids Engineering Conference, Sapporo, Japan, October 2008.

*Corresponding author. Tel.: +82 32 872 309682 2 705 8635, Fax.: +82 32 868 171682 2 712 0799
E-mail address: kykim@inha.ac.kryjkim@skku.edu

in a cross-flow fan.

Eck [1] studied all of the theories on the cross-flow fan using experimental and analytical methods. First, he studied the eccentric vortex through flow visualization. Tsurusaki et al.[2, 3] measured the velocity of the internal flow in the cross-flow fan using particle tracking velocimetry (PTV). They also found the path line and velocity distribution using a digital camera and studied the generation and diffusion of the vorticity induced by the eccentric vortex. Yamafuji and Nishihara [4] clarified the generation procedures of irregular main flow by LDV (laser doppler velocimetry) and disclosed the production of vortex shedding at a blade tip. It has been verified that the principle of the eccentric vortex could be understood from those phenomena. Murata et al. [5-7] inquired that the setting angle, the gaps between components and the rear guider shape, are important design parameters and have an influence on the fan performance. In the case of a small variation in the Reynolds number with a corresponding small change in the diameter and the rotational velocity, it is reported that the flow and pressure coefficients are valid by means of the study on the scale effect versus Reynolds number.

This paper numerically investigates the effects of blade angle on the flow characteristics in the vicinity of diffuser blades of a room air conditioner (RAC).

2. Numerical analysis

The conservative equations for two-dimensional, unsteady, turbulence and viscous flows are as follows:

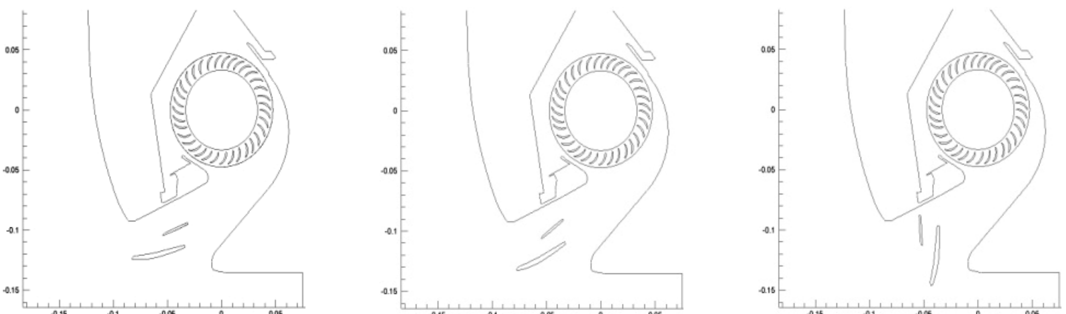
$$\frac{1}{\sqrt{g}}\frac{\partial}{\partial t}(\sqrt{g}\rho)+\frac{\partial}{\partial x_j}(\rho u_j)=0 \qquad (1)$$

$$\frac{1}{\sqrt{g}}\frac{\partial}{\partial t}(\sqrt{g}\rho u_i)+\frac{\partial}{\partial x_j}(\rho\tilde{u}_j u_i-\tau_{ij})=-\frac{\partial p}{\partial x_i}+S_i \qquad (2)$$

These governing equations were discretized by the finite volume method (FVM) to find the solutions for flow variables. The hybrid scheme was used to handle the convection and diffusion terms in the governing equations. The standard k-ε turbulence model with the wall function was adopted to simulate the behaviors of turbulence. Analyses were carried out until solutions reach a steady state.

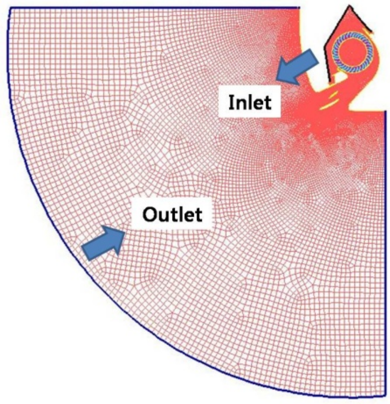
The numerical domain consists of the inlet region, the impeller, the rear guider, the stabilizer and the discharge blade. These are all needed to solve the complicated relations among the elements of the RAC. This study selected three different blade discharge angles (β): Case 1 has β=45.1◦, Case 2 has β=58.6◦ and Case 3 has β=116◦. A numerical model was made by the CAD tool IDEAS, as is shown in Fig. 2.

The computational grid system was formed by the CFD preprocessor GAMBIT, as is shown in Fig. 3. The unstructured grid generation method and especially fine grid system were applied near the impeller since the shape of the impeller in the pump is relatively complex. This area is entirely composed of 2 continuity zones for the flow pattern; the solid wall of the impeller and blade, and the fluid zone inside the modeled RAC. The grid system containing between 60,000 and 80,000 elements was selected. The selection of this region represents the best compromise between computational time and accuracy for this problem.

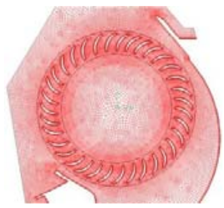


(a) Case 1(β=45.1°) (b) Case 2 (β=58.6°) (c) Case 3 (β=116°)

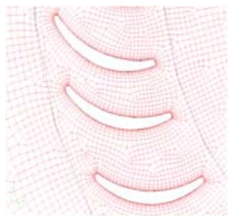
Fig. 2. Configuration of the modeled RAC unit.



Stationary zone

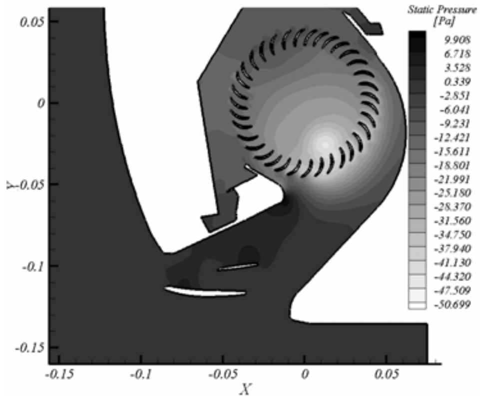


Moving zone

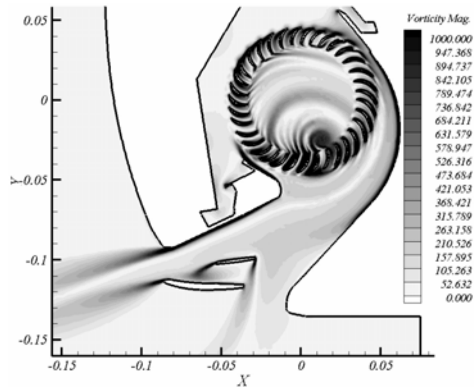


Impeller blade

Fig. 3. Grid systems of the modeled RAC unit.



(a) Pressure



(b) Vorticity

Fig. 4. Typical pressure and vorticity distribution pattern in the RAC.

The numerical analysis was executed with three different discharge angles for the diffuser blade and three different values of the angular velocity: 900, 1050 and 1200 rpm. In general, the impeller of a commercial room air conditioner operates at about 900~1200 rpm. There is no influx from the outside of the RAC unit, and the flow in the RAC unit completely results from the rotating impeller.

Therefore, the no-slip wall boundary condition was applied on all the boundaries, and all the walls were assumed to be adiabatic. Air was used as the working fluid in the system and its viscosity (μ) and density (ρ) are 1.7849 x 10-5 kg/m-s and 1.225 kg/m3, respectively.

Using the commercial CFD code FLUENT, the governing equations were discretized with the finite volume method (FVM) and solutions were obtained by the semi-implicit method for pressure-linked equations (SIMPLE) algorithm [8]. The moving reference frame (MRF) method was applied to embody the effect of the rotating impeller.

3. Results and discussion

Fig. 4 shows the general pattern of the pressure and vorticity distributions of the RAC unit. It is seen that the pressure drops down in the vortex region and this pressure drop occurred intensively near the vortex. This result occurs because the velocity mainly governs the flow behaviors in the RAC due to the small variation in the pressure. The velocity around the vortex is increased as the radius increases. Therefore, the vortex is the forced vortex and the minimum pressure occurs at the center of the vortex. It is also confirmed that the magnitude of the vorticity is the greatest in this field [1].

The velocity distributions in the RAC region with the various values of discharge angle and angular velocity of the impeller are shown in Fig. 5. For all cases, the magnitude of the discharge velocity increases as the angular velocity of the impeller increases. However, the results of Case 3 show different flow natures compared to the other cases because the discharge flow is hindered at the upper blade. This

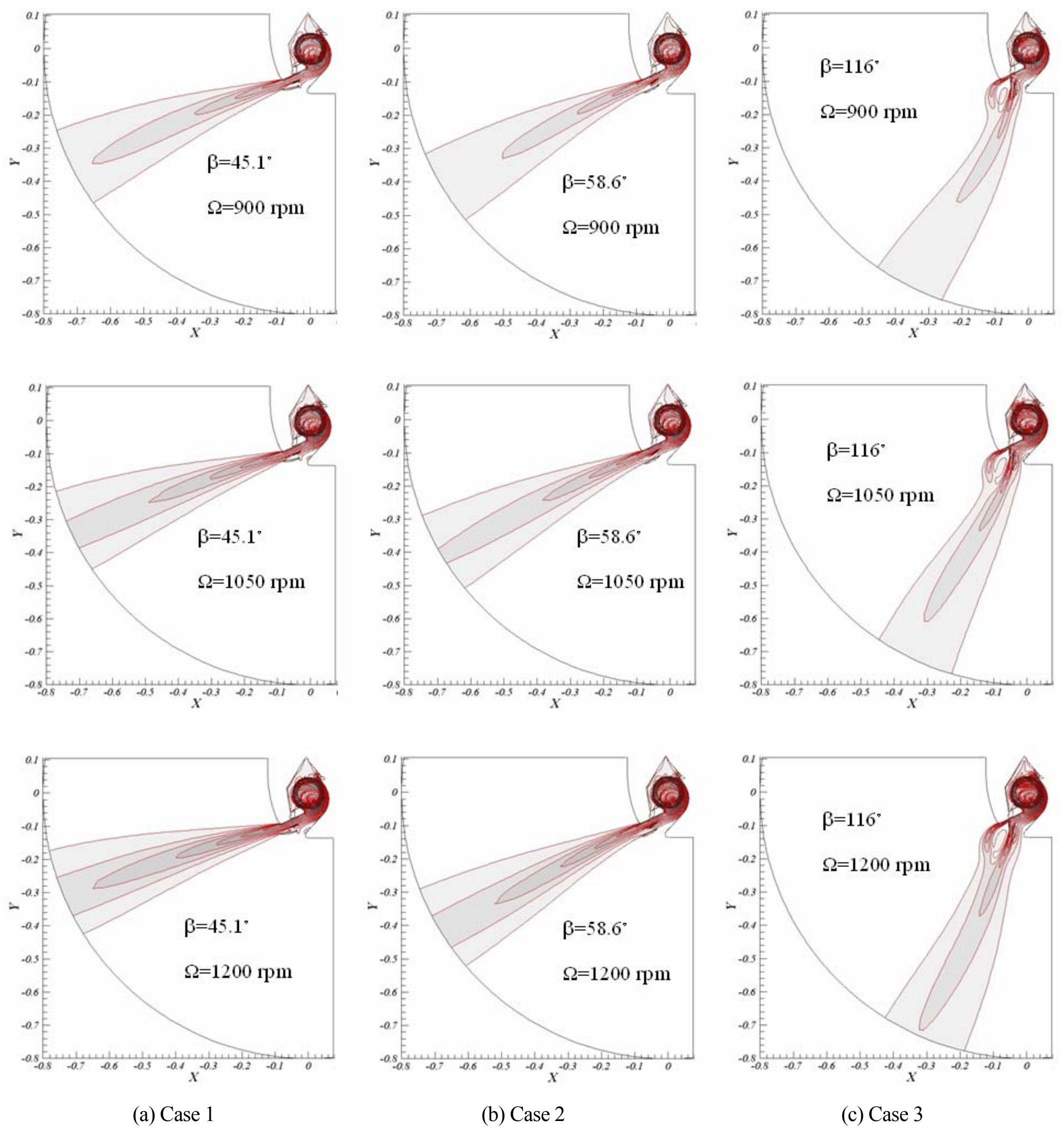
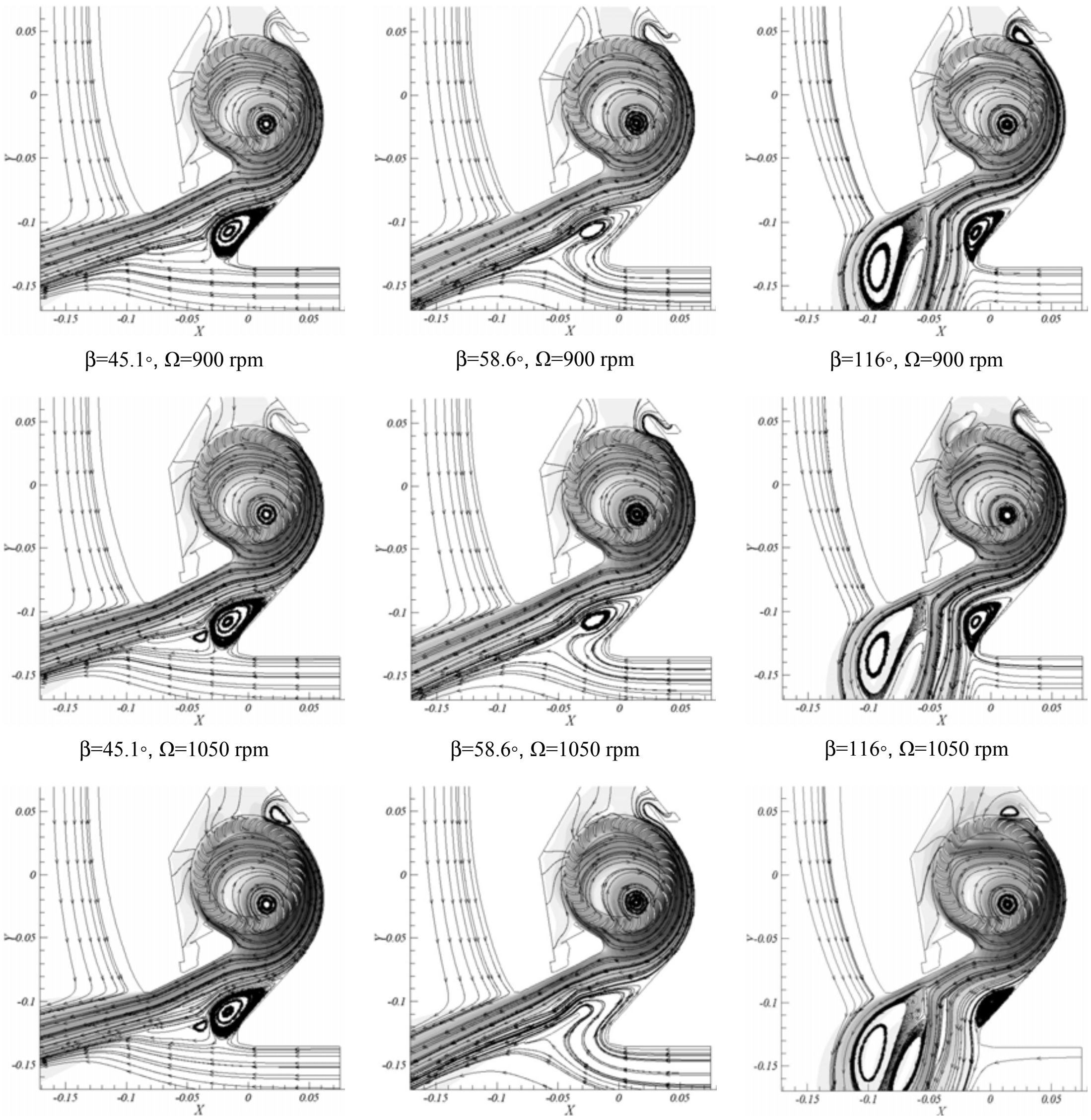


Fig. 5. Velocity distributions with three different values for the discharge angle in the indoor region.

results in a change in the velocity direction.

Fig. 6 presents the computed streamline distributions of the modeled RAC with three different blade shapes and three different values of angular velocity of the impeller. The working fluid passes the first blade cascade through the right side of the impeller and enters the second cascade on the left side. The fluid obtains its momentum from both the centrifugal force and the Coriolis force generated by the rotation of the impeller. The working fluid is also discharged into the rear guider within the discharge region. However, some of the primary discharged fluid is recirculated to the impeller, which results in the eccentric vortex [9].

For all cases of the modeled RAC, there was a generated vortex field in the impeller region and between the diffuser blade and the rear guider. It is seen from the results of the streamline with various discharge angles of the diffuser blade that the vortex field is less distributed between the diffuser blade and the rear guider for Case 2. However, the vortex field is widely formed in the vicinity of the diffuser blade for Case 3. This causes a reduction in the discharge performance of the RAC.



β=45.1◦, Ω=900 rpm β=58.6◦, Ω=900 rpm β=116◦, Ω=900 rpm

β=45.1◦, Ω=1050 rpm β=58.6◦, Ω=1050 rpm β=116◦, Ω=1050 rpm

β=45.1◦, Ω=1200 rpm β=58.6◦, Ω=1200 rpm β=116◦, Ω=1200 rpm

(a) Case 1 (b) Case 2 (c) Case 3

Fig. 6. Streamlines with the various values of discharge angle in the vicinity of the diffuser blade in RAC.

4. Conclusions

Numerical analyses were performed to investigate the effect of the blade angle on the flow characteristics in the vicinity of diffuser blades of an RAC. Additionally three different values of angular velocity were studied. The following conclusions are obtained:

1) The results of Case 2 indicate that the velocity is fully developed and the vortex distributions are their lowest. Judging from these results, we may deduce that the discharge performance is the best for Case 2.

2) The results of Case 3 show that a large vortex zone is formed in the vicinity of the diffuser blade due to the upper blade. This results in a decreasing discharge performance of the RAC unit.

3) The magnitude of the discharge velocity increases as the angular velocity of the impeller increases. It is also confirmed that the rotational frequency of the impeller has little effect on the vortex

fields.

Acknowledgement

Financial aid from the Korean Ministry of Education through the Brain Korea 21 Project is gratefully acknowledged. The authors are also grateful for the financial support provided by Samsung Electronics through the Smart Future Appliance Research Center (SFARC, 2006-0956-000) at the Sungkyunkwan University in Korea.

References

[1] B. Eck, *Fans*, Pergamon Press, New York, (1973).
[2] H. Tsurusaki, H. Shimizu, Y. Tsujimoto and Y. Yoshida, Study of Cross-Flow Fan Internal Flow by Flow, *JSME Int. J. series B*, 39 (1996) 540-545.
[3] H. Tsurusaki, Y. Tsujimoto, Y. Yoshida and K. Kitagawa, Visualization Measurement and Numerical Analysis of Internal Flow in Cross-Flow Fan, *J. of Fluids Engng.*, 119 (1997) 633-638.
[4] S. Yamafuji and K. Nishihara, An Experimental Study of Cross Flow Fan, *Bulletin of JSME*, 19(129) (1976) 314-321.
[5] S. Murata and K. Nishigara, An Experimental Study of Cross Flow Fan (1st Report, Effects of Housing Geometry on the Fan Performance), *Bulletin of JSME*, 19(129) (1976) 314-321.
[6] S. Murata and K. Nishigara, An Experimental Study of Cross Flow Fan (2nd Report, Movement of Eccentric Vortex inside Impeller), *Bulletin of JSME*, 19(129) (1976) 322-329.
[7] S. Tanaka and S. Murata, Scale Effects in Cross-Flow Fans, *JSME Int. J. series B*, 38(3) (1995) 388-397.
[8] H. K. Versteeg and M. Malalasekera, An Introduction to Computational Fluid Dynamics the Finite Volume Method, *John Wiley & Sons*, New York, (1995).
[9] D.-W. Kim, H.-S. Kim, S.-K. Park and Y.-J. Kim, Analyses on Flow Fields and Performance of a Cross-Flow Fan with various Setting Angles of a Stabilizer (in *Korean*), *J. of Comp. Fluids Engng.*, 10(1) (2005) 107-112.

Youn-Jea Kim received his B.S. degree in Mechanical Engineering from Sungkyunkwan University, Korea, in 1982. He then received his M.S. and Ph.D. degrees from the State University of New York at Buffalo in 1987 and 1990, respectively. Dr. Kim is currently a Professor at the School of Mechanical Engineering at Sungkyunkwan University in Korea. Dr. Kim's research interests include gas dynamics, MEMS, and fluid-machineries, etc.