Performance and Internal Flow of Contra-Rotating Small-Sized Cooling Fan

Toru Shigemitsu1*, Katsuhiko Hirosawa2, Keisuke Miyazaki2, Hiroaki Fukuda2

Abstract
High pressure and large flow rate small-sized cooling fans are used for servers in data centers and there is a strong demand to increase its performance because of increase of quantity of heat from servers. Therefore, high rotational speed design is conducted, and rotational speed over 10,000min⁻¹ is employed for cooling fans of servers. Contra-rotating rotors have been adopted for some of high pressure and large flow rate cooling fans to meet the demand. The company's research and development period for the contra-rotating small-sized cooling fan is short and its internal flow condition is not clarified well. Therefore, the internal flow condition was investigated by the numerical analysis.

In the present paper, fan static pressure curves of the contra-rotating small-sized cooling fan with a 40mm square casing are shown by the experimental. Furthermore, the influences of the geometrical shape and design specification of the contra-rotating small-sized cooling fan on the internal flow condition are clarified by the numerical analysis.

Keywords
Cooling fan, Small-sized axial fan, Contra-rotating rotors, Performance, Internal flow

INTRODUCTION
Data center has been built because of spread of cloud computing, establishment of ubiquitous networking society and increase of electric parts in machines. Then, power consumption in data centers, IT devices and machines has been increasing significantly1. Electrical power used for cooling of IT devices for data centers is huge the same as that used for IT devices itself in data centers and the electric power consumption of it is growing rapidly. In the point of view of global warming and energy savings, there is a strong demand for reduction of power consumption in above facilities and equipments2. Axial fans are used as cooling fans of servers and desk top computers. Then, researches of axial fans on fan noise and effect of inlet geometry were conducted3,4. High pressure and large flow rate small-sized cooling fans are used for servers in data centers and there is a strong demand to increase its pressure and flow rate because of increase of quantity of heat from servers. The increase of the pressure and flow rate by increase of a fan diameter is restricted because of limitation of space. Therefore, high rotational speed design is conducted, and rotational speed over 10,000min⁻¹ is employed for cooling fans of servers. Contra-rotating rotors have been adopted for some of high pressure and large flow rate cooling fans to meet the demand. On the other hand, low rotational speed design5 and advantages on the performance of contra-rotating fans and pumps were verified by experimental results6,7,8. The company's research and development period for the contra-rotating small-sized cooling fan is short and its internal flow condition is not clarified well. Therefore, the performance of each front and rear rotor and internal flow condition were investigated by the numerical analysis with the simple numerical model without the spokes9,10. In the case of the contra-rotating rotors, it is necessary to design the rear rotor considering the unsteady circumferential velocity distributions at the outlet of the front rotor11. Further the fan noise becomes larger than the conventional rotor stator type fan because of the interaction between the front and rear rotors. Then, the passive noise reductions with the perforated blade was proposed for the contra-rotating fan12. It is important to clarify the influence of the wake from the front rotor to the rear rotor and the potential interaction between the front and rear rotors to increase the performance and to reduce the fan noise13. The blade row distance between front and rear rotors is a key parameter to consider the wake and potential interaction for the contra-rotating fan. The influence of the blade row distance between the front and rear rotors and pressure fluctuation on the casing wall were investigated for the counter rotating fan with fan diameter D=375mm8,14. On the other hand, the conventional design method and the theory for the turbo machinery should be modified for small-sized axial fans because small-sized axial fans applied to electrical devices belong to extremely small size field in the turbo machinery15. Therefore, there is the strong demand to establish the design method for small-sized axial fans based on the internal flow between the front and rear rotors. Furthermore, the performance characteristics of each front and rear rotor and internal flow condition of the contra-rotating small-sized cooling fan having the spokes are not clarified well. Therefore, a design method of the contra-rotating small-sized cooling fan is not established.

In the present paper, the performance characteristics of each front and rear rotor of contra-rotating small-sized axial
fan with 40mm square casing are shown by the experimental and numerical results. Then, the internal flow condition of the contra-rotating small-sized cooling fan having spokes are discussed based on the numerical analysis results.

1. EXPERIMENTAL APPARATUS AND METHOD

A picture and primary dimensions of the high pressure and large flow rate contra-rotating small-sized cooling fan (R40W-A) are shown in Fig.1 and Table 1 respectively. The rotors of R40W-A are set in a 40mm square casing, and the hub tip ratio of the front and rear rotors are $D_{h}/D_{r}=25\text{mm}/37.2\text{mm}=0.67$ and $D_{h}/D_{r}=28\text{mm}/37.2\text{mm}=0.70$ respectively. Operating flow rate is $Q_{o}=0.55\text{m}^3/\text{min}$ and a tip clearance is $c=0.6\text{mm}$. Each rotational speed of the front and rear rotors of R40W-A is extremely high as $N=15000\text{min}^{-1}$ and $N=14000\text{min}^{-1}$. Figure 2 shows a schematic diagram of an experimental apparatus. The experimental apparatus was designed based on the Japanese Industrial Standard and air blown in a test section passes the rotors, a chamber, a measurement duct and a booster fan and blow out in the ambient atmosphere. Each rotor is driven by a brushless motor set inside of the hub and the motor is supported by spokes connected to the casing. The rotational speed of each rotor was kept constant ($N=15000\text{min}^{-1}$, $N=14000\text{min}^{-1}$) by PWM control when a performance test was conducted. The fan static pressure ($\Delta P$) is measured by a pressure difference between static holes downstream of the rotor installed at the chamber and ambient air. Further, the rotational speed was evaluated using a pulse of the motor measured by an oscilloscope. Flow rates were measured by an orifice meter set at the measurement duct and the pressure curve from a cutoff flow rate to a large flow rate was investigated in the experiment. There are three spokes downstream of the front and the rear rotors to support the motors and the casing has curved corners at the inlet and outlet of it and circular diffuser and nozzle between the front and rear rotor as can be confirmed by a sectional view of R40W-A in Fig.3. Therefore, there are some geometrical restrictions for the high pressure and large flow rate small-sized cooling fan.

2. NUMERICAL ANALYSIS CONDITIONS

Commercial software ANSYS-CFX16.2 was used to investigate the flow condition which couldn't be measured by the experiment. Three dimensional unsteady numerical analysis was conducted because the complicated structure existed in this cooling fan. Figure 4 shows the numerical model. In the numerical model, a simplification of the geometry was conducted by extension of the length of the front rotor hub to the rear rotor inlet as shown in Fig.4 in order to reduce the computational cost. Numerical grids used for the numerical analysis are shown in Fig.5. The numerical domains comprise inlet, front rotor, spoke, rear rotor, chamber and outlet duct regions. The numerical grid points are 617,090 for the inlet region, 1,479,336 for the chamber region and 237,628 for the outlet duct region respectively. The numerical grid points are 3,924,468, 362,848 and 3,107,155 for the front rotor, spoke and rear rotor regions respectively. The tip clearance was kept 0.6mm as the same with the experimental apparatus in the numerical analysis. At an inlet boundary, constant flow rate was given and constant pressure was given as an outlet boundary condition. The coupling between the front and rear rotors was accomplished by the transient rotor stator. The scalable wall function was used as a near wall treatment and the LRR Reynolds Stress
model, which provides high accuracy for some complex flows, was used as a turbulence model. The unsteady numerical flow analysis was conducted at the operating flow rate \( Q_o = 0.55 \text{m}^3/\text{min} \) and 4 other flow rate points \( 0.4Q_o, 0.7Q_o, 1.3Q_o \) and \( 1.6Q_o \). A time step number per one rotor rotation was 140 and the time step was \( t = 2.857 \times 10^{-5} \) s. The data of one rotor rotation were obtained after 10 rotor rotations in the unsteady numerical analysis and the computational time for 10 rotor rotations at each flow rate was about 1 week. The fan static pressure for the numerical analysis was calculated by the static pressure difference between the static holes positions installed at the chamber and the inlet boundary of the numerical domain, which was almost the same with the experiment. The air was assumed incompressive fluid, so the density of the air was constant in this research. The static pressure efficiency was evaluated by a ratio of a multiplication of the static pressure and the flow rate to the shaft power obtained by the numerical analysis.

3. EXPERIMENTAL AND NUMERICAL ANALYSIS RESULTS

3.1 Performance curve of R40W-A

Figure 6 shows the fan static pressure curves of the test fan at the design rotational speed \( N_f/N_r = 15000 \text{min}^{-1}/14000\text{min}^{-1} \) obtained by the experiment. Numerical analysis results are also shown in Fig.6. A horizontal axis is the flow rate \( Q \) and a vertical axis is the fan static pressure \( \Delta P_s \). The fan static pressure of each front and rear rotor obtained by the numerical analysis, in which the static pressure difference in the spoke region is excluded, is also shown in Fig.6. The test fan showed the negative slope of the pressure curve from the operating flow rate \( Q_o = 0.55 \text{ m}^3/\text{min} \) to the maximum flow rate \( Q = 1.08 \text{ m}^3/\text{min} \) and the maximum fan static pressure \( \Delta P_s = 510 \text{ Pa} \) was obtained at the operating flow rate \( Q_o = 0.55 \text{ m}^3/\text{min} \) in the experiment. The maximum fan static pressure of the numerical analysis was also obtained at the operating flow rate \( Q_o = 0.55 \text{ m}^3/\text{min} \) and its value was \( \Delta P_s = 500 \text{ Pa} \), which was within the 2% accuracy of the experiment. A positive slope of the pressure curve appeared from the partial flow rate \( Q = 0.50 \text{ m}^3/\text{min} \) to the operating flow rate \( Q_o = 0.55 \text{ m}^3/\text{min} \). After that, the pressure curve became flat in partial flow rates region \( Q = 0.35-0.50 \text{ m}^3/\text{min} \). It was confirmed that the operating conditions of the test fan were in severe condition as the fan static pressure was set extremely high and the design flow rate existed near the flow rate range with the positive slope of the fan static pressure curve. The whole of the numerical data accorded with the experimental data and the trend of the fan static pressure curve of the experiment could be captured well by the numerical analysis quantitatively. Then, the performance of each front and rear rotor, which could not be obtained by the experimental method, was investigated by the numerical analysis. The fan static pressure of the front rotor was extremely lower than that of the rear rotor and the fan static pressure decreased according the increase of the flow rate. On the other hand, the fan static pressure of the rear rotor was high and about 77% of the total fan static pressure was obtained by the rear rotor at the operating flow rate \( Q_o = 0.55 \text{ m}^3/\text{min} \). The fan static pressure of the rear...
rotor showed the maximum value at Q_o=0.55m³/min⁻¹, where the maximum total fan static pressure obtained. The static pressure efficiency of each front rotor, rear rotor and its total including static pressure difference in spoke region are shown in Fig.7. The static pressure efficiency was obtained by the unsteady numerical analysis. A horizontal axis is the flow rate Q and a vertical axis is the static pressure efficiency η_s. The maximum total fan static pressure obtained at the operating flow rate Q_o=0.55m³/min⁻¹ was low as η_s=46.7% because of the design specification of high pressure and large flow rate cooling fan with the 40mm square casing. The fan static pressure of the rear rotor was higher than that of the front rotor in all flow rates, where numerical analysis conducted and the difference of the static pressure efficiency between the front and rear rotors became large with the increase of the flow rate. It was found from the performance curve of the test fan that the static pressure increased significantly in the rear rotor and its tendency became large with the increase of the flow rate. Therefore, the internal flow condition of each front and rear rotor was investigated using the numerical analysis results to clarify the difference of each front and rear rotor performance.

### 3.2 Internal flow condition of R40W-A at operating flow rate Q_o

The internal flow conditions at the operating flow rate Q_o=0.55m³/min were investigated by the numerical analysis results. Blade-to-blade relative velocity vectors and static pressure distribution at each radial positions $r/r_c=0.74$, $r/r_c=0.85$ and $r/r_c=0.96$ are shown in Fig.8 and Fig.9. The flow rate is the operating flow rate Q_o. r and r_c means radius, where the data obtained, and inner radius of the casing. It was observed from the internal flow near the hub in Fig.8(a) and Fig.9(a) that the stagnation point at the leading edge of the front and rear rotors existed near the suction surface of the blade at the operating flow rate Q_o=0.55m³/min and the small separation occurred on the suction surface of the rear rotor. On the other hand, the relative velocity of the suction surface of the rear rotor was high because of the circumferential velocity at the outlet of the front rotor and decreased suddenly at the outlet of the rear rotor. Therefore, the static pressure increased significantly in the rear rotor, although the static pressure increase of the front rotor was small. These flow condition was similar to those at other radial positions in Fig.8(b),(c) and Fig.9(b),(c) and corresponded to the result that the fan static pressure of the rear rotor was large compared to the front rotor in Fig.6. The leakage flow from the pressure surface to the suction surface near the mid of the blade chord were confirmed for both front and rear rotors at shroud region in Fig.8(c). The meridional velocity vectors and static pressure on the vertical plane at the operating flow rate Q_o=0.55m³/min are shown in Fig.10 and Fig.11. The rotational direction of the front rotor is front side of the paper and that of the rear rotor is back side of the paper. The vortex occurred at the inlet corner curve and this vortex could be caused by the separation of the main flow around the inlet corner. In general, the corner separation occurred at the inlet of the casing and this separation vortex influenced on the inlet flow condition. In this test fan, this vortex was basically got stuck in the inlet corner, so the flow condition near the inlet shroud wasn’t influenced by this.
vortex. The leakage flow from the blade tip was observed for both front and rear rotors in Fig.10 and these low velocity regions related to the leakage flow was observed in wide region near the shroud on the meridional plane. There is the circular diffuser and nozzle region on the shroud between the front and rear rotors to improve the performance and noise. The back flow occurred in the diffuser region and the static pressure was high in this region. The low velocity region spread radially inner section between the front and rear rotors by the influence of the back flow region. Then, the main flow inclined to radial mid and hub region between the front and rear rotors. The back flow region disappeared in the circular nozzle region with the increase of the velocity and uniform flow was achieved at the inlet of the rear rotor. The static pressure increased significantly in the rear rotor and its distribution was uniform in radial direction as could be seen in Fig.11.

4. CONCLUDING REMARKS

The performance characteristics of the high pressure and large flow rate small-sized contra-rotating axial flow fan were investigated by the experiment and the numerical analysis. Then, the internal flow conditions at the operating flow rate were clarified by the numerical analysis results. As a result, following concluding remarks were obtained.

1. The fan static pressure of the front rotor was extremely lower than that of the rear rotor and the fan static pressure of the front rotor decreased according the increase of the flow rate. On the other hand, the fan static pressure of the rear rotor was high and about 77% of the total fan static pressure was obtained by the rear rotor at the operating flow rate \( Q_o = 0.55 \text{m}^3/\text{min}^{-1} \).

2. The maximum total fan static pressure obtained at the operating flow rate \( Q_o = 0.55 \text{m}^3/\text{min}^{-1} \) was low as \( \eta_s = 46.7\% \) because of the design specification of high pressure and large flow rate cooling fan with the 40mm square casing. The fan static pressure of the rear rotor was higher than that of the front rotor in all flow rates.

3. The stagnation point at the leading edge of the front and rear rotors existed near the suction surface of the blade at the operating flow rate \( Q_o = 0.55 \text{m}^3/\text{min}^{-1} \). On the other hand, the relative velocity of the suction surface of the rear rotor was high because of the circumferential velocity at the outlet of the front rotor and decreased suddenly at the outlet of the rear rotor. Therefore, the static pressure increased significantly in the rear rotor, although the static pressure increase of the front rotor was small.
4. In general, the corner separation occurred at the inlet of the casing and this separation vortex influenced on the inlet flow condition. In this test fan, this vortex was basically stuck in the inlet corner, so the flow condition near the inlet shroud wasn’t influenced by this vortex.

5. There is the circular diffuser and nozzle region on the shroud between the front and rear rotors. The back flow occurred in the diffuser region and the static pressure was high in this region. The low velocity region spread radially inner section between the front and rear rotors by the influence of the back flow region. Then, the main flow inclined to radial mid and hub region between the front and rear rotors. The back flow region disappeared in the circular nozzle region with the increase of the velocity and uniform flow was achieved at the inlet of the rear rotor. The static pressure increased significantly in the rear rotor and its distribution was uniform in radial direction.

ACKNOWLEDGMENTS

The authors wish to show our special thanks to the supports by Nidec Corporation, the project research aid from Tokushima University, Japan Science and Technology Agency and Komiya research aid.

REFERENCES


