Investigation of Fluidic Performance of Cooling Fan in the High Power Automatic Transmission for Commercial Vehicle

In Guk Jeong[1], Ji Hun Yun[1], Chung Seob Yi[2], Chul Ki Song[3], Jeong Se Suh[3#], Kwon Hun Yoon[4] and Kwang Hyun Kim[5]
[1] Graduate School, Gyeongsang National University, 900, Gajwa-dong, Jinju-si, Gyeongsangnam-do, South Korea, 660-701
[2] 2nd Stage BK21(AMAEEP), Gyeongsang National University, 900, Gajwa-dong, Jinju-si, Gyeongsangnam-do, South Korea, 660-701
[3] School of Mechanical Engineering & ERI, Gyeongsang National University, 900, Gajwa-dong, Jinju-si, Gyeongsangnam-do, South Korea, 660-701
[4] Youngdong Tech Co.,LTD., 456-30, Nae-dong, Seongsan-gu, Changwon-si, Gyeongsangnam-do, South Korea, 645-360
[5] S&T Dynamics Co., LTD., 853-5, Oe-dong, Seongsan-gu, Changwon-si, Gyeongsangnam-do, South Korea, 642-020
Corresponding Author / E-mail: jssuh@gnu.ac.kr

Abstract: - This study has investigated numerically and experimentally the flow characteristic of air-cooling fan for transmission oil cooler in the large-size diesel engine. Impellers of cooler were composed of eight normal-scale and eight small-scale blades in the zig-zag pattern. In order to increase the discharge pressure of cooling fan, turbo type of fan blade is proposed in the impeller for transmission oil cooler. The fluidic performance of cooling fan has been estimated numerically by using the commercial code and experimentally carried out with reference on AMCA Standard 210-99. As a result, it is confirmed that the numerical result for performance curve is in a good agreement with experimental data.

Key-Words: - Cooling fan, Impeller, Blade, Theoretical power, Fan performance, Numerical Analysis

1 Introduction

Recently, interest in the high thermal efficiency of diesel engine has been increased in a high power vehicle and the fuel efficiency of diesel engine has been also continuously improved to yield the high power in a large-scale vehicle. Due to the improved thermal performance and output power of engine, there has been an increasing in the application of a high performance automatic transmission to a large vehicle. Especially, an automatic transmission has more complicated structure than that of manual transmission. Torque converter is also required in an automatic transmission to prevent car from being stalled during driving as well as degradation due to the lower transmission level than manual transmission.[1-3)]

The torque converter transfers rotating power from engine to wheel by using the fluid coupling system in a type of small turbine. But the transmitting rate of power energy can be inevitably reduced in the increasing of torque force of transmission when the car is accelerated or ascended. Eventually, the reduced power of torque energy turns into heat and it cause the temperature rise of oil in the transmission. The high temperature of transmission system yields the lubricant oil to be oxidized and poor lubrication and damage to inner components of transmission. To prevent the operating temperature of oil overheating, oil cooler system should be installed in automatic transmission. Generally, for an oil cooler system, water cooling system is applied in small size engine and air cooling system with forced circulation cooling fan in large size engine.

In this work, the oil cooler system will be investigated for automatic transmission for large size engine. This oil cooling system sucks the air into oil cooler by cooling fan. automatic transmission can operate normally in temperature range from 50°C to 120°C and its performance becomes lowering rapidly over 130°C. So it is very important to be ready to cooling this. In the case that forced circulation air cooling system is installed for acquiring cooling capacity of transmission's oil, the performance of cooling fan developing the flow field is the one of factors affect directly to cooling transmission's oil.[4-5)]

ISBN: 978-1-61804-078-7

Thus, this study is conducted to verify the performance curves of oil cooling fan with turbo type by CFD based on experimental result. Also, it tries to secure the data for product development through further optimized design and reliability of result by CFD.

2. Setting up the Problem

2.1 Impeller

Fig.1 shows cooling fan system and impeller shape used in this study. As shown in Fig.1, blade shape of impeller applied in this study has 70^{o} of inlet angle and 56^{o} of outlet angle. Also, twisted angle between upper and bottom of the blade is 22.5^{o}. It rotates in a clockwise direction at 5300 rpm. The impeller consists of 16 blades, includes 8 big blades and small blades each and small blades are positioned between big blades. And it is made of aluminum alloy (AC4C-T6).

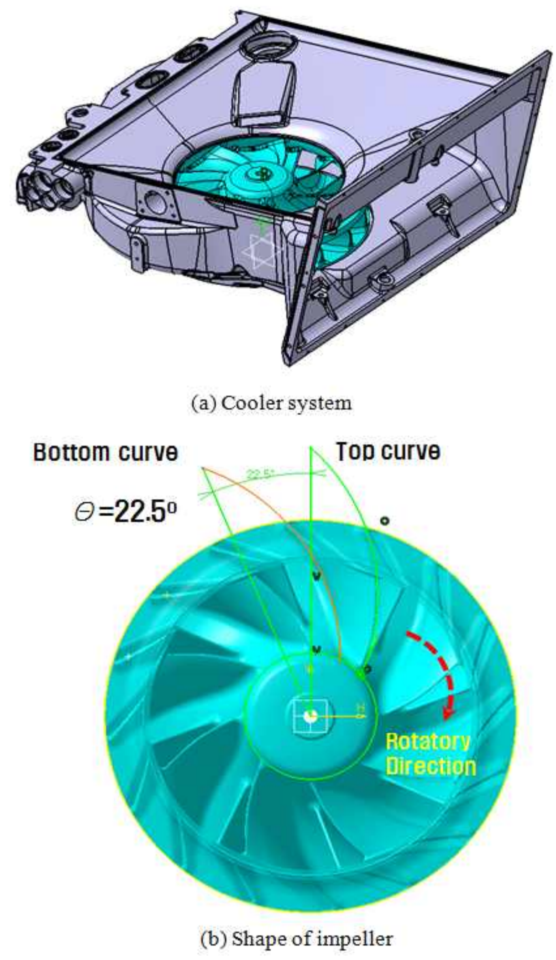


Fig.1 Schematic configuration of cooler system and impeller shape

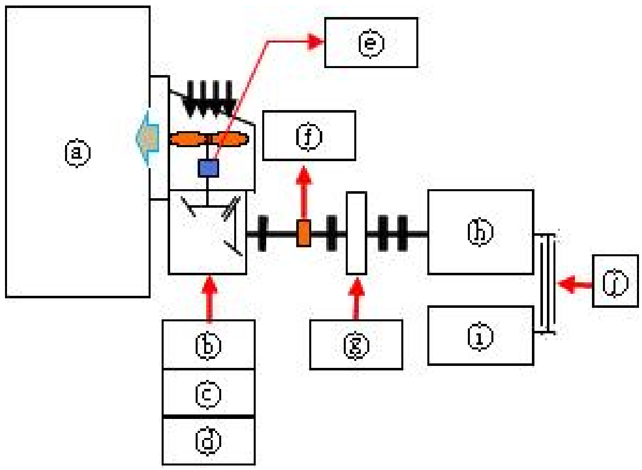
2.2 Research Method and Numerical Conditions

2.2.1. Experimental Method

In this study, experiment on cooling fan is conducted according to AMCA Standard 210-99 as Fig.2 and measurements of the experiment are given in Table 1. As decreasing 10% each from the maximum static pressure at 5300rpm, the flow rate was measured. And the measured data was converted to the pressure in standard atmosphere condition, and then the results were drawn by converting the value to pressure in operating temperature condition

Table 1 Measurement items for experiment

No.	measurement	unit	range
1	Input revolution	rpm	5300
2	static pressure	Pa	0~10000
3	suction flow rate	kg/s	0~8.5
4	output torque	N,m	0~450
5	ambient temperature	℃	10~30



ⓐ	chamber	ⓕ	torque meter (Max 100kg•m)
ⓑ	cooling fan housing	ⓖ	Accelerator (3.5:1)
ⓒ	cooling fan	ⓗ	torque meter (Max 100kg•m)
ⓓ	gear box	ⓘ	motor (200HP)
ⓔ	telemetry	ⓙ	time belt

Fig.2 Schematic diagram of cooling fan experiment setup

2.2.2 Numerical Method

In order to analyze flow characteristics of the cooling fan efficiently, it was assumed to three-dimensional steady flow. With high Reynolds number, standard *k*-ε model, which is considered as the industry standard model, was selected to the turbulence model. It was solved including energy equation regarding effects of air compressibility by high rotation of fan and temperature rise by air friction. And the air was assumed to ideal gas considering the change of air properties by temperature rise. Eq. (1) to Eq. (5) were applied as governing equations for numerical analysis of current research object.

Continuity equation

$$\frac{\partial}{\partial x_i}(\rho\, u_i) = 0 \tag{1}$$

Momentum equation

$$\frac{\partial}{\partial x_j}(\rho\, u_j u_i) = \frac{\partial P}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_i} + S_u \tag{2}$$

Energy equation

$$\frac{\partial(\rho\, u_i T)}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\frac{\mu}{\mathrm{Pr}} + \frac{\mu_t}{\mathrm{Pr}_t}\right)\frac{\partial T}{\partial x_i} + S_T \tag{3}$$

Turbulence kinetic energy equation

$$\frac{\partial(\rho\, u_j k)}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j} + \mu_t G - \sigma\varepsilon \tag{4}$$

Turbulence kinetic disappearance equation

$$\frac{\partial(\rho\, u_j \varepsilon)}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j} + \frac{\varepsilon}{k}(C_1\mu_t G - C_2\rho\varepsilon) \tag{5}$$

Where,

$$\mu_t = \frac{C\mu\rho k^2}{\varepsilon} \tag{6}$$

$$\tau_t = -(\mu + \mu_t)\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \tag{7}$$

$$S_u = -\frac{2}{3}(\mu + \mu_t)\,\frac{\partial}{\partial x_i}\left(\frac{\partial u_i}{\partial x_i}\right) \tag{8}$$

$$\mu_t G = \mu_t\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\frac{\partial u_i}{\partial x_j} - \frac{2}{3}\left(\rho k + \frac{\partial u_i}{\partial x_i}\right)\frac{\partial u_j}{\partial x_j} \tag{9}$$

$C_1 = 1.44,\quad C_2 = 1.92,\quad C_\mu = 0.09 \quad \sigma_k = 1.0 \quad \sigma_\varepsilon = 0.9$

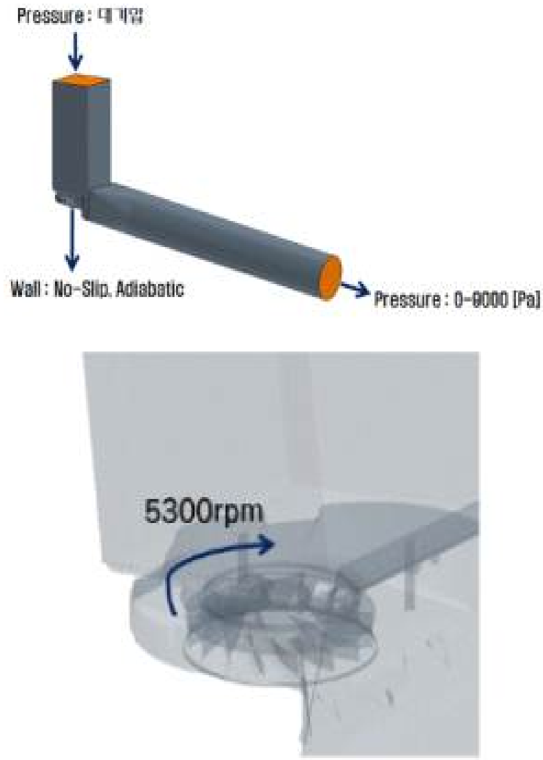


Fig.3 Boundary conditions and fan rotating speed

As a link the pressure and velocity at the governing equation, segregate flow algorithm by SIMPLE(semi-implicit method for pressure-linked equations) was used, and the commercial CFD software, named as Star-CCM, were used to analyze the cooling fan.

For convergence judgment of a dependent variable under steady-state to obtain the numerical result, solving repetition was proceeded until the outlet flow rate came to a certain value. To grant rotating power to the impeller, MRF(Moving Reference Frame), which is the function applying rotating power to object, was configured.

Standard *k*-ε model among the Reynolds-Averaged Navier-Stokes equation was selected as turbulence model. At this moment, to raise the solving accuracy on the wall is critical. Thus, to achieve this, two-layer was inserted on the whole wall.

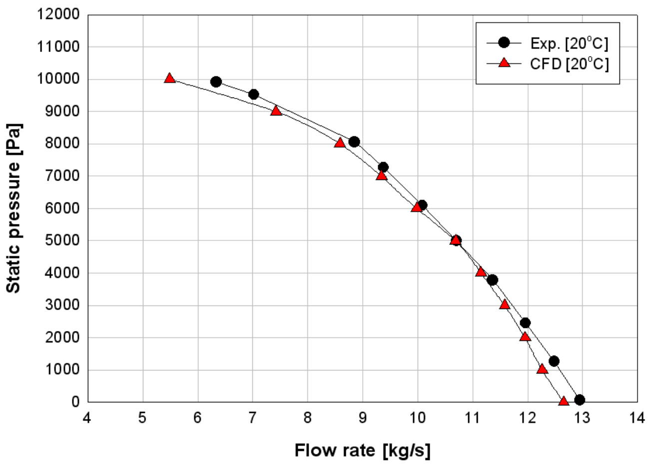
Pressure boundary condition of the inlet in Fig.3 was assumed to the atmospheric pressure, and in the inlet, the turbulence intensity was set to 5% and the turbulent mixing length was 10% of outlet diameter. The outlet pressure range was from zero to 9000Pa. Rotating Speed of Impeller was set to 5300rpm. Polyhedral type was used as the computation grids and the control volume consisted of about 130,000 solid cells.

3. Results and Discussion

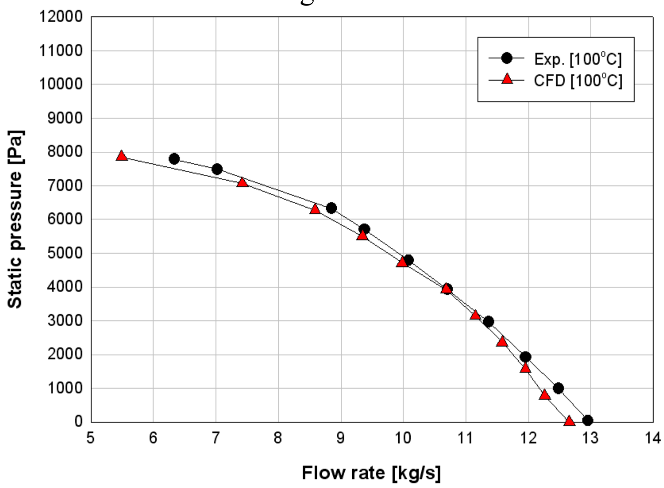
As the result of numerical analysis on the flow characteristics of the oil-cooling fan, the pressure distribution like Fig.4 was obtained. To verify the numerical result, tendency of the numerical analysis was grasped comparing with numerical result by starting from the experimental result following the AMCA Standard 210-99, Analyzing the error range between experiment and numerical analysis, foundation datum for design in real were acquired.

Fig.4 shows the distribution of static pressure at 20°C. In a case that there is no load at the fan outlet, or both inlet and outlet are opened to the atmosphere, it shows the maximum flow rate.

At this, the error between experiment and numerical analysis is about 3%, and it can be verified that the result of numerical analysis indicates a lower flow rate. But at the actually used flow rate 7.5 kg/s, the pressure by experiment is about 9000 Pa and numerical analysis is 8800 Pa.

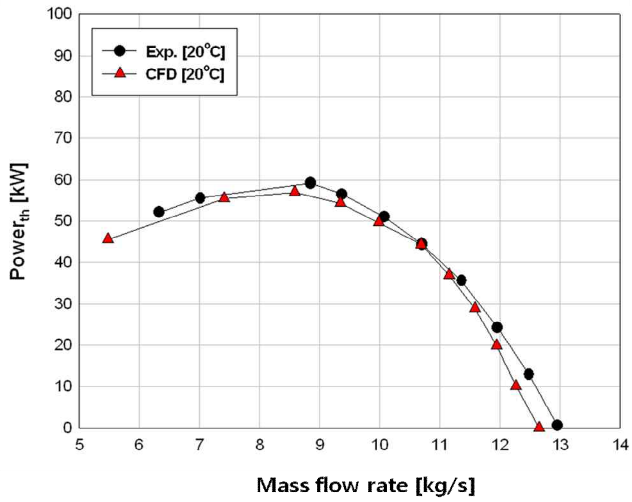


Working fluid=20°C

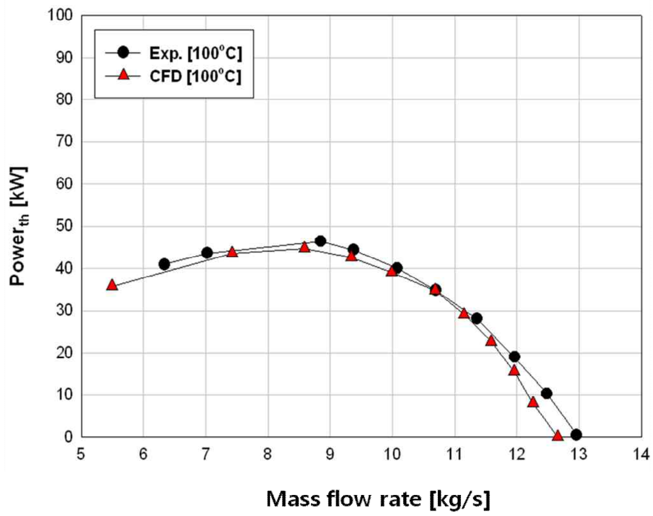


Working fluid=100°C

Fig.4 Distributions of static pressure according to flow rate



Working fluid=20 °C



Working fluid=100 °C

Fig.5 Distributions of theoretical power according to flow rate

In the respect of the broad tendency, the results by experiment and numerical analysis are almost identical and it can be judged that generated error can be covered in design. Moreover, converting the data based on the results at 20°C, the results at 100°C of atmosphere were compared. In the experiment, the mass flow rate shows 7.65 kg/s at 7100 Pa and , in numerical analysis, it show at 6900 Pa. It could be confirmed that 200 Pa of gap between each result was generated.

Fig.5 indicates the comparison of each result for theoretical power. There was no result on the theoretical power in experimental method. It only included the result on the axial power. So the theoretical power was obtained by calculating back to equation (10). The results were represented in Table 2.

Table 2 Comparison of shaft power with theoretical power.

Axial power (kW)	Static efficiency (%)	Theoretical power (kW)
101.85	0.58	0.59
102.05	12.82	13.08
102.42	23.69	24.26
103.39	34.39	35.56
103.52	42.66	44.16
102.15	49.53	50.59
100.84	55.48	55.95
99.14	58.97	58.46
86.91	62.97	54.73
81.13	63.31	51.36

$$P_{th}[\mathrm{kW}] = \frac{Q[\mathrm{m^3/min}] \cdot P_a[\mathrm{mmAq}]}{6120} \quad (10)$$

Unlike the axial power, the theoretical power is entirely generated by fluid so it is different from real axial power. As the theoretical power is acquired by multiplying theoretical power by efficiency, if real axial power is decided, proper motor can be selected. In this study, it could be verified that the theoretical power obtained by numerical analysis was lower than that by experiment.

Fig.6 indicates the efficiency curve. When the outlet is completely opened, the efficiency shows very low. And, when the pressure acts to outlet of the cooling fan, the flow rate becomes low. At that time, it could be verified that, as the flow rate becomes low, the efficiency is increased.

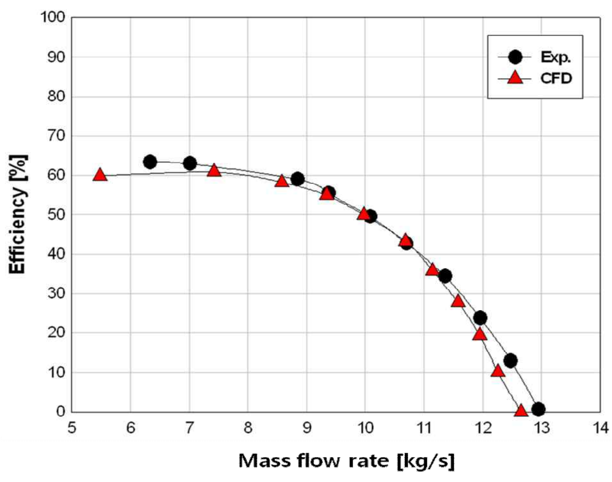


Fig.6 Comparison of efficiency to experiment and CFD

To verify the accuracy of numerical analysis objectively, the peripheral velocity of impeller theoretically obtained by equation (11) was compared with Fig.7. As the result theoretically obtained peripheral velocity was 161.7[m/s] and numerically obtained peripheral velocity of impeller was about 170[m/s], that is, similar results of two could be acquired.

The velocity distributions by numerical analysis show a little bit difference according to the position of casing. It because the pressure distributions are different according to the position of casing eddy chamber and the peripheral velocity of fan shows difference locally by the pressure effect. But the global velocity distribution was similar with the theoretical velocity. Thus, it can be confirmed that the reliability of result by numerical analysis of this study.

$$D_2 = \frac{60 u_2}{\pi N} \qquad u_2 = \frac{\pi D_2 N}{60} = 161.7 \text{ m/s} \quad (11)$$

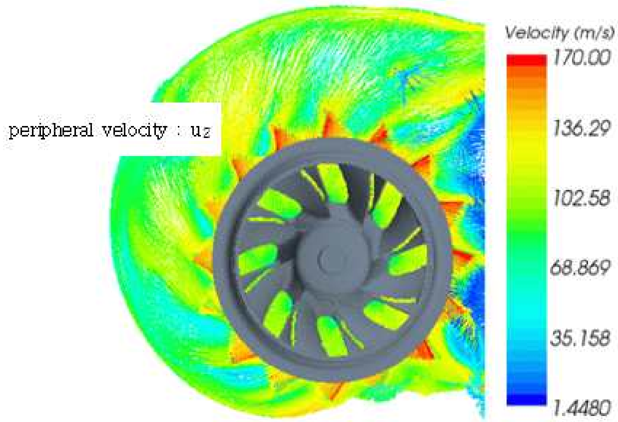


Fig.7 Result of velocity vector in cooling fan at middle section.

4. Conclusions

To verify the performance curve of oil-cooling fan applied in large size diesel engine, numerical analysis was conducted and similar result was acquired.

(1) It could be verified the gap between experiment and numerical analysis was about 3%, which means that the result by numerical analysis showed somewhat lower flow rate.

(2) It could be confirmed that mass flow rate was 7.65 kg/s at 7100 Pa by experiment and 7.65 kg/s of mass flow rate was represented at 6900 Pa by

numerical analysis. There was 200 Pa of gap between experiment and numerical analysis.

(3) About 60% of efficiency was represented at the flow rate 7.5 kg/s which is the operating section of cooling fan.

(4) It could be confirmed theoretical power in real operating section is a little bit low.

ACKNOWLEDGEMENT

This research was financially supported by the second stage BK21 Project, Korea Institute for Advancement of Technology (KIAT) through the Human Resource Training Project for Regional Innovation and the Dongnam Leading Industry office of Ministry of Knowledge Economy.

References:

[1] Junsu Park, Haksu Kim, Duckjea Lee, Siyoul Jang and Kwansoo Han, A Study on the Transmission Fluid Flow in the Main Shaft of the Automatic Transmission, *Proceeding of the 2006 KSAE Spring Annual Meeting*, 2006, pp.2427-2432.

[2] Junsu Park, Daehwan Oh, Duckjae Lee, Haksu Kim, Siyoul Jang, Kwansoo Han and Jin-Sung Kim, A study on the Lubrication Behaviors of Automatic Transmission Fluid inside the Main Shaft through the Oil Holes by Fluent, *Proceeding of the 2007 KSAE Spring Annual Meeting*, 2007, pp.547-552.

[3] Dong Hoon Park, Tae Seok Seo, Do Gi Lim, Hee Bock Cho, Theoretical Investigation on Automatic Transmission Efficiency, *SAE*, 1996, pp.49-62.

[4] Michael A Kluger, Douglas R. Fussner, Bob Roethler, A Performance Comparison of Various Automatic Transmission Pumping Systems, *SAE*, 1996, pp.33-40.

[5] Werner, F. and Frik, S., Optimization of an Automotive HVAC Module by Mean of Computational Fluid Dynamics, *SAE*, 1995, 950439.

[6] Maeng, J. S., Yoon, J, Y., Ahn, T, J., Yoon, J, E. and Hahn, D, J., An Experimental Study for Flow Characteristics Inside the Rotor of a Multi blade Fan/Scroll System, *Trans. of the KSME(B),* Vol. 23, No. 5, 1999, pp. 646~652.

[7] Jeon, W, H., Baek, S, J. and Kim, C, J., Analysis of the Aeroacoustic Characteristics of the Centrifugal Fan in a Vacuum Cleaner, *Journal of Sound and Vibration*, Vol. 268, Issues. 5, 2003, pp. 1025~1035.

[8] Lee, D, W. and Yoo, S, Y, A Numerical Study for Performance of Automotive HVAC System, *Korean Journal of Air-Conditioning and Refrigeration Engineering*, Vol. 16, No. 11, 2004, pp.1084~1091.