THE 5TH ASIAN COMPUTATIONAL FLUID DYNAMICS
BUSAN, KOREA, OCTOBER 27~30, 2003

Optimum Design of a Cross Flow Fan

Dong-Hoon Kim [1]**, *Hyung-Koo Park*** [2]
1. School of Mechanical Systems Engineering, Chonnam National University, coldhoon@hotmail.com
2. School of Mechanical Systems Engineering, Chonnam National University, park@gas.chonnam.ac.kr

Abstract

Cross-flow fans are widely used in various applications, due to their large capacity of mass flow and size compactness. The flow field of the cross-flow fan is, however, complex and has many design parameters. Thus, the general design guide has not been sufficiently established yet and the design strategies of cross-flow fans have been based on experiments. In the present study, the performance and their two-dimensional flow characteristics are numerically analyzed by using the STAR-CD(commercial computational fluid dynamics code). The simulation is done by varying the several design parameters such as the impeller blade shapes and the gap between the stabilizer and impeller. The computational results are compared with the experimental data at the fan outlet region. Finally some helpful guides for the optimum design of cross-flow fans are proposed.

Keyword: *Cross-Flow Fan, Stabilizer, Impeller, Blade, STAR-CD, ICEM-CFD*

1. Introduction

It is possible for cross-flow fans to control mass flux by adjusting to the length of the fan without changing the revolutions and diameter of impeller. The axi-directional flow is can be neglected due to the fact that the flow crosses the fan and forms a two-dimensional vortex flow. Since the design factors of a cross-flow fan have interrelation and performance is changeable according to compounding variables, experimental researches have some limitations. Therefore, numerical analysis is promising, and commercial CFD programs which are improved in accuracy and the ability of mesh generation are used broadly[5,6]. STAR-CD is used for numerical analysis in this work. This study considered how the shape of blade, stabilizer and impeller gab affect the performance of cross-flow fan systems which are applied to the ventilator of refrigerators.

2. Geometry and specification of cross-flow fan

Fig. 1 shows the geometry of the basic model and the component name. The detail blade shape of the cross-flow fan is shown in Fig. 2. And inner angle(β1), outer angle(β2), inner radius and outer radius of impeller are specified.

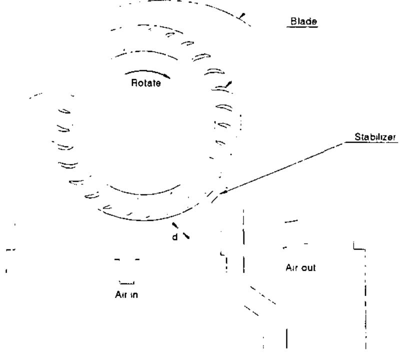

Fig. 1 Geometry of the basic model

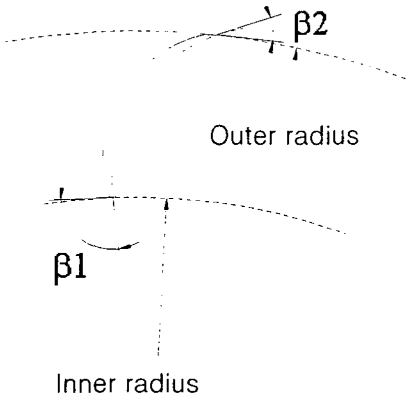

Fig. 2 Blade geometry

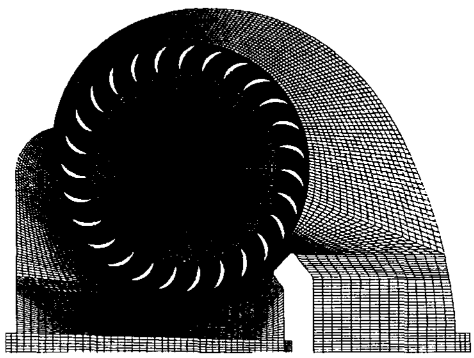

Fig. 3 Computational mesh

3. Mesh generation

A commercial mesh generation code, ICEM-CFD was used for the fast mesh generation processes. Fig. 3 shows the generated mesh shape of the two-dimensional basic model. Hexa-mesh was used, and the total number of the basic model meshes was about 42500. Among the entire computational domain, the complicated flow variation is expected at the adjacent region of blades and the gap between the stabilizer and impeller. So the mesh refinement was used at these regions. Arbitrary couple [7] was applied at the boundary which has different mesh number for the continuity of meshes.

4. Computational conditions

Both sides of the axi-direction faces in cross-flow fans were considered as a symmetric boundary condition to be assumed as the two-dimensional. A pressure boundary condition was applied at the inlet and outlet of the domain. The whole domain was divided into a moving and fixed domain, and then a moving mesh method was used. The moving mesh method changed the mesh positions of the two domains without changing the mesh number as time changes. Total number of 360 events are used per single rotation of the impeller. In case of 1200 rpm, hence, a single event has the time-step of 0.00013889 sec. An attached boundary was used between the moving and fixed meshes.

5. Computational results and discussion

5.1 Flow field of the basic model

In Fig. 4, the entire flow patterns are shown. The eccentric vortex is especially well shown near the stabilizer. Another vortex flow is shown at the inlet region. It intimates that the change of the inlet region shape leads to the performance improvement of cross-flow fans. Fig. 5 shows that the pressure values are lowest at the center of eccentric vortex. Near the initial point of the scroll, a forced flow by impeller causes high pressure values. So the change of the scroll shape leads to the variation of the flow performance.

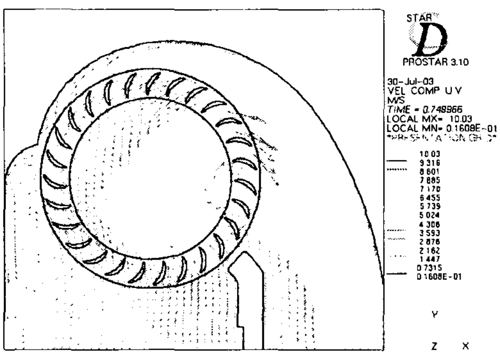

Fig. 4 Velocity vectors of basic model

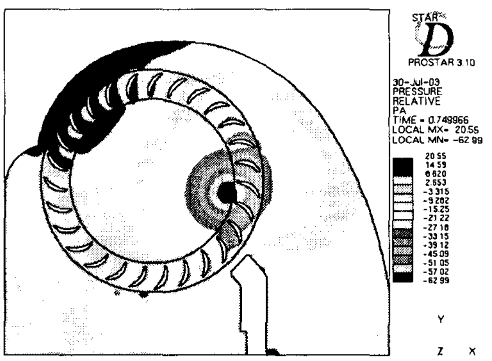

Fig. 5 Pressure distribution of basic model

Table 1 Variation of blade angles

	Inner angle (β1)	Outer angle (β2)
Basic model	95°	24°
Model 1	85°	24°
Model 2	105°	
Model 3	95°	14°
Model 4		34°

Table 2 Flow rate vs. blade shapes

	Flow rate [m^3/min]
Basic model	1.453
Model 1	1.449
Model 2	1.365
Model 3	1.454
Model 4	1.377

5.2 Influence of the gap between impeller and stabilizer

The computational analysis is done by changing the gap between impeller and stabilizer from 2.0mm to 5.0mm. The entire flow patterns and the position of eccentric vortex are almost unchanged and the flow rate is changed only slightly. The pressure value decreases slowly in the beginning, it has the smallest value at the center of eccentric vortex, and then it increases again. From the static pressure distribution, it is known that the change of the gap between impeller and stabilizer has an effect on the pressure field of the inlet region. Accordingly, when the gap between impeller and stabilizer is set up, the shape of scroll and inlet region must be considered.

5.3 Influence of blade shapes

As shown in Table 1, computation is done by changing the inner angle(β1) and the outer angle(β2) independently. Table 2 summarized the computational results for the each model. In comparison with the flow rate of basic model, the results of Model 1 and Model 3 are similar, but those of Model 2 and Model 4 are decreased. This is caused by the change in vortex flow at the inlet region. In the velocity field for each Model, it is known that the vortex region at the inlet is expanded. It is the main reason of the declination in performances. Accordingly, when the blade shape is changed for the performance improvement, the shape of inlet region must be considered.

6. Conclusions

In this study, cross-flow fan systems in the ventilator of refrigerators is numerically analyzed. And we can conclude as follows based on the obtained computational results.

(1) The change of the gap size between the stabilizer and impeller influences the variation of pressure at the inlet region.

(2) According to the decrease of the inner angle(β1) and the outer angle(β2), the vortex of the inlet region is enlarged and the flow rate is decreased. Therefore, for the blade shape the geometry of the inlet region and the starting point of scroll must be considered.

References

[1] Eck B., *"Fan Design and Operation of Centrifugal, Axial-Flow and Cross-Flow Fans"*, Pergramon Press, (1973).

[2] Murata S., Nishihara K., "An Experimental Study of Cross Flow Fan", *Bulletin of JSME*, Vol. 19, No. 129, (1976), pp 314-321.

[3] Fukano T., Hara Y., Yamashita Y., Kinoshta K., "Experimental Study on Cross Flow Fan Noise", *The 4th ASIAN International Conference on Fluid Machinery.*

[4] D. S. Park, J. S. Maeng, S. H. Lee, "Experimental Study for Stabilizer Design of Cross Flow Fan", *Trans. of SAREK*, (1995), pp 238-246.

[5] Akaike S., Kitada M., Taira H., Kikuyama K., "Flow Analysis of Cross-Flow Fan", *STAR-CD News*, Issue 3, (1995), pp 2-4.

[6] N. K. Hur, W. Kim, S. H. Kang, "A Numerical Study on Flow through a Cross Flow Fan: Effect of Blade Shapes on Fan Performance", *Journal of Fluid Mech.*, Vol. 2, No. 1, (1999), pp 96-102.

[7] *"STAR-CD Version 3.15A Manuals"*, Computational Dynamics Limited, (2002).