Flows in Duct for Axial Fan

Jaewon KIM

Department of Mechanical Engineering, Sunmoon University

100, Kalsan, Asan City, Chung Nam 336-704

SOUTH KOREA

http://fluid.sunmoon.ac.kr

and

E. Y. AHN

Department of Multimedia, Hanbat University

16-1, Duckmyung, Yuseong-gu, Daejon 305-719

SOUTH KOREA

http://www.hanbat.ac.kr

Abstract: - Axial flow fan has been used for large amount of flow rate in air handling units of industrial application fields. The fan involves a shrouding duct as flow guidance for uniform flows. Pressure difference between before and ahead of rotating blades causes the flows movements. The case where the flow resistance such as a heat exchanger is located prior to the rotors is typical in-field model for several air conditioning systems. Main interest lies on the looking for an optimal position of axial fan in a circular duct. Experimental apparatus for fan performances and numerical research for detail analysis are adopted in this investigation in parallel. The optimal location of an axial fan is recognized by both no reverse flows and larger amount of flow rate in the down stream. All numerical data are validated by experimental measurements using a wind tunnel.

Key-Words: - Axial fan, CFD, Fan tester, Noise level, Performance, Shroud
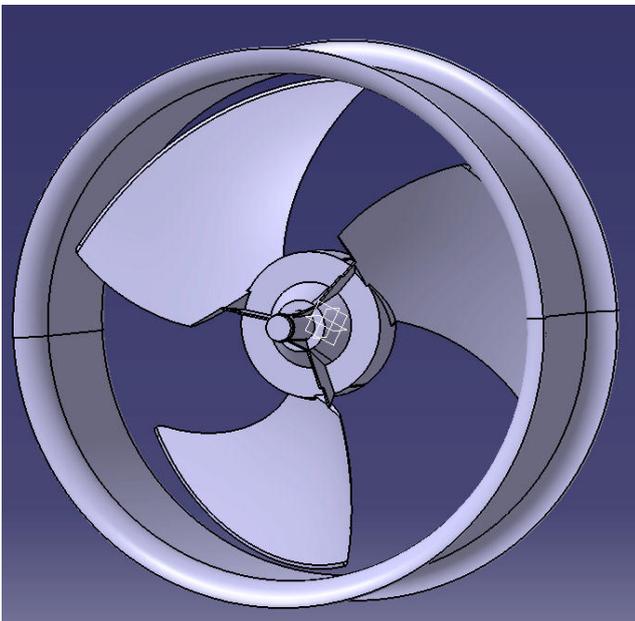
1   Introduction

Fig. 1 Fan model and its circular duct

Axial flow fan has been widely used for large amount of flow rate in air handling units of industrial application system [1~3]. The fan involves a shrouding duct as guidance for uniform flows both in and out flow [4~5]. Rotation of the blades of an axial fan causes pressure difference between before and ahead of it and the pressure difference generates axial flows parallel with the axis of the fan. However, the rotation effects of the blade are pronounced near the tip and those make the flows move toward the radial direction and even for reverse in axial direction. According to Maaloum et al. [6], the upstream turbulence affects the flow instability concerned with flow rates. The presence of duct in axial fan system is recognized as another sources of noise [7~8]. The prior works dealt with the upstream flows due to the installation of a circular duct. However, simultaneous consideration of the both flow patterns of upstream and down stream is significant for the improvements of fluid dynamic performance of an axial blowing system.

In this investigation, the effects of in-duct system are examined using numerical and experimental approach. Figure 1 shows the flow model configuration with a circular duct involving an axial fan whose blades are three in circumferential direction. The model used to flow out from the series of heat exchanger in a heat pump.

The numerical investigation has been carried out using a commercial program (SCREW-TETRA) with pre and post processor [9]. Numerical data are compared with experimental estimations using a wind tunnel connected with an anechoic chamber for noise evaluation. Parallel experimental measurements tune the conditions and fluid and flow properties of the numerical prediction.

The position of model fan in a duct is proposed by different axial locations that come from the in-field data in practical application field.
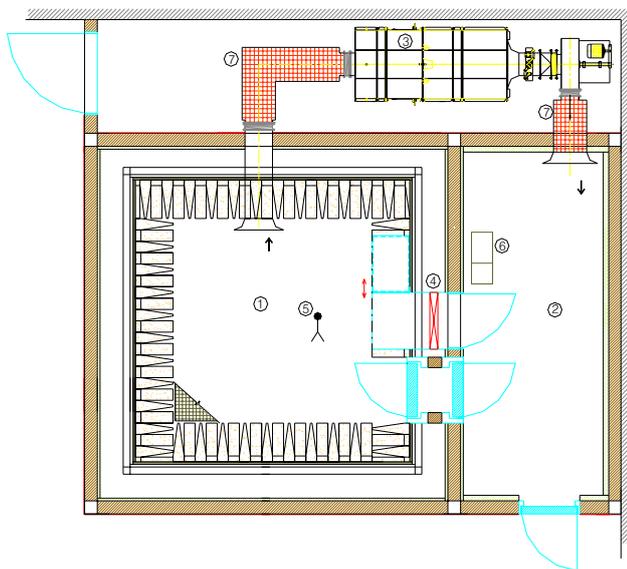
2   Experimental Approach

Fig. 2 Wind tunnel (③) connected to anechoic chamber (①) with microphone (⑤), silence (⑦) and test unit (④)

In order to evaluate the performance of the present system, well-constructed wind tunnel is used. The experimental rig involves sensors for measurements and data acquisition unit connected to personal computer. The present wind tunnel follows ASHRAE standards [10] for accurate obtained data and can measure static pressure in plenum chamber of the wind tunnel and flow rates by manipulating of nozzles of the fan tester. Wind tunnel system for capacity of flow rate and static pressure is composed of discharging blower with a butterfly valve for adjustments of flow rate in the tester and rectangular duct. Schematic diagram of the wind tunnel and fan system is shown in Figure 2. In the apparatus, static pressure is measured at four points in peripheral position and averaged. Pressure difference ahead and behind the nozzles is also found for calculating the flow rates. The range for flow rate covers up to 80

CMM (Cubic Meter per Minute). Driving motor for the test fan is connected with torque meter for estimation of total efficiency of the fan. In addition, the entrance of the wind tunnel is connected to a full anechoic chamber for simultaneous measurements of noise level of the unit. With this combined system, it is able to estimate the aerodynamic performance and noise level of the unit, simultaneously.

Performance test of a fan is looking for accurate data on flow rate, static pressure, rise, and efficiency for examination of the approximate results at different operating conditions of the flow system. Fluid dynamic performance is displayed in terms of flow rate and static pressure rise by the rotating fan. In general, pressure behind a rotating fan is approximately equal to atmospheric one (upstream pressure). The pressure rise is obtained by just measuring pressure in front of the fan. In order to evaluate the pressure rise according to flow rate, an automatic controlled motor system is adopter for adjustment of damper (practically butterfly valve).

Four microphones and FFT analyzer are utilized for noise analysis for the model fan. Test code for noise level estimation is applied by the standard regulation prepared by ISO/TC117 [11]. Background noise level of the chamber is 17dB(A) and its cut-off frequency is 125Hz.

The axial location of fan in a duct is prepared from the practical field conditions. Table 1 summarizes the dimensional information of the model fans, including diameter and impeller width.

Table 1 Specification of fan for test

Fan Diameter(mm)		725
Hub Diameter(mm)		225
Blade	Inlet angle (degree)	17
	Outlet angle(degree)	27
	Number	7
Rotational Direction		CW

Uncertainty of the experimental measurements is evaluated for flow rate, pressure rise, and efficiency. Flow variables for flow rate and pressure rise utilize electric signal from pressure transducer whose

accuracy is equal to ±0.1%. Torque for the efficiency estimation receives a voltage signal in the accuracy of ±0.05%. Therefore, whole uncertainty level of the flow rate is in the range of ± 0.04658% to ±0.10691% because of coefficient of flow rate for the present venturi meter in the wind tunnel, and for efficiency, it is between ±0.07% and ±0.11%.

3   Numerical Predictions

Details of flows around the impeller are recognized by numerical predictions based on transient Navier-Stokes equations with boundary and initial conditions. A commercial program, SCRYU-TETRA, is used for this calculation. The code has performed numerical integration in finite volumes cover the whole domain. Figure 3 shows the flow geometry and install location of an impeller, axial width L, with respect to the concentrate circular duct, in axial length D. The model number depicted in Fig. 3 illustrates the relative axial position of a fan from H=0 to H=D+L.

#1:H=L
#2:H=L/2
#3:H=0
#4:H=D
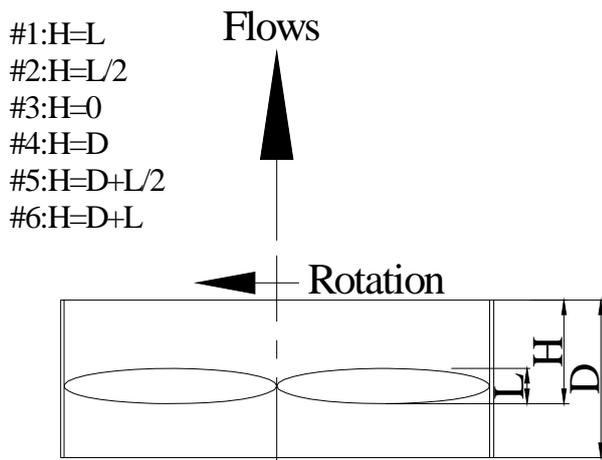#5:H=D+L/2
#6:H=D+L

Flows

Rotation

Fig. 3 Axial position of impeller in duct

The number of finite volumes shaped polyhedral is equal to 1091228 and among them; rotating part divides into 261363 cells and 829865 for stationary part. For flow consistency, the calculation domain is widely considered 12 times of fan diameter in up and down stream direction, respectively. Boundary conditions are no-slip condition and angular momentum imposed on the rotating fan.

Fluid dynamic performances such as flow rates and static pressure rise are obtained and details of flows near the rotor explain the macroscopic performances.

4   Results and Discussions

Fluid dynamic performance and noise level are measured by an anechoic chamber connected with wind tunnel. Figure 4 and 5 display engineering performances and noise level for model fan #4, respectively. The reason for choosing the reference model as #4 is due to the installation is widely used in practical applications. Maximum efficiency appears in $\Phi \doteqdot 0.2$ and flow rate reversely proportional to static pressure rise. No surging found in the performance chart. In of noise level measurements, the positions of micro phones are 1m away from the fan, i.e., front, back, right, left and top of the system, respectively. The flow rate is validated by velocity, measured by hot wire anemometer, and cross sectional area. Table 2 compares the flow rates according to the model fan. As shown in table 2, maximum flow rates, in unit of cubic meter per second, are found in model fan #6, because the inflows towards the fan comes from the larger region and the outflows are guided by the duct.

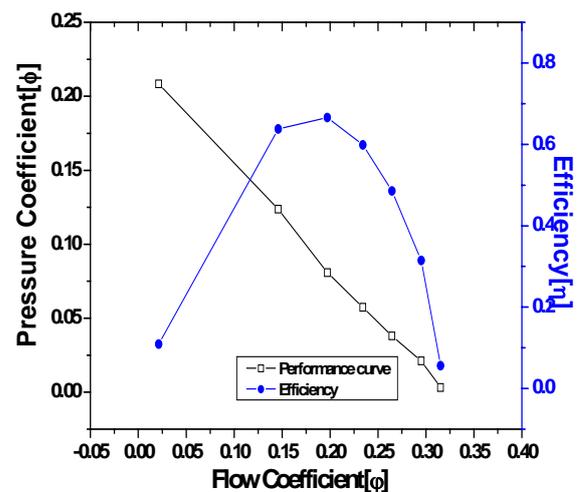| Table 2 Comparison of flow rate	
	Volume flow rate(CMS)
#1	0.942676
#2	0.951229
#3	0.945622
#4	0.770594
#5	0.895982
#6	1.04663

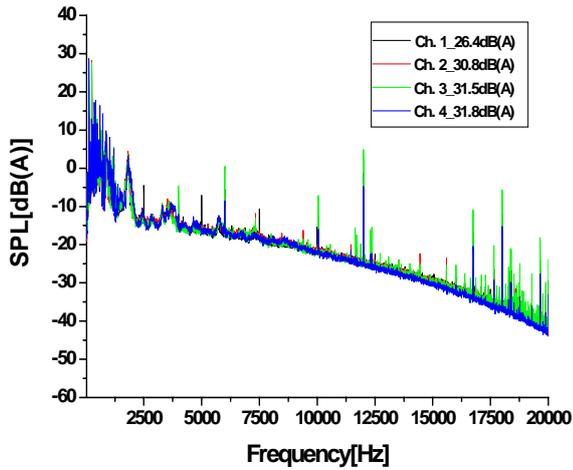**Fig. 4 Fluid dynamic performance of #4 model**

Fig. 5 Noise level of #4 model fan

Details of flow patterns according to the axial position of the fan in a duct are plotted in Fig. 6. As the fan insert in the duct, the unwanted flows such as reverse flows and circulating flows are found after a fan. Flows behind and after the rotor are stable and uniform in the case of model fan #6, because of larger inflow region. The present results explain the duct is effectively available for the guidance of out flows and restricted wall for inflow region.
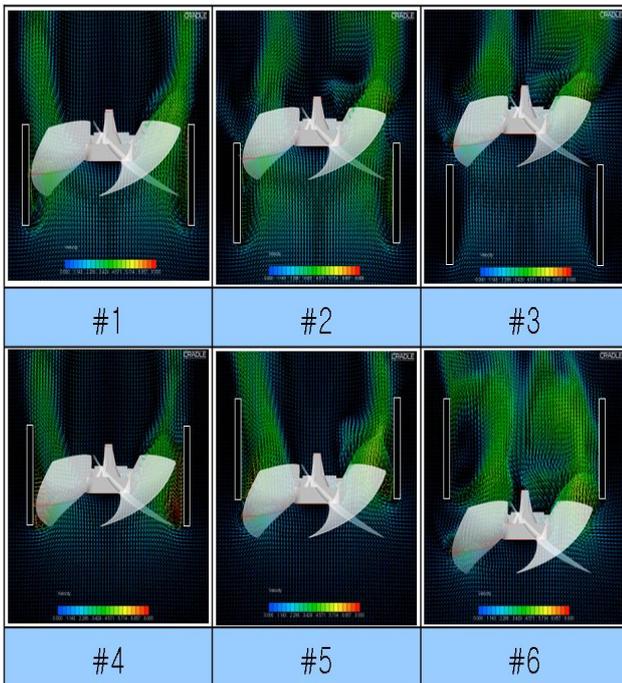

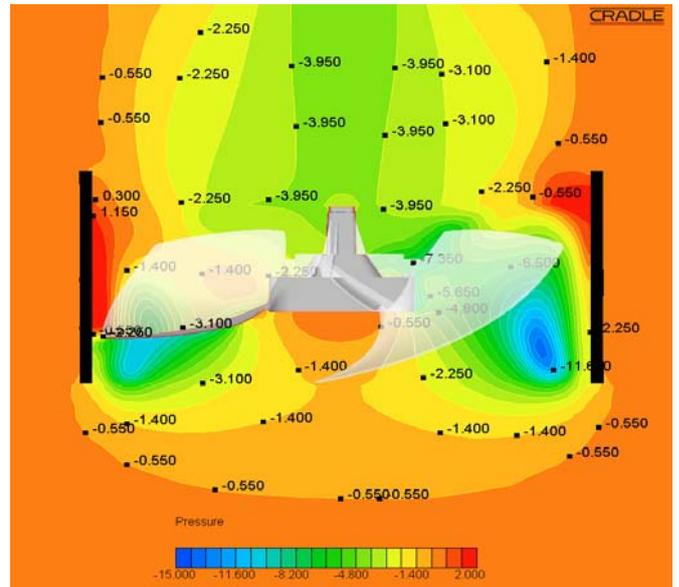**Fig. 6 Comparison of flow details for different model fan**

Fig. 7 Pressure distribution around impeller for model #4

Figure 7 illustrates the pressure distribution in the calculation domain. Higher level region for pressure is found near the wall and those physics causes the reverse flows at that region. However, pressure distribution for model fan #6 shows no reverse flow region. Figure 8 is for model fan #6.
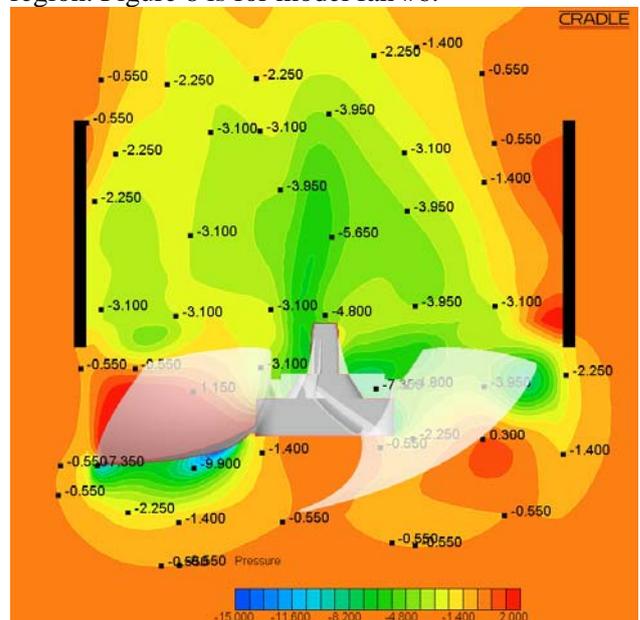

Fig. 8 Same as Fig. 7 except for model #6

4   Conclusion

Plots of flow details are prepared for the different axial conditions of impeller in a circular duct. Consequently,

the duct is useful for outflow guidance. However, the duct disturbs and restricts the inflow. In addition, the flow rates are decreased 26% of the maximum one for the model #4.

ACKNOWLEDGEMENTS

Reference:
[1] Eck, B., "Fans," 1973, Pergamo*n Press, New York.*
[2] Maaloum, A.; Kouidri, S.; Bakir, F.; Rey, R.., 1997, "Effect of inlet duct contour and lack thereof on the noise generated of an axial flow fan," ASME J. Fluids Engineering, Vol. 119, pp. 633~646.
[3] Lazzaretto, A., Macor, A., Martegani, A. D., and Martina, V., 1997, "Experimental Optimization of the Casing of a Cross Flow Fan.(In Italian), " Proceeding of 52° Convegno Nazionale ATI, Cernobbio (co),22-26 September, SGEditoriali-Padova.
[4] Matsuki, K., Shinobu, Y., and Takushima, A., 1988, "Experimental Study of Internal Flow of a Room Air Conditioner Incorporating a Cross-flow Fan," ASHRAE Transactions, Vol. 94, pp. 450~364.
[5] Murata, S. I., and Nishihara, K., 1976a, "An Experimental Study of Cross Flow-1st Report, Effects of Housing Geometry on the Fan Performance," Bull. JSME, Vol. 19, No. 129, pp. 461~471.
[6] Murata, S. I., and Nishihara, K., 1976, "An Experimental Study of Cross Flow – 2nd Report, Movements of Eccentric Vortex inside Impeller," Bull. JSME, Vol. 19, No. 129, pp. 472~491.
[7] Porter, A. M. and Markland, E., 1970, "A Study of the Cross Flow Fan," J. Mechanical Engineering Science, Vol. 12, pp. 421~431.
[8] Martegani, A.D, Navarro, G., Macor, A., Lazzaretto, A., Masi, M., Antonello, M., and De Lorenzi, S., 1999, "Experimental and numerical analyses of a cross flow fan(In Italian), "Experimental and numerical analyses of a cross flow fan(In Italian),"Proceeding of 54° Congresso Nazionale ATI, Vol. 2, pp 1409-1423, L'aquila, 14-17 September. SGEditoriali-Padova.
[9] ASHRAE Standard, 1985, "Laboratory Methods of Testing Fans for Rating," American Society of Heating, Refrigerating and Air-Conditioning Engineers.
[10] Combes, J. F. and Marie, L., 1994, "Numerical Modeling of the Flows in a Cross-Flow Fan," Proceedings of the ASME Fluids Engineering (Lake Tahoe), pp. 29~34.