

Development of New Cooling System using Nacelle Duct

Minho Ha, SeungHeo, Cheolung Cheong and Park K. Y.

Abstract—In this paper, a new cooling system using a nacelle duct is proposed for the mechanical room in the household refrigerator. The conventional mechanical room consists of a condenser, a compressor and an axial fan. The axial fan is mainly responsible for cooling the condenser and the compressor. The new cooling system is developed by replacing the axial fan with the nacelle duct including the small centrifugal fan. The parametric study is carried out to find the optimum designs of the nacelle duct in terms of performance and efficiency. Through this study, it is revealed that the new system can reduce the space, electrical power and noise compared with the conventional system

Keywords—Centrifugal Fan, Cooling Fan, Nacelle Duct, Refrigerator

I. INTRODUCTION

MECHANICAL room of a household refrigerator usually consists of a condenser, a compressor and an axial fan. The axial fan is used to cool the condenser and the compressor [1]. Since the condenser and the compressor are mainly responsible for the refrigeration cycle, the axial fan cooling these equipments is also an important part deciding the overall efficiency of the refrigerator. The basic layout of the mechanical room doesn't change much since the General Electric produced the "Monitor-Top" that was the first refrigerator finding the widespread use in 1927. In this paper, a new cooling system is consisting a nacelle duct and a small centrifugal fan is proposed to replace the conventional axial fan. The proposed systems are numerically analyzed to assess their performance and efficiency. This study shows that the new system have potential to replace the conventional one.

II. STRUCTURE OF NEW COOLING SYSTEM

Fig. 1 shows the ideal layout of a mechanical room employing a new cooling system using a nacelle duct, instead of an axial fan. It reveals that the space required by the new mechanical room can be drastically reduced in comparison with the conventional one where the axial fan is generally used to cool the condenser and compressor.

Minho Ha is with School of Mechanical Engineering, Pusan National University, Busan, 609-735 Republic of Korea (e-mail : hmh@pusan.ac.kr).

SeungHeo is with School of Mechanical Engineering, Pusan National University, Busan, 609-735 Republic of Korea (e-mail : s.heo@pusan.ac.kr).

Cheolung Cheong is with School of Mechanical Engineering, Pusan National University, Busan, 609-735 Republic of Korea (e-mail : ccheong@pusan.ac.kr).

Park K. Y. is with the LG Electronics Inc. Refrigeration Division, Changwon, 641-110 Republic of Korea (e-mail:kilyong.park@lge.com).

However, this advantage will be achieved only if the performance of new cooling system is equivalent to that of the axial fan.

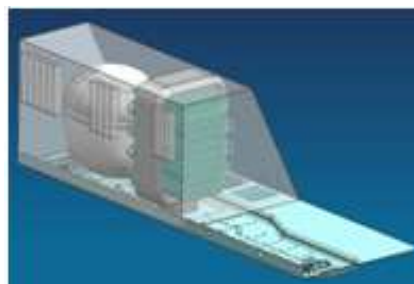


Fig. 1 The image of new mechanical room

The nacelle duct includes a small centrifugal fan in the intake region, which supply the air to the nacelle duct. The air driven by the fan passes through the internal duct of the nacelle and exits through the exhaust gap located on the ring of the nacelle duct. The air discharging from the nacelle duct pulls the surrounding air due to the geometry of the nacelle duct [2]. Fig. 2 shows the geometry of circular nacelle duct.

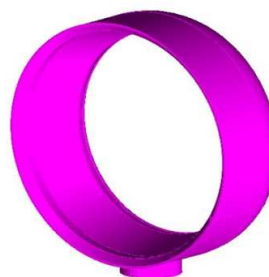


Fig. 2 The image of nacelle duct

To develop the new cooling system, the Computational Fluid Dynamics (CFD) techniques are utilized. First, the performance and efficiency of the existing axial cooling fan is analyzed. Then, on a basis of these reference results, parametric study is carried out to find the optimum design of the new nacelle duct. For the parametric study, the sectional shape of the nacelle duct and the installed locations are varied, which are detailed in the following sections, respectively.

III. SECTIONAL SHAPE OF THE NACELLE DUCT

The reference sectional shape of a nacelle duct is set to be an airfoil shown in Fig. 3(a). The airfoil shape reduces the drag when the air is driven by the nacelle duct. The detailed dimensions of the nacelle duct are as follows: the cord length is 60 mm, the chamber 25 mm, the exhaust gap 1.3 mm and the diameter of the nacelle ring 180mm. A square box is introduced to take into account the actual shape of mechanical room, as shown in Fig. 6. The dimensions of the square box are as follows: the width and height are the same as 200 mm and the length is 600mm. The nacelle duct is located inside the square box. The seven different models from the reference, shown in Figs. 3(b)-(h), are introduced and analyzed for the parametric study. Design factors selected for the parametric study are the angle of attacks, locations of the exhaust of the nacelle duct, and the gap of the exhaust, as shown in Fig. 4.

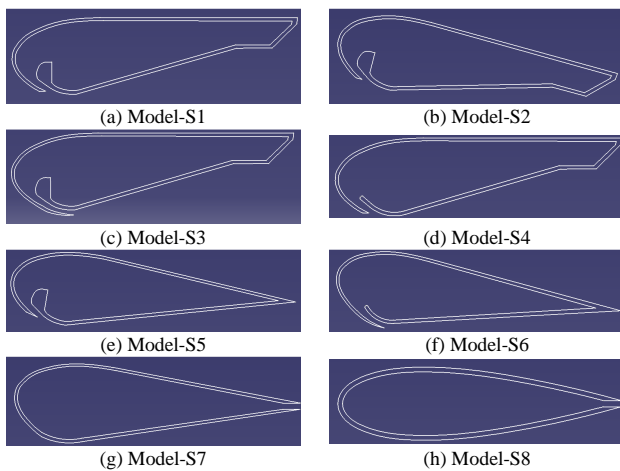


Fig. 3 The images of sectional shapes

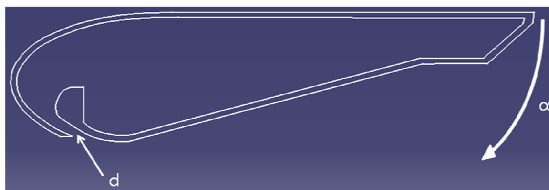


Fig. 4 The definition of additional design factors

The angle of attack of the sectional airfoil affects the direction of the air exiting from the exhaust of the nacelle duct. The pattern of the airflow is driven by the nacelle duct changes according to the location of outlet position. Outlet area changes the velocity of the air exiting from the exhaust of the nacelle duct. Velocity of discharging air is an important factor to determine the flow rate of additional fluid driven by the nacelle duct.

The CAD data of these models are made by using the commercial program, CATIA. On a basis of these CAD data,

the mesh for CFD computations is generated using the ICFM CFD program in the ANSYS. The total elements of the grids are about 400,000. The CFD computations are carried out by using the STAR-CD program. The standard k- ϵ model is used for the turbulence modeling. The boundary conditions for inlet and outlet regions of the square box is set to be constant pressure, and the entrance of nacelle duct is set to be constant velocity of 1m/s for simplicity. Figure 5 shows the typical result for the velocity distributions over the computational domain.

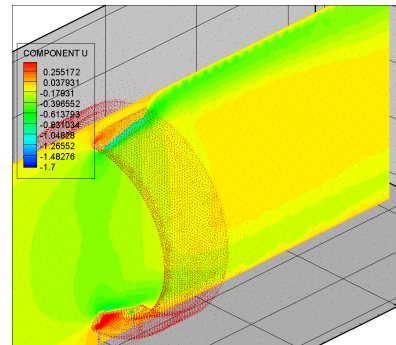


Fig. 5 Result of the velocity of Model-S3

TABLE I
RESULT OF FACTORS IN THE SECTION OF NACELLE DUCT

Case	Nacelle duct flow rate (CMM)	Additional flow rate (CMM)	Total flow rate (CMM)
Model-S1	0.060	0.407	0.468
Model-S2	0.060	0.281	0.341
Model-S3	0.060	0.448	0.508
Model-S4	0.060	0.414	0.474
Model-S5	0.060	0.423	0.484
Model-S6	0.060	0.426	0.497
Model-S7	0.060	0.458	0.519
Model-S8	0.060	0.420	0.481

The volume flow rate for each case is computed and summarized in Table 1. The flow rate denoted by the “Nacelle duct” means the volume flow rate at the inlet of the nacelle duct and “Additional flow” denotes the flow driven by the nacelle duct. The total flow rate is the sum of these two flows. Additional flow rate can be calculated at the cross sectional plane in the front of square box and total flow rate can be calculated at the plane downstream of the nacelle duct. The volume flow rates of additional flow induced by each of the models are seen to vary. Model-S3 obtained by modifying the outlet increases 10 percent of volume flow rate compared with the original. Model-S6 also induces more flow rate than the original. However, Model-S2 with the modified angle of attack does not show the increased volume flow rate. The reason for this is that the airfoil with the increased angle of attack induces stronger vortex, which results in the adverse effects on the flow. Overall, it is found that the largest volume flow rate is obtained from the Model-S7 where the angle of attack, the trailing edge and the position of the exhaust are changed into that shown in Fig. 1(g).

IV. POSITION OF A NACELLE DUCT IN A MECHANICAL ROOM

Position of a nacelle duct is also an important factor affecting the flow rate in the mechanical room, because the additional flow due to the nacelle duct is driven by the difference between the condition of the exhausting air and that of surrounding air [3]. We investigate the effects of the relative positions of the condenser, compressor and the nacelle duct on the volume flow rate.

The condenser, the compressor and the nacelle duct are located in a box of which the cross-section is trapezoidal. The dimensions of the box are determined the same as the actual mechanical room of a household refrigerator in the market. The detail dimensions of box are as follows: the top side is 171mm, the bottom side is 289mm, height is 239mm and the length is 600mm. The inlet and outlet regions of the box are located on the left side and bottom, as shown in Fig. 6.

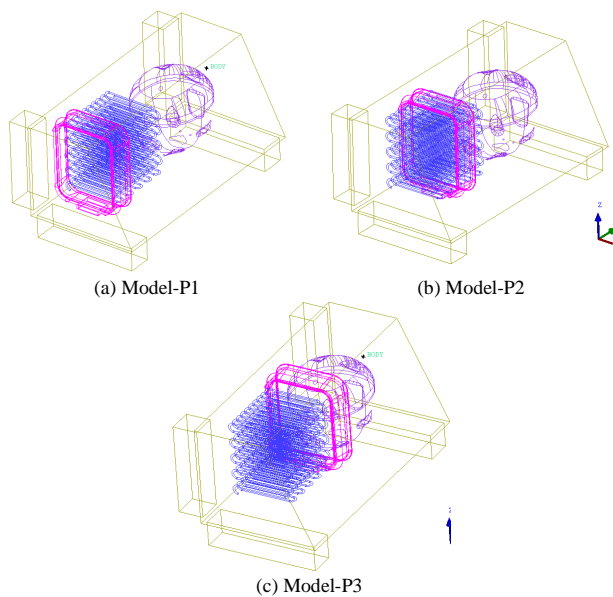


Fig. 6 The schematic view of arrangement of the condenser, compressor and nacelle duct in a mechanical room

The mesh is created by using the commercial software, ANSYS ICEM CFD. The total number of grids is approximately 2,000,000. The CFD computation is carried out by using the STAR-CD. The boundary condition of the duct inlet and outlet is the prescribed static pressure. The inlet boundary condition at the intake of nacelle duct is the prescribed velocity of 1m/s. The standard $k-\epsilon$ model is used for the modeling of the turbulence.

TABLE II
RESULT OF FLOW RATE IN THE ARRANGEMENT OF PARTS

Arrangement	Nacelle duct flow rate (CMM)	Add flow rate (CMM)	Total flow rate (CMM)
Model-P1	0.122	0.226	0.348
Model-P2	0.120	0.185	0.306
Model-P3	0.120	0.409	0.529

Fig. 7 shows the typical results of the velocity distribution over the whole computation domain. Table 2 summarizes the predictions for the volume flow rates according to the arrangement of the compressor, the nacelle duct, and the condenser. It can be seen that there is large difference between the predicted volume flow rates due to the different arrangements. The arrangement in the order of the compressor, the nacelle duct, and the condenser shows the largest volume flow rate that is approximately 40 % larger than the other two arrangements. This result shows that the performance of the nacelle duct is significantly affected by the object placed upstream and downstream of the nacelle duct.

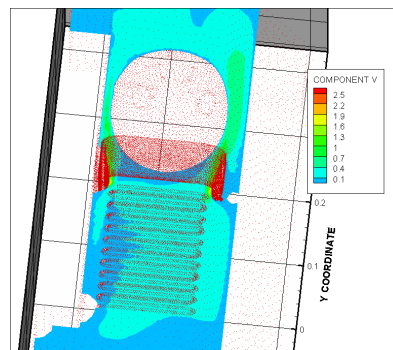


Fig. 7 Iso-contours of the velocities for Model-P3

V. FLOW RATE OF NACELLE DUCT WITH A FAN

In the previous two sections, the intake flow of the nacelle duct is simulated by using the prescribed inlet velocity. In this section, to increase the accuracy of the computations, a centrifugal fan is placed in the entrance of the nacelle duct. The selection of the centrifugal fan is due to the fact that the high pressure drop by due to the frictional loss of the inside wall of the nacelle duct needs to be recovered in order to drive the airflow from the intake of the nacelle duct to the exhaust of the nacelle duct.

The selected fan has a diameter of 110 mm and 10 blades. The characteristics of this fan were analyzed in the recent study [4], [5]. Because the diameter of the intake of the nacelle duct is smaller than that of the fan, the dimensions of the nacelle duct are changed: the cord length is 100 mm, the chamber 25 mm and the width of the square ring 180 mm. The cross-section of actual mechanical room is a trapezoidal shape. Fig. 8 shows the whole computation domain.

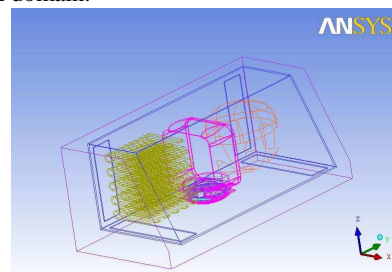


Fig. 8 The image of whole computation domain

The total number of mesh is 3,500,000. The CFD computation is performed by using the Fluent. Multiple Rotating Frame (MRF) method is used to consider the rotation of the fan blades. The rotation speed is set to be 2150rpm that is rated speed for the fan.

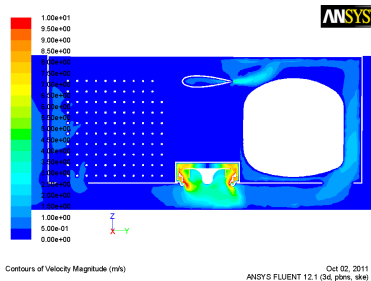


Fig. 9 The iso-contours of velocities at the symmetric plane

TABLE III
RESULT OF FLOW RATE IN NEW MECHANICAL ROOM WITH FAN

	Nacelle duct flow rate (CMM)	Add flow rate (CMM)	Total flow rate (CMM)
Prototype	0.2991	0.5328	0.8317

Fig. 9 presents the contours of velocity on the symmetric plane. Table III summarizes the predicted volume flow rates for the case using the actual fan. It is seen that the additional flow is less than two times of the nacelle duct flow, which is less than those predicted without the fan. This result shows that there is a need for optimizing the intake duct flow passage to increase the flow rate. However, the detailed reason for this needs to be more investigated in the near future. On a basis of these results, experiments with prototypes will be carried out to compare the proposed system with the existing one.

VI. CONCLUSION

A new cooling system using a nacelle duct is proposed for a household refrigerator. The performances of new cooling systems are assessed by using the CFD techniques. First, sectional shapes of the nacelle ducts are varied to find the optimum design for the maximum flow rate. The, installed location of the nacelle duct in a mechanical room is determined for the maximum flow rate. The optimum cross-sectional shape of the nacelle duct is found to be Model-S7, where the angle of attack is increased, the location of the exhaust is moved toward the trailing edge and the shape of the trailing edge is made more shaper and smoother. This shape is found to induce highest velocity of the air exiting from the exhaust of the nacelle duct than others. The position of the nacelle duct located between the condenser and the compressor allows more air flow rate. This means that the performance of the nacelle duct is significantly affected by an object located upstream and downstream of the nacelle duct. Finally, a centrifugal fan is applied in the inlet of

the nacelle duct, instead of the simple inlet boundary condition. The total flow rate predicted for this case is less than the previous cases, which necessitates the further research for the optimization of the inlet flow driven by the centrifugal fan. However, the results shows that the proposed new cooling system have potential to replace the conventional one in that the required space can be reduced only if the other performance parameters such as the electrical power and the noise of the new system can be retained equivalent to the existing one.

ACKNOWLEDGMENT

This work was supported by the Human Resources Development of the Korea Institute of Energy Technology Evaluation and Planning (KETEP) grant funded by the Korea government Ministry of Knowledge Economy (No. 20114010203080)

REFERENCES

- [1] J. Y. Kim, S. H. Kim, T. H. Kim, S. M. Kwak, D. S. Jang, Y. H. Lee, Y. W. Lee, "Characteristics of Heat Flow in a Machine Room of Refrigerator," Journal of The Korean Society for Power System Engineering, pp. 15-354, 2001.
- [2] OKABE MASUMI, HONJO SHIGERU, "FAN," Japan Patent, 55-071979 (1980)
- [3] B. Y. Han, Y. K. Kim, T. H. Kim and Y. W. Lee, "Optimization of Refrigerator Machine Room for Energy Saving," Journal of The Korean Society for Power System Engineering, pp. 73-77, 2005.
- [4] Seungyub Lee, SeungHeo and Cheolung Cheong, "Prediction and reduction of internal blade-passing frequency noise of the centrifugal fan in a refrigerator," International Journal of Refrigeration, Volume 33, Issue 6, September 2010, Pages 1129-1141
- [5] SeungHeo, Cheolung Cheong, Tae-Hoon Kim, "Development Of Low-Noise Centrifugal Fans For A Refrigerator Using Inclined S-Shaped Trailing Edge," International Journal of Refrigeration 34 (2011), pp. 2076-2091