DEVELOPMENT OF HIGH EFFICIENCY FAN SYSTEM FOR OUTDOOR UNIT OF AIR-CONDITIONER

Taku IWASE[1], Erika KATAYAMA[1],
Sachio SEKIYA[1], Tetsushi KISHITANI[2]

[1] Hitachi, Ltd., Research & Development Group, 832-2, Horiguchi, Hitachinaka, Ibaraki, Japan

[2] Hitachi-Johnson Controls Air Conditioning, Inc., 390, Muramatsu, Shimizu-ku, Shizuoka, Japan

SUMMARY

Recently, it is important to consider influence of equipped condition in the development of fan system. We therefore estimated flow rate and fan shaft power considering the condition equipped with the fan systems in the air-conditioner by computational fluid dynamics (CFD). High efficiency fan system for outdoor unit was then developed by CFD and experiment. Three ways for development of high efficiency fan system; improvement of bell mouth shape, increase of number of fan and improvement of heat exchanger shape, were investigated. Actually, we employed the long bell mouth, 2 fan type, and sigma shaped heat exchanger. As a result, we confirmed that the fan shaft power of the developed outdoor unit decreased by 50 %.

INTRODUCTION

Developing high efficiency air-conditioners is currently a pressing demand for environmental conservation and energy saving. Fan power consumption comprises a large percentage of the total energy consumption of an air-conditioner. High efficiency fan systems therefore directly contribute to energy saving of the air-conditioner. We had developed fan systems by using the method of optimization of propeller shapes independently [1]. Another researchers also developed design methods and blade shapes [2][3]. However, the air-conditioner is a product that equips with components which are propeller, bell mouth, motor and heat exchanger etc. in a unit. It is important to consider influence of equipped condition.

We therefore estimated flow rate and fan shaft power considering the condition equipped with the fan systems in the air-conditioner by computational fluid dynamics (CFD). The goal of this study is the development of high efficiency fan system for air-conditioner. We proposed the new fan system that were improved long bell mouth, 2 fan type and sigma shaped heat exchanger by using CFD and experiment.

METHOD OF NUMERICAL SIMULATION

Figure 1 shows computational model. We can consider an influence of components which are propeller, bell mouth, motor and heat exchanger etc. The computational model consisted of doom shaped region, region of the outdoor unit and rotating region around propeller. The diameter of the doom shaped region was 12,880 mm. This was 20 times of propeller diameter of old type outdoor unit and was large enough compared to the outdoor unit.

Figure 2 shows computational grids. The grids were generated by using ANSYS ICEM CFD Ver.14.0. The grids were composed of tetra and prism elements. The prism elements are set on blade surfaces. The rotating region was in the rotating frame of reference. The other regions were in the stationary frames. The numbers of elements were approximately 25 million grids in 1 fan type outdoor unit and 35 million grids in 2 fan type. The numbers of elements has influence in calculated result. We then surveyed the dependence of results from 3 to 25 million grids in 1 fan type outdoor unit. Accordingly, we confirmed that the accuracy of calculated flow rate was improved by the 25 million grids. Moreover, y+ plus values on blade surfaces of the propeller were smaller than approximately 20 in the 25 million grids. We therefore applied the grids of the 25 million grids in this study. The 35 million grids in 2 fan type outdoor unit were generated by the same specification of the 25 million grids.

The numerical simulation code employed an incompressible Reynolds-averaged Navier-Stokes simulation (RANS) solver (ANSYS CFX Ver.14.0). It also employed a k-epsilon turbulence model and wall function. Residual type was root mean squares of momentum and mass. The residual target was 1.0×10^{-4} in convergence criterion. Calculations were run by 8 cores parallel computing.

Heat exchanger was simulated by porous model. The porous model is standard function of ANSYS CFX. Pressure loss of porous model is defined by equation (1).

$$\Delta p_i = C_{R1} u_i + C_{R2} |u| u_i \tag{1}$$

Here, Δp is pressure loss per unit length. C_{R1} and C_{R2} are linear and quadratic resistance coefficients. u is velocity. Subscript i shows x, y, z directions respectively. The resistance coefficients were measured by element experiment of heat exchanger.

We compared between experimental and calculated flow rate. We confirmed that accuracy of calculated flow rate was 7 % in the old outdoor unit in Fig.1 by the 25 million grids and the porous model.

Figure 3 shows examples of calculated results in the old outdoor unit. We can predict flow rate and fan shaft power. Moreover, we can analyze total pressure loss in the outdoor unit by investigating calculated results.

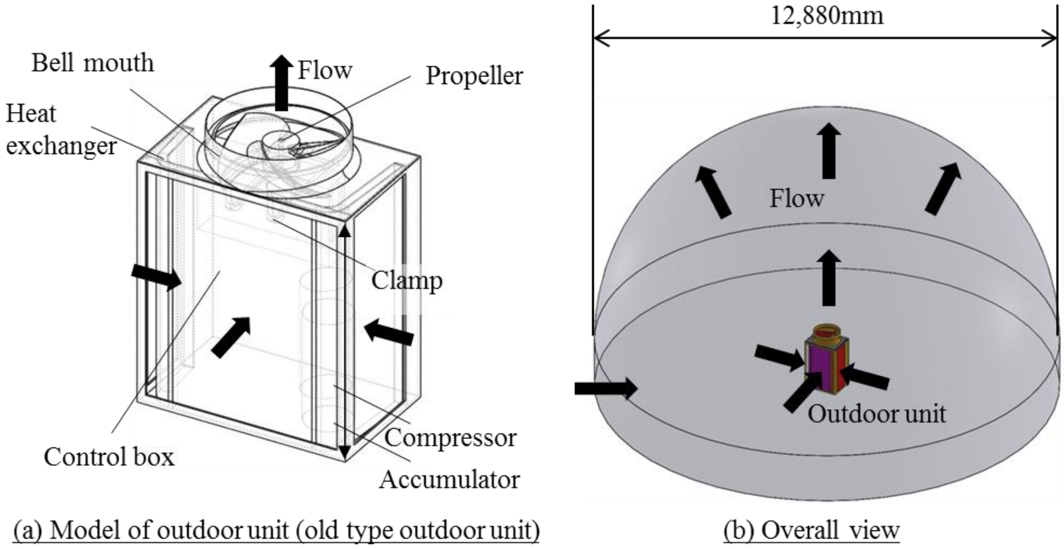


Figure 1: computational model

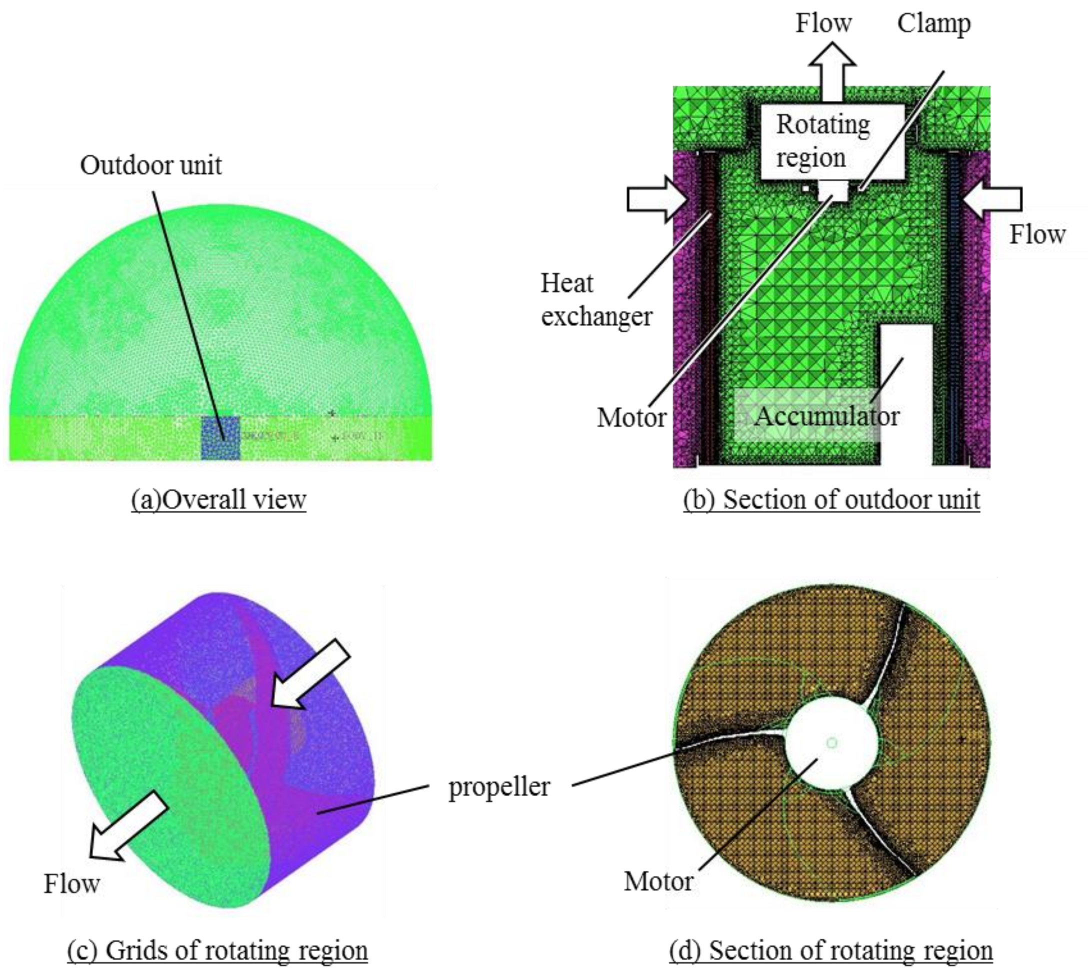
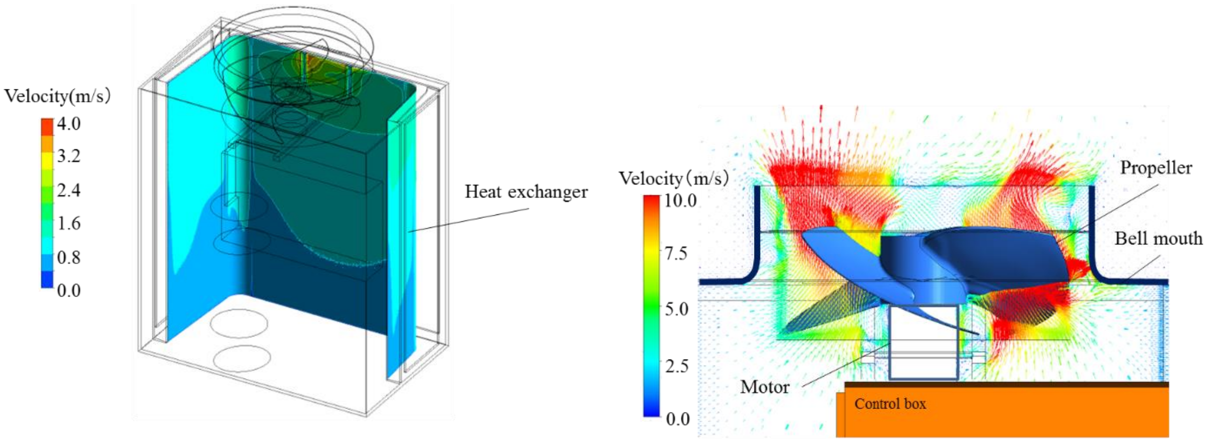


Figure 2: computational grids



(a) Velocity distribution on inner surface of heat exchanger

(b) Velocity vectors around propeller

Figure 3: example of calculated results

DEVELOPMENT OF OUTDOOR UNIT

Analysis of total pressure loss of old outdoor unit

Figure 4 shows analysis results of total pressure loss in the old outdoor unit. We evaluated the total pressure loss of each component. The total pressure losses were calculated as difference of total pressure between inlet and outlet section of each components. For example, the loss of heat exchanger was difference of total pressure between inlet and outlet of heat exchanger. "100 %" in Figure 4 denotes Euler head of the fan. The Euler head was defined by equation (2).

$$\Delta P_{Euler} = L/Q \qquad (2)$$

Here, ΔP_{Euler} is Euler head of the fan. L is fan shaft power of propeller. Q is flow rate of the fan system. The inner loss of fan was difference between Euler head and summation of outlet loss of fan, loss of motor, control box etc. and heat exchanger. The inner loss of fan was comprised of blade loss, tip clearance loss etc. The outlet loss of fan was comprised of dynamic pressure at the bell mouth outlet, mixing loss, loss by non-uniform velocity distribution at propeller outlet etc. The outlet loss was the largest. The inner loss was second and the loss of heat exchanger was third.

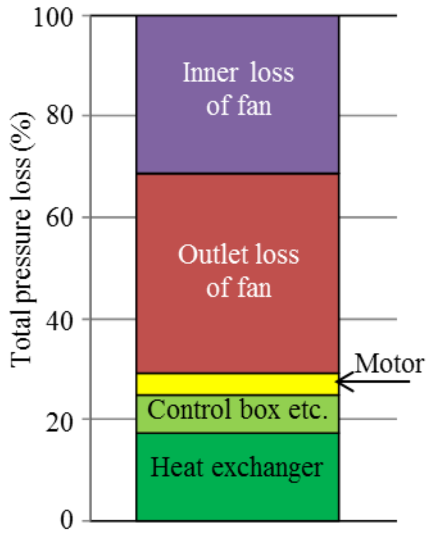


Figure 4: analysis results of total pressure loss of old outdoor unit

Three ways for development of high efficiency fan system

We can propose the three ways for development of high efficiency fan system in the outdoor unit. Actually, we investigated improvement of bell mouth shape, increase of number of fan, and improvement of heat exchanger shape. Figure 5 shows old type outdoor unit. Figure 6 also shows developed outdoor unit.

The improvement of bell mouth aims decrease of the outlet loss by uniformizing velocity distribution of propeller outlet. It also aims decrease of the inner loss by optimizing velocity distribution of propeller inlet and improving blade efficiency. We confirmed that optimization of bell mouth length had great influence of the fan efficiency [4]. We therefore employed improved long bell mouth compared to the old outdoor unit. Moreover, tip clearance of the long bell mouth was decreased from 18 mm to 10 mm. The tip clearance of the old outdoor unit was 18 mm. It is clear that the tip clearance is effective parameter in fan efficiency. Figure 7 shows the developed bell mouth and propeller. The propeller was developed individually by CFD and numerical optimization method [1]. As it turned out, the blade of the propeller was corresponding shape by half forced vortex design.

The increase of number of fans aims decrease of the outlet loss by increasing area of propeller outlet. It also aims decrease of inner loss by changing of high efficiency flow rate. That is to say, flow rate coefficient moved from low efficiency to high efficiency operating point. Actually, we change number of fan from 1 fan type (propeller diameter 644 mm) to 2 fan type (propeller diameter 544 mm) in this development.

The improvement of heat exchanger shape aims decrease of pressure loss of heat exchanger. Several shapes of heat exchanger were surveyed by the CFD. As a result, we confirmed that sigma shaped heat exchanger could not only decrease pressure loss but also uniformize velocity distribution of propeller inlet.

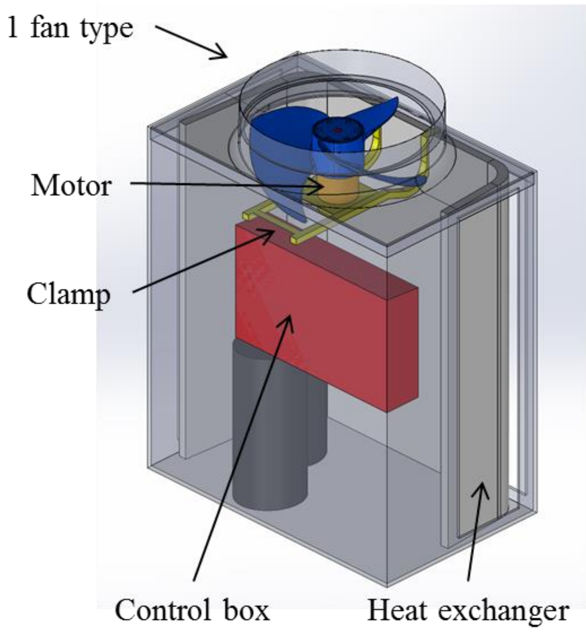


Figure 5: old type outdoor unit

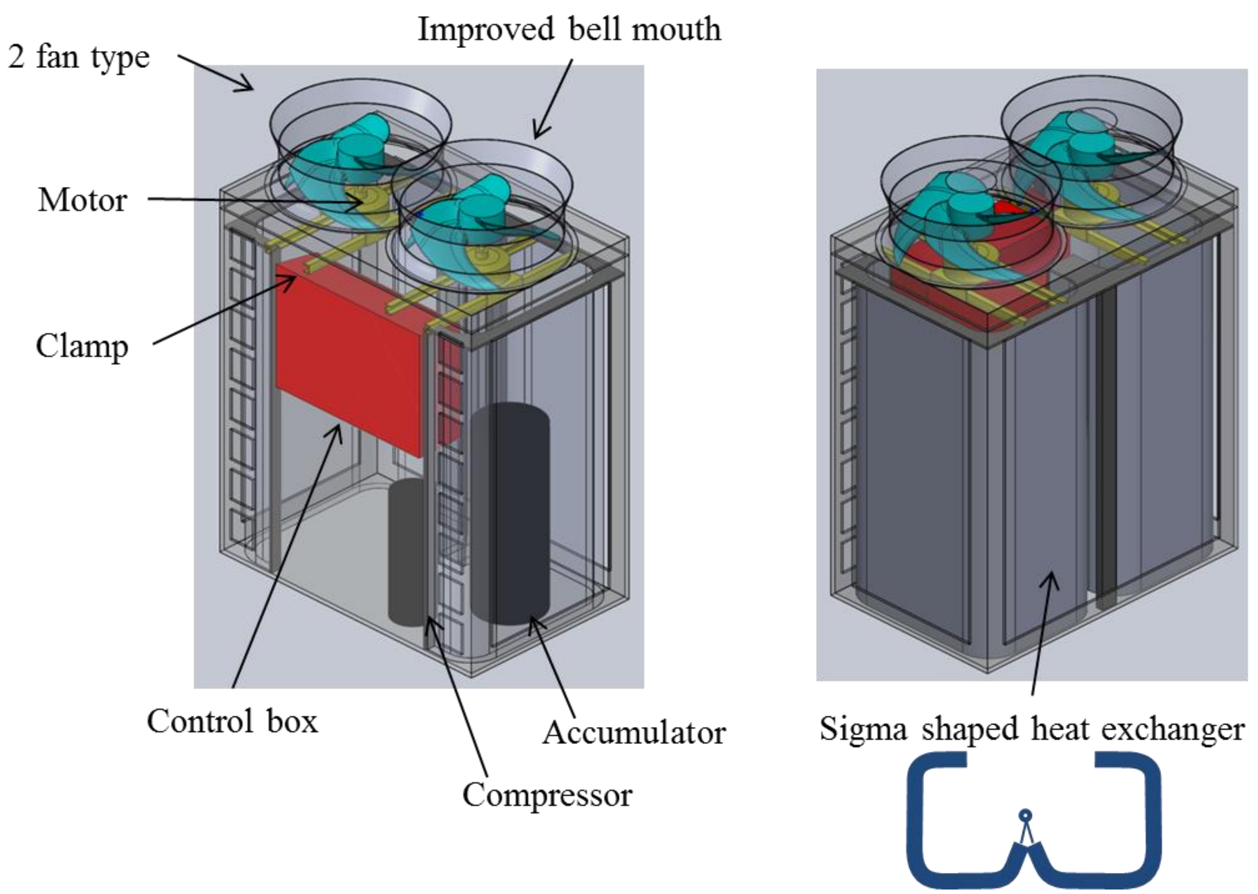
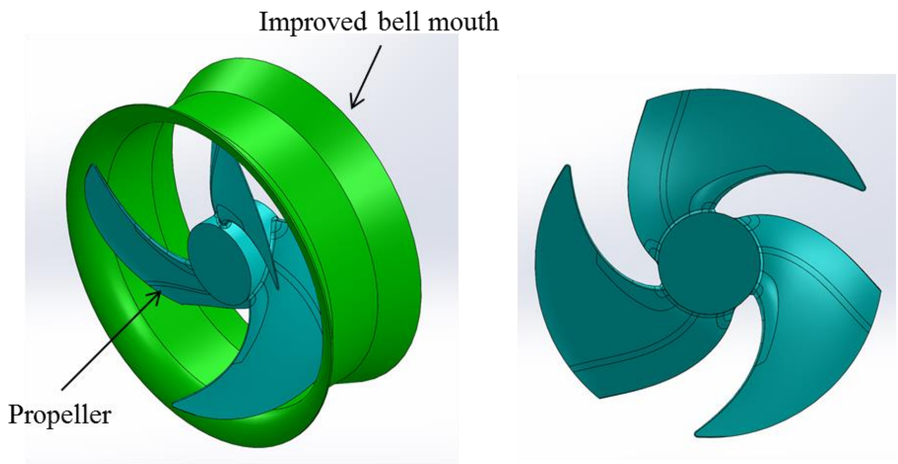


Figure 6: developed outdoor unit



(a) Developed fan (b) Propeller

Figure 7: developed outdoor unit

HIGH EFFICIENCY EFFECTS OF DEVELOPED OUTDOOR UNIT

Figure 8 shows calculated high efficiency effects by three ways. Table 1 summarized calculated outdoor units. Calculated fan shaft power was adjusted by rotating speed in such a way that flow rates of all cases were the same in Fig.8. The fan shaft power of the developed outdoor unit decreased by 56 % compared to the old outdoor unit. The effect of the improvement of the bell mouth shape was 22 %. That of the increase of number of fan was also 22 %. That of the improvement of heat exchanger shape was 12 %.

We confirmed the high efficiency effects experimentally. Figure 9 shows photograph of the experimental apparatus. Actually, flow rate and fan shaft power were measured. The propeller was operated by outer rotational equipment. Torque of the propeller was directly measured. Though the propeller shaft of actual outdoor unit is vertical direction, the shaft of the rotational equipment was horizontal direction in this experiment. Note that the outdoor unit was laid along.

Figure 10 shows measured flow rate and fan shaft power. We confirmed that the fan shaft power of the developed outdoor unit decreased by 50 % experimentally.

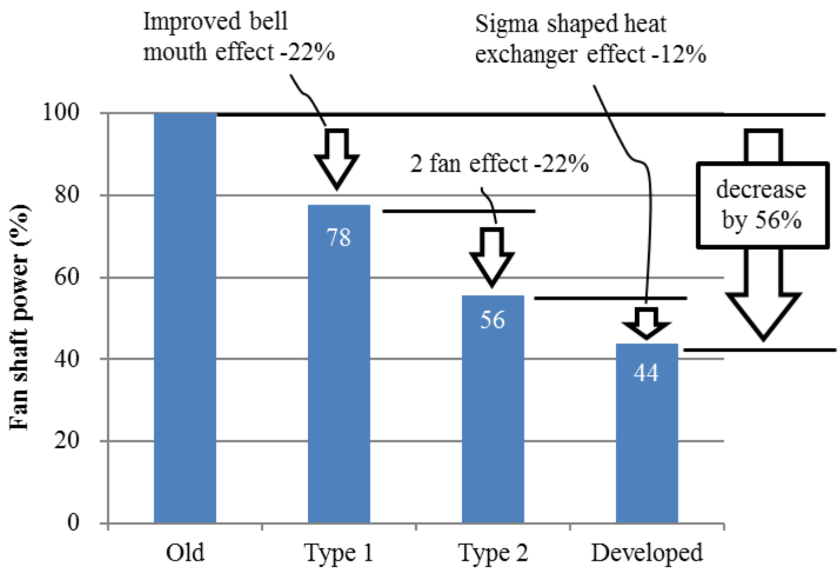


Figure 8: calculated high efficiency effects by three ways

Table 1: Calculated Outdoor Units

No.	Name	Contents
1	Old	Old type outdoor unit
2	Type 1	Improved bell mouth in the old type outdoor unit
3	Type 2	2 fans in the type 1
4	Developed	Developed outdoor unit

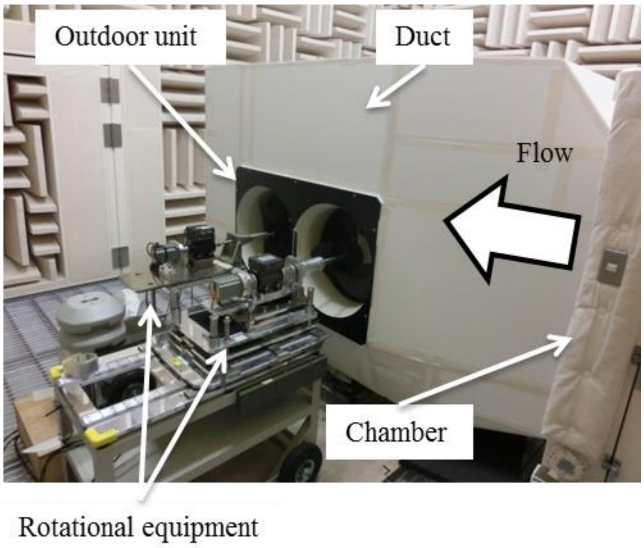


Figure 9: experimental apparatus

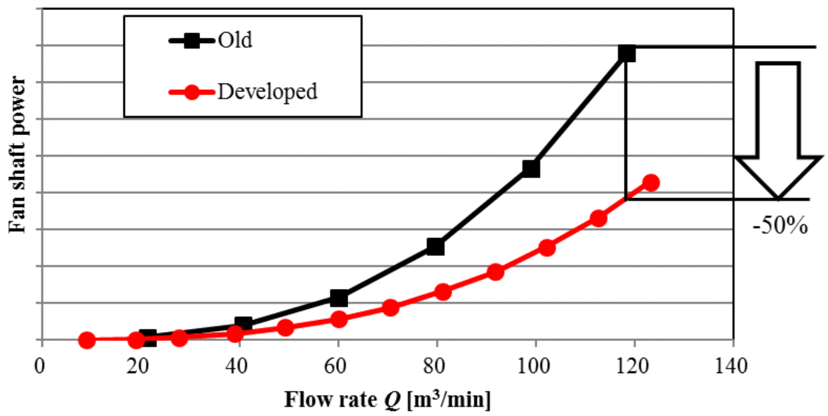


Figure 10: measured flow rate and fan shaft power

CONCLUSIONS

High efficiency fan system for outdoor unit was developed by CFD and experiment. Three ways for development of high efficiency fan system; improvement of bell mouth shape, increase of number of fan and improvement of heat exchanger shape, were investigated. Actually, we employed the long bell mouth, 2 fan type, and sigma shaped heat exchanger. As a result, we confirmed that the fan shaft power of the developed outdoor unit decreased by 50 %.

BIBLIOGRAPHY

[1] Taku IWASE, Tetsushi KISITANI – *Development of High-efficiency Half-Ducted Propeller Fan for Air Conditioners.* Proceedings of the ASME 2014 4th Joint US-European Fluids Engineering Division Summer Meeting, FEDSM2014-21200, Chicago, Illinois, USA, **2014**

[2] Hidenobu OKAMOTO, Akira GOTO, Masato FURUKAWA – *Design of a Propeller Fan Using 3-D Inverse Design Method and CFD for High Efficiency and Low Aerodynamic Noise*. Proceedings of the ASME 2009 Fluids Engineering Division Summer Meeting, FEDSM2009-78454, Vail, Colorado, USA, **2009**

[3] Konrad BAMBERGER, Thomas CAROLUS – *Optimization of Axial Fans with Highly Swept Blades with Respect to Losses and Noise Reduction.* Proceedings of Fan 2012, Senlis, France, **2012**

[4] Shigehisa FUNABASHI, Taku IWASE, Ryoji SATO, Tetsushi KISITANI, Makoto NAGAI – *Two-blade Propeller Fan for Package Air Conditioners.* The International Symposium on New Refrigerants and Environmental Technology 2004, **2004**