한국기계가공학회지, 제19권, 제1호, pp.56～62(2020.01) ISSN 1598-6721(Print)
Journal of the Korean Society of Manufacturing Process Engineers, Vol. 19, No. 1, pp.56~62(2020.01) ISSN 2288-0771(Online)

https://doi.org/10.14775/ksmpe.2020.19.01.056

Thermal and Fluid Analysis on Air Distribution in a Elevator Car

Kyung Taek Chung*, Chung Seob Yi*,#
*School of Lift Engineering, Korea Lift Collage

엘리베이터 카 내부 기류분포에 관한 열 유동해석

정경택*, 이중섭*,#
*한국승강기대학교 승강기공학부

(Received 5 September 2019; received in revised form 14 September 2019; accepted 17 October 2019)

ABSTRACT

The purpose of this study is to observe the visualization of the flow field for air flow distributed in the car from the ventilation fan installed in the ceiling of the passenger elevator car through the numerical analysis using computational fluid dynamics. STAR-CCM+, which is a code used for the numerical analysis, was used to predict the airflow distribution inside the elevator car. The numerical analysis of the distribution of the air current in the elevator was carried out. As a result, the analysis results for each point and the visualization of the air current distribution and the temperature distribution in the elevator car and were obtained. It was found that heat transfer was actively occurring inside the car due to the influence of the flow field discharged from the ventilation vent installed in the ceiling in the elevator car, and especially the convection heat transfer of Model-2 was more active than that of Model-1. As a result, the temperature distribution inside the car was found to be relatively low. In addition, the temperature distribution at a cross-section of 1700mm height in the elevator car shows that Model-2 is the location of the ventilation vent which makes people feel more comfortable.

Key Words : Elevator Car(엘리베이트 객실), Air Distribution(기류분포), CFD(전산유체역학), Ventilation(환기)

1. Introduction

Typically, elevators have served as a means of transporting people or cargo inside high-rise buildings.

Corresponding Author : csyi@klc.ac.kr
Tel: +82-55-949-2266, Fax: +82-55-949-2200

However, given recent improvements in people's quality of life, there has been increasing interest in ventilation systems inside elevator cars. In particular, the effects of yellow and fine dust are raising both interest and concern about air quality in Korea. Currently, most elevators are not equipped with special air conditioning or ventilation systems.

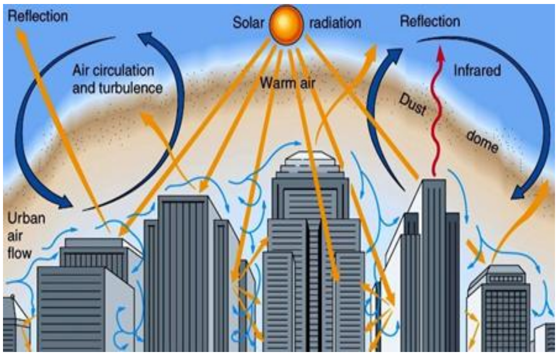


Fig. 1 Phenomenon occurs of urban heat island effect

This study numerically analyzes the airflow distribution inside an elevator car by selecting a model in which a ventilation fan is installed and forced to operate. Airflow distribution is analyzed to determine any changes in the position of the elevator car's internal vents.

2. Methods and Targets of hermal-Fluid Analysis

2.1 Targets of Thermal-Fluid Analysis

To conduct a numerical analysis of the airflow distribution in elevator cars, one of ‘F’ company's elevator models was selected, as Fig. 2 shows. This model features an airflow window installed on the elevator car’s ceiling, allowing the airflow to be discharged directly into the elevator car via the ventilation fan. Regarding this fan, the flow rate and pressure are given as the boundary condition based on the fan’s specifications provided by the manufacturer.

In this study, the flow rate of ventilation fans used in elevator cars was 385 m^3/h, which was then converted to a mass flow rate of 0.128 kg/s. A boundary condition was given to allow ventilation fans with a pressure of 88Pa to blow air into the elevator cars.

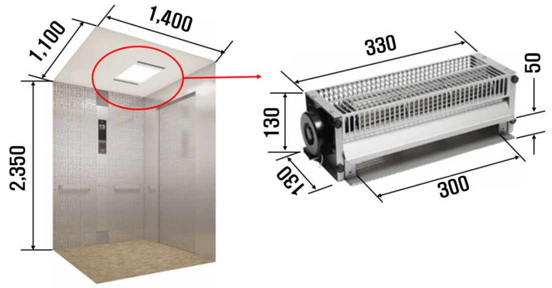


Fig. 2 Configuration of elevator car and forced ventilation fan

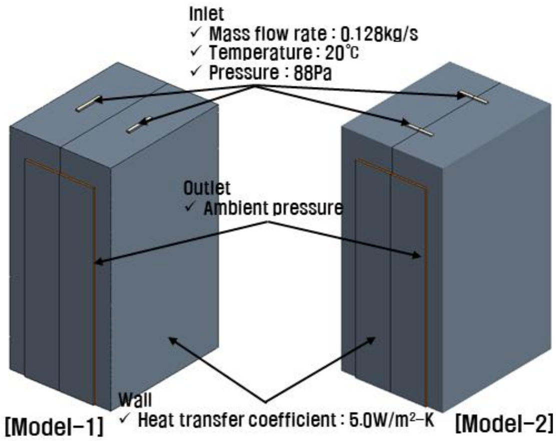


Fig. 3 Boundary conditions

2.2 Thermal-Fluid Analysis Method

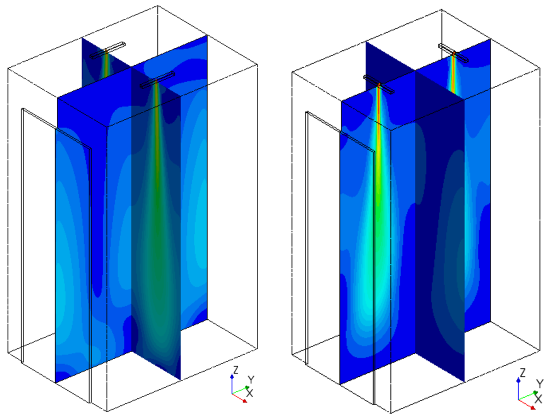
In order to ensure the reliability of the flow analysis results, it is necessary to select the proper turbulence models for accurate modeling and evaluate the analytical models to be employed. In addition, it is important to check the reliability of the calculation grid. Moreover, since the choice of the Solver has a significant impact on the calculation accuracy, commercial S/W analyses were performed using STAR-CCM+.

In this study, the flow within the elevator car is assumed to be three-dimensional steady-state turbulence, as determined by flow analysis. The governing equations used to calculate the velocity distribution and pressure inside the elevator car include a continuity equation and a momentum equation. The turbulence model was based on the $k-\varepsilon$ model, which was previously validated in the industry.[7-8] In terms of a numerical analysis technique, the connection of pressure and velocity in the governing equation used the Segregate Flow algorithm according to the Semi-Implicit Method for Pressure-Linked Equations (SIMPLE). STAR-CCM+, a numerical analysis code applied in this study, is calculated by the following numerical algorithm to analyze piping flow. The method of numerical analysis for obtaining the analysis results was used as the basis for determining the convergence of dependent variables in the normal state when the exit's flow rate remains unchanged and the calculation results are displayed at a fixed and stable value. Additionally, when calculating iterations in a steady state, the dependent variables' convergence determination was used as the basis when the residual value reached 10^{-3} or below.

3. Thermal-Fluid Analysis Results and Findings

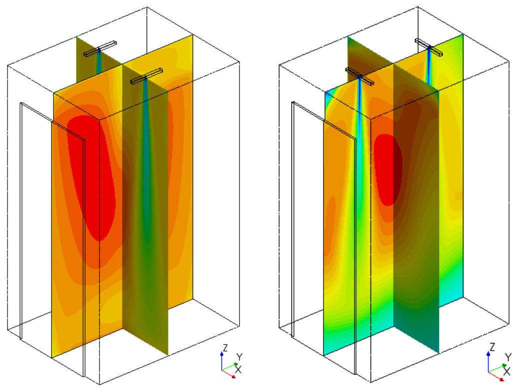
Fig. 4 illustrates the distribution of the velocity size of each model's central section. As the figure shows, in the case of Model-1, there are left and right vents installed at the entrance of the elevator car, and air from outside of the elevator is drawn through the fan. On the other hand, in the case of Model-2, ventilation ducts are installed both before and after the elevator car's entrance door.

When the ventilation fan's mass flow rate of 0.128 kg/s was introduced into the elevator car from each vent, it was found that a maximum flow rate of approximately 13m/s was discharged from the ventilation vent's outlet. This created a flow field inside the elevator car, gradually reducing the flow rate. In addition, since the same blower was applied in each model, no significant difference was found in flow rate changes because the flow rate and pressure were under the same conditions. However, since the flow is released vertically from the car's upper vent, it was possible to confirm that the flow becomes trapped in the space between the vents. Particularly, the study confirmed that Model-1's flow had a distribution of air trapped in the central area compared to Model-2. Overall, the airflow distribution did not appear to have a well-circulated structure inside the elevator car. Thus, it was possible to confirm that the role of the currently installed ventilation fan was simply to supply fresh air.



(a) model-1 (b) model-2

Fig. 4 Results of velocity magnitude contour in the elevator car at center sections



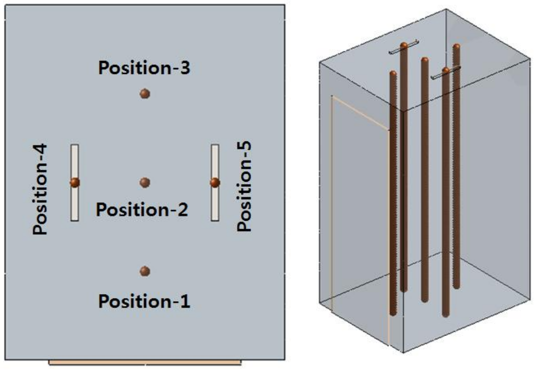
(a) model-1 (b) model-2

Fig. 5 Results of temperature contour in the elevator car at center sections

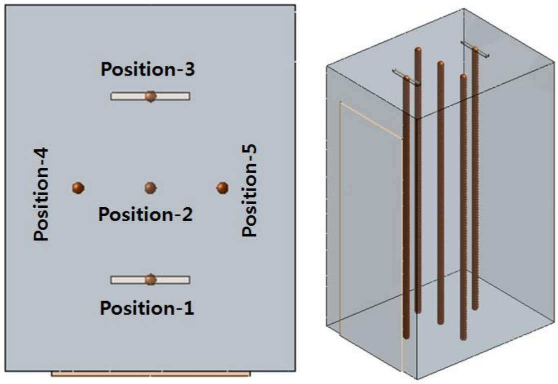
Fig. 5 shows the temperature distribution of each model's central section. As described in the above-mentioned velocity distribution, the temperature was found to be relatively high as the ventilation vents installed on both sides caused the flow field to become trapped and stagnant. In the case of Model-1, especially, the ventilation vents' clearance was relatively narrow compared to that of Model-2, so it was possible to note that the flow was trapped between the ventilation vents and that the temperature distribution was high. This is considered to be essentially favorable to airflow diffusion because the interval between ventilation vents is relatively long.

Fig. 6 designates a total of six virtual sensors employed to obtain data about airflow distribution inside the elevator car. Thus, the sensors measured the velocity and temperature values in the vertical direction.

Fig. 7 illustrates the velocity distribution at five different points on each section in the elevator car's vertical direction. As the figure shows, in the case of Model-1 it was possible to verify the same results since Position-4 and Position-5 are symmetrical points on both sides and ventilation vents were installed in these locations. Meanwhile, Position-1 and Position-3 had similar velocity profiles. Position-2, the elevator car's central point, was found to have a lower flow rate at the top, which decreased again at a height of approximately 750 mm after the flow rate increased as it descended to the floor. Additionally, for Position-1 and Position-3, an increased flow rate was once again found. These findings suggest that, in the case of Position-2 (which is the center), the flow rate decreased as the flow became trapped in the central area due to the effect of the downward flow in both ventilation vents. In Model-2, the highest flow rate distribution was identified in Position-1 and Position-3 with ventilation vents installed.



(a) model-1

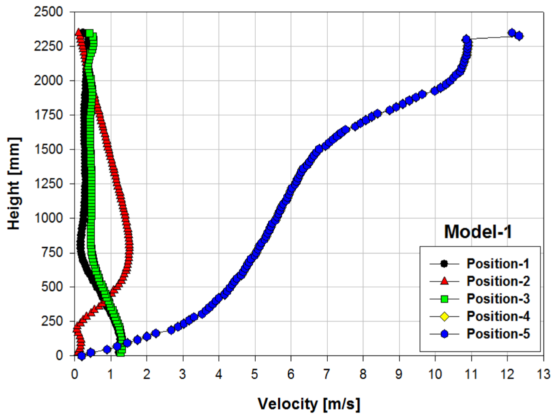


(b) model-2

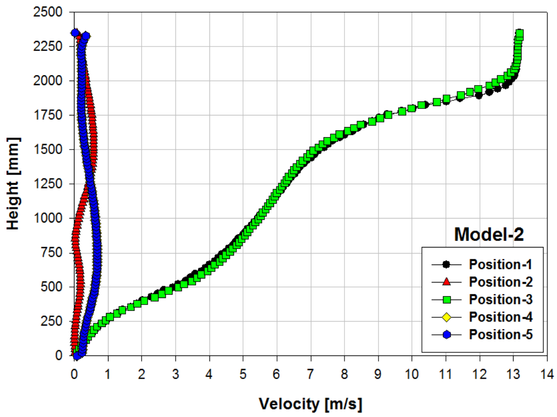
Fig. 6 Virtual sensing positions

Position-4 and Position-5 had symmetrical positions with the same flow rate distribution. Regarding Position-2, the elevator car's center, the flow rate was found to be relatively low compared to Model-1. This is because the ventilation vents' clearance is relatively wider than that in Model-1.

Fig. 8 successfully confirms that, at the point where the temperature distribution was discharged, the temperature was discharged at 20°C and then gradually increased as the flow was carried out to the bottom of the elevator car. This phenomenon is the result of the relatively high temperature and heat transfer inside the elevator car.

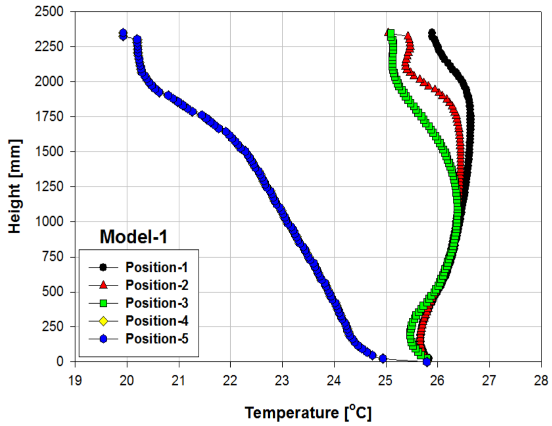


(a) model-1

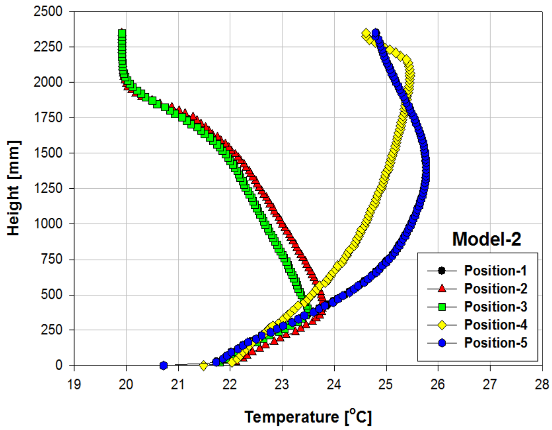


(b) model-2

Fig. 7 Results of velocity distribution at center section each positions



(a) model-1

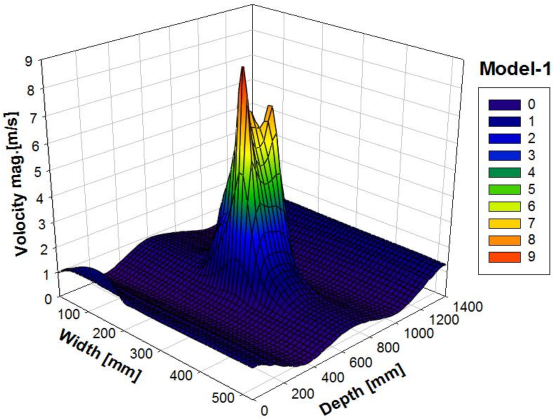


(b) model-2

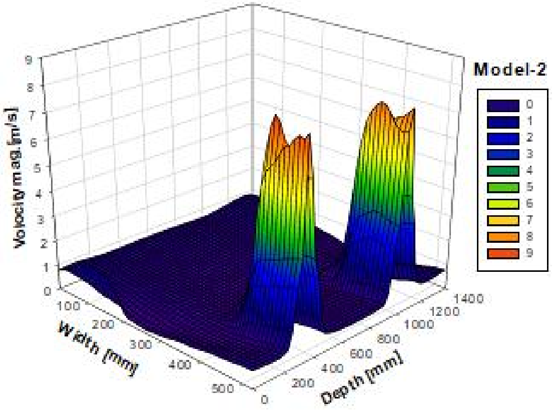
Fig. 8 Results of temperature distribution at center section each positions

The temperature in Model-1 was found to be low to the floor, while the temperature in Model-2 was high, exhibiting a temperature drop from about 350 mm. This is likely due to the good diffusion of the flow from the vent into the elevator car, as seen in Fig. 5's temperature distribution section. Moreover, this can be explained by the ventilation vents' wider spacing, as described in the above-mentioned finding of velocity distribution. In turn, this allows for the airflow to be distributed relatively wider than in Model-1, where airflow is trapped in the center of the car and does not allow the flow to spread.

Fig. 9 shows the velocity distribution for the section at the elevator car's height of 1,700 mm. As the figure illustrates, one side of the symmetry is displayed in the thermal-fluid analysis. Therefore, only the half starting at the center of the elevator car's door represents the analysis results. In the case of Model-1, the flow is discharged from the left and right sides of the elevator door. The highest flow rate was identified at the location near the discharge outlet and, in terms of the surrounding area, the flow rate was found to be low.

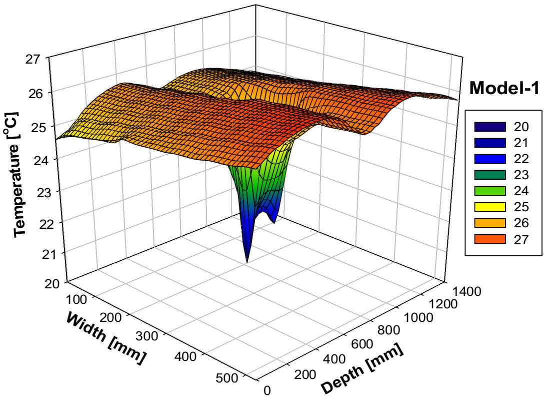


(a) model-1

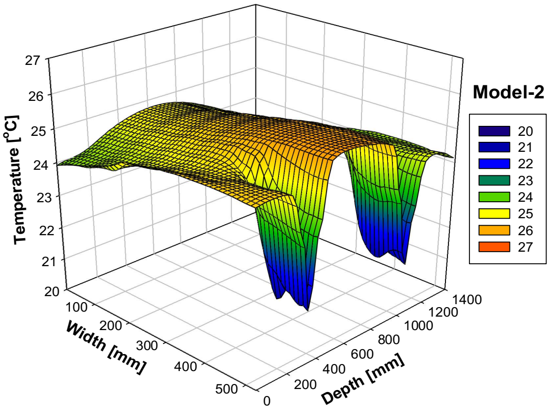


(b) model-2

Fig. 9 Results of velocity distribution at height 1700mm



(a) model-1



(b) model-2

Fig. 10 Results of temperature distribution at height 1700mm

In the case of Model-2, a flow rate of about 8 m/s was found to occur near the ventilation vents in the front and rear positions of the center of the elevator doors. Moreover, this flow rate was distributed evenly compared to Model-1. Overall, the study found that a flow rate of approximately 1 m/s was distributed at a height of 1,700 mm, except for around the vent outlet.

Fig. 10 illustrates the temperature distribution for the section at the elevator car's height of 1700 mm. This height represents the location of an adult's head while in the elevator. In this location, the transfer of heat is facilitated because the head is exposed to the human body's atmosphere and the heat emission in this area is also large. Therefore, when boarding the elevator car, this study focused on checking the temperature distribution at this height

4. Conclusion

By performing a numerical analysis of the airflow distribution inside an elevator car, this study obtained a visualization of the airflow distribution and temperature distribution inside the elevator car. Moreover, an analysis was conducted for each point.

Due to the effects of the flow field discharged from ventilation vents installed on the inside ceiling of the elevator car, active heat transfer was identified inside the elevator car. This was particularly the case in Model-2, which featured active convection heat transfer when compared to Model-1. This resulted in a relatively low temperature distribution inside the car. Finally, the temperature distribution in the section at the elevator car's height of 1700 mm confirmed that the ventilation vents' location was more comfortable in the case of Model-2.

REFERENCES

1. Kim, B. G., Choi, H. W., Yong, H. T., and Lee, M. H., "Horizontal Air-jet Effect on the Natural Convection around a Range-hood System", Korean Journal of Air-Conditioning and Refrigeration Engineering, Vol. 17, No. 4, pp. 333-340, 2005.
2. Lim, K. B., Lee. K. S., and Lee. C. H., "A Numerical Study on the Flow Characteristics of Kitchen Hood System", Korean Journal of Air-Conditioning and Refrigeration Engineering, Vol. 18, No. 4 pp. 359-369, 2006.
3. Lim, K. B., Lee. K. S., and Lee. C. H., "A Numerical Study on the Characteristics of Flow Field, Temperature and Concentration Distribution According to Changing the Shape of Separation Plate of Kitchen Hood System", Transactions of the KSME B, Vol. 30, No.2, pp.177-185, 2006.
4. Lee, S. S., Moon. J. H., Lee. T. G., Lim. Y. C., and Lee. J. H., "Improvement of Working Environment by the Space Ventilation System in a Large Welding Factory", SAREK winter conference, pp. 345-350, 2004.
5. Koo, N. Y., Lee. Y. K., and Young. Y. S., "An Experimental Study on the Air Curtain Range Hood Interrupting the Diffusion of Polluted Air", SAREK winter conference, pp. 382-387, 2002.
6. Witt, P. J., Solnorda. lC. G., Mittoni. L. J., Finn. S., and Pluta. J., "Optimizing the Design of Fume Extraction Hoods using a Combination of Engineering and CFD Modelling", Applied Mathematical Modelling", Vol. 30, pp.1167-1179, 2006.
7. Yi, C. S., Chin, D. H., "A Study on the Thermal and Flow Characteristics of Wind and Radiant Heat Shield for Offshore by using Mesh Screen ", Journal of the Korean Society of Manufacturing Process Engineers, Vol. 11, No. 3, pp. 166-173, 2012.
8. Yi, C. S., Lee, B. H., Chin, D. H., "Numerical Analysis on the Thermal and Fluid in Air Conditioning Duct for Marine Offshore", Journal of the Korean Society of Manufacturing Process Engineers, Vol. 17, No. 5, pp. 23-29, 2018.