44 International Journal of Air-Conditioning and Refrigeration Vol. 9 No. 2 (2001)/pp. 44~50

Computational Analysis of the Three-Dimensional Flow Fields of Sirocco Fan

Jae Hong Hah*, Young J. Moon** and Jin Moo Park**

Key words: Sirocco fan, Three-dimensional Navier-Stokes computation, Quasi-static analysis

Abstract

The Sirocco fan performance and its three-dimensional flow characteristics are numerically predicted by STAR-CD. Turbulent flow computations are performed using approximately 500,000 mesh points, and the performance results of two computational methods, transient and quasi-static flow analyses are compared with experimental data. In the present study, our attention is focused on the three-dimensional flow characteristics of the Sirocco fan, especially the through-flow characteristics of the Sirocco fan blades and the secondary flow structure in the scroll. For a design optimization study, the scroll shape is tilted by 10° to modify the secondary flow structure, which yields some improvement of the fan performance.

1. Introduction

Sirocco fans are widely used for industrial fans as well as in home-appliances, due to its high pressure rise and large capacity of flow rate compared to the size compactness. Because of the complexity of the fan geometry and the three-dimensionality of the internal flow structure, researches have been pursued with some limitations.

Raj and Swim(1) found that the blade active area (or called "wetted area") depends on the circumferential location of the impeller and the flow rate, and Cau et al.(2) attempted to analyze the secondary flow inside the Sirocco fan scroll by applying a simple flow theory. Im et al.(3) investigated the three-dimensional flow structure by using a 5-hole pitot tube, and Maeng et al.(4) experimentally studied the flow characteristics inside the rotor and the hub effect. Gronier and Gilotte(5) performed the three-dimensional flow computations using STAR-CD, and Shin et al.(6) predicted the flow-induced noise by a time-dependent flow analysis.

In the present study, fan performance and three-dimensional flow characteristics are investigated by the transient and quasi-static flow analyses with STAR-CD. The computed results are verified by the experiment and a design optimization study is pursued by modifying the scroll shape, which in fact yields some improvement of the fan performance.

---

* Graduate School, Korea University, Seoul 136-701, Korea

** Department of Mechanical Engineering, Korea University, Seoul 136-701, Korea

2. Background theories

2.1 Numerical methods

The internal flow characteristics of the Sirocco fan are often turbulent. For a rotational speed of 2,500 RPM and a flow rate of 5.0 CMM, the local Reynolds number is roughly 2,700 at the blade passages and 38,000 along the scroll. In this study, the three-dimensional incompressible Navier-Stokes equations are solved in coupled with a standard $k-\varepsilon$ turbulence model. For the convective flux terms, a hybrid scheme is used to locally damp out the numerical instabilities developed by mesh irregularities or by largely separated flows. At the inflow boundary, a uniform flow condition is imposed, while at the outflow boundary a constant pressure condition is applied.

Two computational methods are used in the present study; (i) transient flow analysis and (ii) quasi-static flow analysis. In the transient flow analysis, a moving coordinate is used for the grid rotating with the impeller, while in the rest of the stationary grids the governing equations are solved in an inertia frame of reference. In this approach, computations are performed in a time-accurate manner by a PISO algorithm, and at the interface between two grid systems a sliding mesh technique is utilized to allow for the transfer of transient solutions. In the quasi-static analysis, a rotational coordinate system is used instead, which is a method of solving for the steady-state flow. In this approach, a rotational effect is taken into account by the source terms in the governing equations, and solutions are obtained by a SIMPLE method.

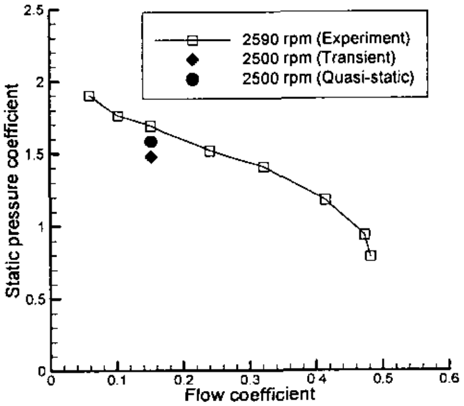


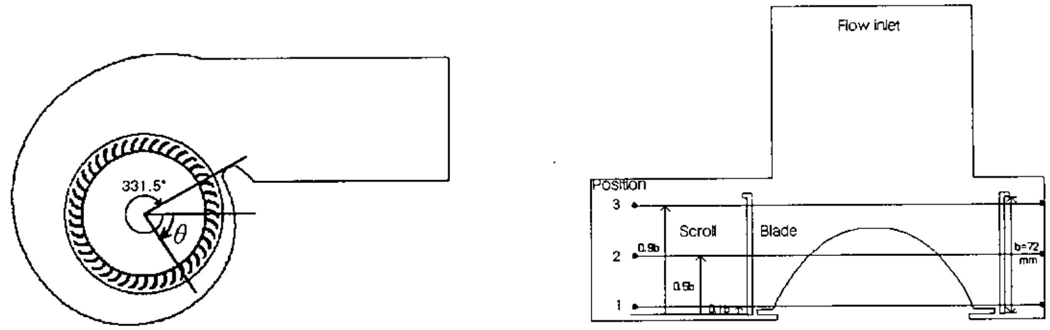
Fig. 1 Performance comparison of transient and quasi-static analyses.

Both computational methods described above are tested for the case of 2,500 RPM and 3.0 CMM. The numerical predicted performances are well compared with experimental data$^{(3)}$ in Fig. 1. The result of quasi-static calculation is slightly under-predicted than that of the transient analysis, but in overall both methods yielded reasonably well matched solutions. In the present study, the quasi-static analysis is further used for investigating the three-dimensional flow characteristics of the Sirocco fan, since this method is computationally much less expensive than the transient flow analysis.

2.2 Fan geometry and grid generation

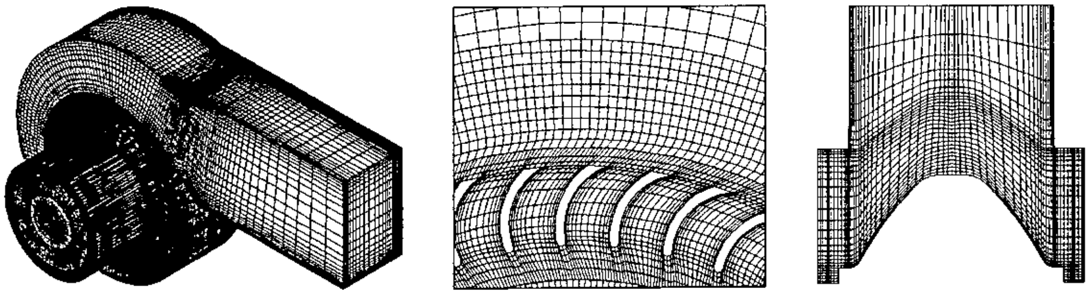
The Sirocco fan has an impeller consisted of 44 forward-curved blades. The blade thickness is 1.5 mm and its span is 72 mm. The outer diameter of the impeller is 147 mm and the inner diameter is 121 mm. The blade inner angle β_1 is 80°, the outer angle β_2 is 140°, the setting angle θ_b is 20°, and the forward-leaning angle δ is 60°. Figure 2 shows the frontal and cross-sectional views of the Sirocco fan geometry.

The grid used in the present Sirocco fan computation is presented in Fig. 3. Figure 3 (a) shows a perspective view of the grid consisted of 494,252 mesh points in total (305,184 points are used for the impeller blades and the scroll). A detailed view of meshes near the blades is shown in Fig. 3 (b). With limited computational resources, only 12×10 mesh points are used within each blade flow passage and mesh sizes



(a) Frontal view and definition of scroll angle (b) Cross-sectional view

Fig. 2 Sirocco fan geometry.



(a) Perspective view (b) Blades (c) Cross-section

Fig. 3 Computational meshes.

are doubled as the radial direction increases. For the case where the mesh points are not aligned, an arbitrary couple technique(8) is used at the interface. Figure 3 (c) also shows a cross-sectional view of the grid, omitting the impeller blade and scroll regions. In the modeling of the tip clearance region between the blade and the hub wall, only 4 mesh points are used across it.

3. Computational results and discussion

3.1 Performance prediction

Performance of the Sirocco fan operating at 2500 RPM is numerically predicted for the two cases of flow rates, 3.0 CMM and 7.0 CMM. The computational results are compared with experimental data(3) obtained at 2,590 RPM. The performance curve is plotted with the static pressure coefficient versus the flow coefficient. Figure 4 indicated that the present three-dimensional computations predict the perfor-

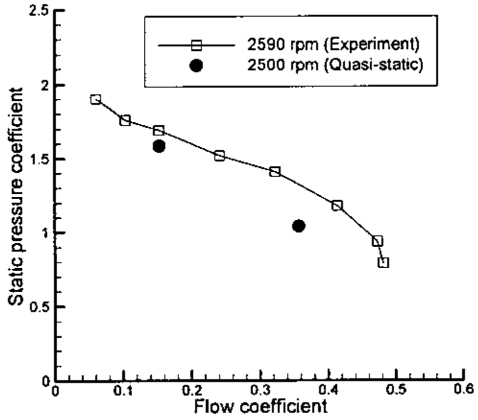


Fig. 4 Comparison of performances between experiment and computation.

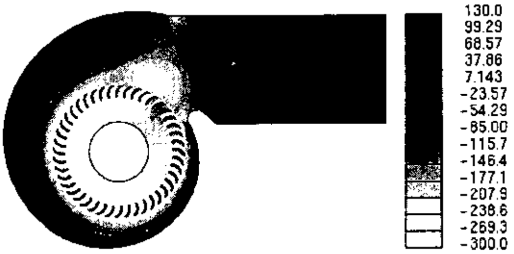


Fig. 5 Static pressure contours on a mid-span plane.

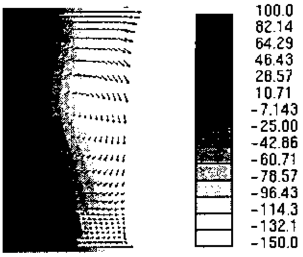
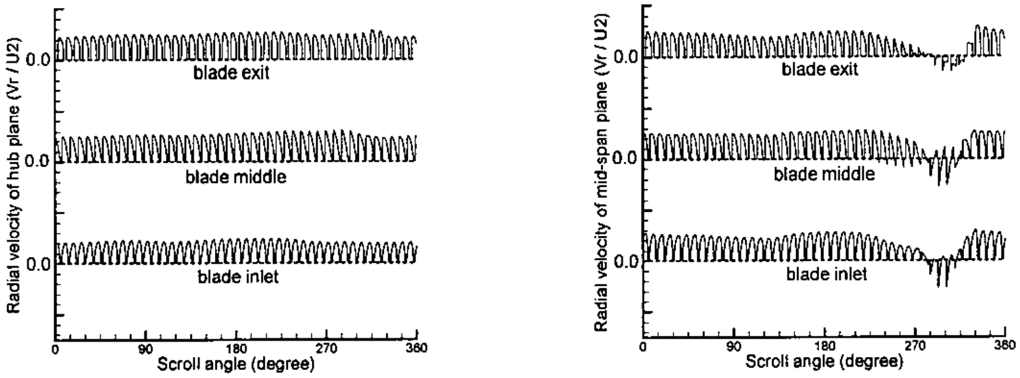


Fig. 6 Secondary flow vectors on a scroll cross-section at 135°.

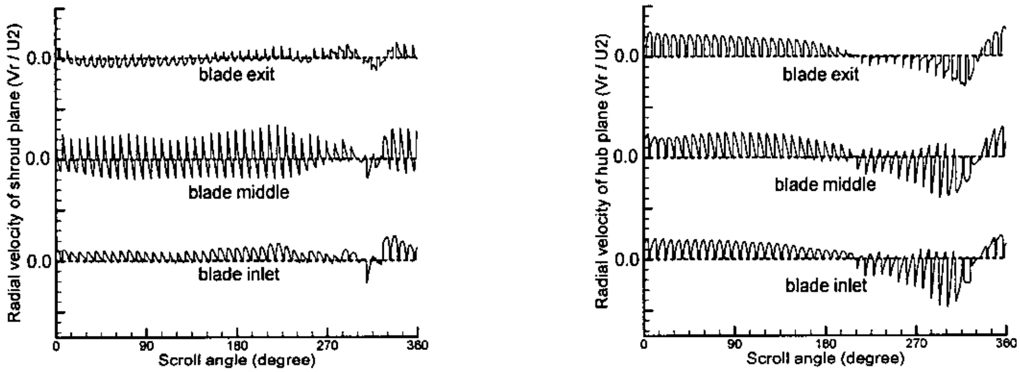
mance of the Sirocco fan with reasonable accuracy.

3.2 Three-dimensional flow characteristics

The pressure field at the mid-plane is presented in Fig. 5 for the Sirocco fan operating at 2,500 RPM and 7.0 CMM. This figure exhibits a typical flow characteristics of the pressure field: pressure rises in the radial and circumferential directions due to the centrifugal force and the energy transfer by the impeller, respectively. One can also notice the flow separation occurrence near the cut-off region.



(a) Hub (7 CMM) (b) Mid-span (7 CMM)



(c) Shroud (7 CMM) (d) Hub (3 CMM)

Fig. 7 Radial velocity profiles.

A secondary flow structure in the cross-section of the scroll is visualized at a circumferential angle 135° by the cross-sectional velocity vectors. Figure 6 exhibits a two-cell structure of the counter-rotating vortices: the upper vortex is in a clockwise rotation, while the lower vortex is in the opposite direction. In this figure, the right vertical line is the blade side and the left one is the scroll side wall.

Figure 7 (a) ~ (c) show the radial-velocity distributions of the blade passage flow at the blade inlet, middle, and exit, all plotted against the scroll angle θ (or the circumferential angle) for 2,500 RPM and 7.0 CMM. The three-dimensional flow structure of the blade passage flow can be more easily understood by investigating the distributions at the three planes near the hub, mid-span, and near the shroud. The velocity vectors shown in the figure are normalized by the blade tip velocity at the impeller outer-diameter. One can see that the radial velocities near the hub plane are all positive, meaning that flows are well passing through the blades near the hub plane. However, at the mid-span plane, a reverse flow starts to occur at $285° < \theta < 315°$, and near the shroud plane flows are not passing through the blades at all scroll angles. In fact, the flow is largely re-circulating in the middle of the blade passages. Figure 7 (d) also shows the blade passage flows near the hub plane at 3.0 CMM, which indicates a large flow-reversal occurrence at $\theta > 200°$. This is a typical characteristics of the flow instability occurred at lower flow rates.

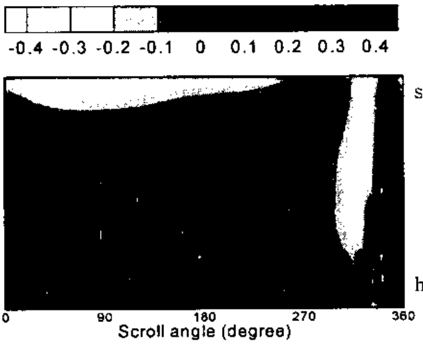


Fig. 8 Radial velocity contours at an impeller rotor exit plane (7 CMM).

Figure 8 presents a radial-velocity distributions at the impeller flow-exit plane (i.e., $\theta - z$ plane 6 mm away from the blade exit) for 2,500 RPM and 7.0 CMM. The velocity magnitude is similarly normalized as in Fig. 7. This can be interpreted as a through-flow map of the blade. As already mentioned, one can easily identify the dead-flow region near the shroud and most importantly the occurrence of the flow separation near the cut-off region ($\theta > 270°$). The location and the shape of the cut-off is directly related to the three-dimensional flow characteristics of the Sirocco fan. This particular region also accompanies the flow unsteadiness, which is strongly associated with the aero-acoustic noise of the Sirocco fan.

3.3 Design optimization of scroll shape

The primary energy loss of the Sirocco fan comes from the separated flow near the cut-off region as well as from the secondary flow along the scroll flow passage. This mechanical energy loss is directly related to the Sirocco fan performances, and therefore the design parameters that determine the shapes of the cut-off or the scroll need to be optimized. In the present study, the scroll shape is modified to reduce the flow loss caused by the secondary

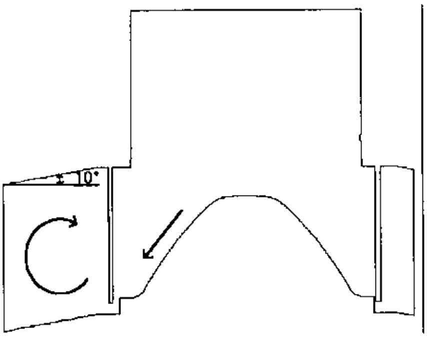


Fig. 9 Sirocco fan with 10° tilted scroll.

flow development.

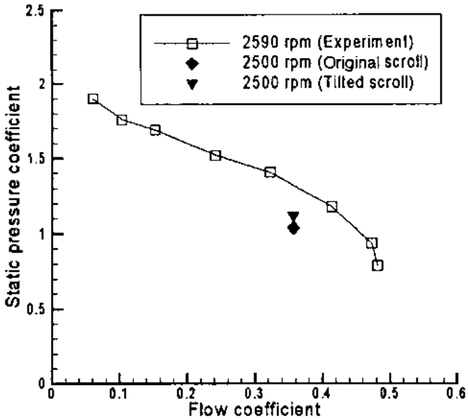


Fig. 10 Comparison of performances between original and tilted scrolls.

Table 1 Comparison of circulations between original and tilted scrolls

Scroll angle		Original (m^2/sec)	Tilted (m^2/sec)	Reduction (%)
135°	Γ^+	+0.560	+0.534	4.64%
	Γ^-	−1.002	−0.963	3.89%
180°	Γ^+	+0.596	+0.586	1.68%
	Γ^-	−1.043	−1.023	1.92%

Figure 9 shows a Sirocco fan with 10° tilted scroll, which is supposed to weaken the secondary flow development in the scroll. The modified configuration of the tilted scroll is tested for computation at 2,500 RPM and 7.0 CMM, and its flow and performance characteristics are examined. The fan performances with original and modified scrolls are presented in Fig. 10. The tilted scroll fan yielded 8.1% improvements of the performance over the original scroll shape.

At the scroll angle 135°, the secondary flow strengths of the original and modified scrolls are examined by evaluating the circulations at the cross-sectional area of the scroll. In Fig. 11, contours of the streamwise vorticities generated by the secondary flows are compared. The computed results indicated that a dominant vorticity strength near the hub is approximately 500 rad/sec. Its converted value in terms of the rotational speed is 2,387 RPM, which is then roughly close to the rotational speed of impeller, 2,500 RPM. Therefore the strength of the lower vortex near the hub must be weakened to improve the fan performance. The circulation amounts of the two scroll shapes are summarized in Table 1 at the scroll angles, 135° and 180°. At both angles, the 10° tilted scroll yielded 5% and 2% reductions of the circulations in the scroll flow passage.

4. Concluding remarks

(1) In the present study, it was verified that the computational prediction of the Sirocco fan

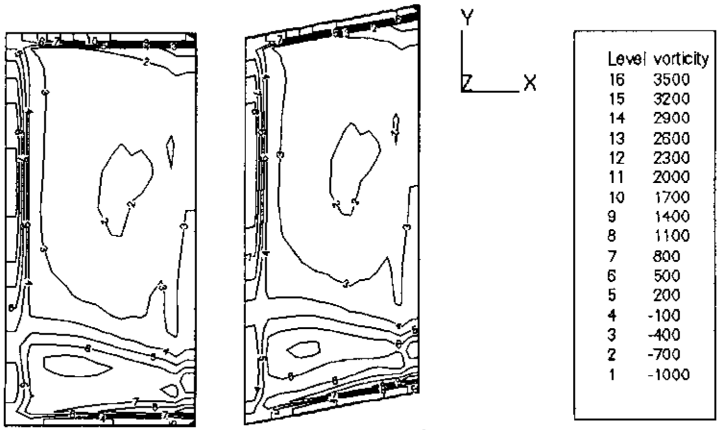


Fig. 11 Comparison of vorticity contours at the scroll cross-sections (135°).

performance is equally valid by both transient and quasi-static analyses. Therefore the flow characteristics at the quasi-steady states and the performance of the Sirocco fan are investigated by the latter approach with much less computational time required.

(2) The present three-dimensional computation of the Sirocco fan is validated by comparing the predicted performances with experimental data.

(3) The three-dimensional flow structure analysis in the blade flow passage of the present Sirocco fan reveals that flows are largely separated near the cut-off and the flows near the shroud are re-circulating within the blade passages, i.e. a possible dead-flow zone.

(4) The tilted scroll weakens the strength of the vorticity developed near the hub, which improves the fan performances.

Acknowledgements

The present study was supported through 1999 Grant for University Research Institutes (97-005-E00199) by Korea Research Foundation.

References

1. Raj, D. and Swim, W. B., 1981, Measurements of the mean flow velocity and velocity fluctuations at the exit of an FC centrifugal fan rotor, J. Engng for Power, Vol. 103, pp. 393-399.
2. Cau, G., Mandas, N., Manfrida, G. and Nurzia, F., 1987, Measurement of primary and secondary flows in an industrial forward-curved centrifugal fan, J. Fluids Engng, Vol. 109, pp. 353-358.
3. Im, J., Moon, Y. J. and Choi, Y., 1997, Experimental measurements of the three-dimensional flow field in Sirocco fan, Proc. of the 5th Asian Int. Conf. on Fluid Machinery, Vol. 3, pp. 803-808.
4. Maeng, J. S., Yoon, J. Y., Ahn, T. B., Yoon, J. E. and Hahn, D. J., 1999, An Experimental Study for Flow Characteristics Inside the Rotor of a Multiblade Fan/Scroll System, KSME Journal B, Vol. 23, No. 5, pp. 646-652.
5. Gronier, P. and Gilotte, P., 1996, Airflow simulation of an automotive blower for a HVAC unit, SAE, 960961.
6. Shin, D. S., Im, J. S., Kim, C. S., Rho, O. H. and Lee, S. G., 1999, Flow Field Analysis of a Centrifugal Fan, Journal of Fluid Machinery, Vol. 2, No. 1, pp. 43-49.
7. Moon, Y. J., 1997, Thermal Flow Analysis of HVAC System; Experiment and Computation, Hyundai Motor Company Research Report.
8. 1996, STAR-CD Version 3.0 Manuals, Computational Dynamics.