한국기계가공학회지, 제17권, 제5호, pp.16～22(2018.10) ISSN 1598-6721(Print)
Journal of the Korean Society of Manufacturing Process Engineers, Vol. 17, No. 5, pp.16~22(2018.10) ISSN 2288-0771(Online)

https://doi.org/10.14775/ksmpe.2018.17.5.016

A Study on the Flow Analysis of Ventilation Louver for Polar Ship

Chung-Seob Yi*, Do-Hun Chin**,#
*Gyeognam National University of Science and Technology
**R&D Center, Dawon Tech LTD., CO.

극지운항 선박용 루버 환기창 유동해석에 관한 연구

이중섭*, 진도훈**,#
*경남과학기술대학교, **다원텍(주)기술연구소

(Received 25 June 2018; received in revised form 5 July 2018; accepted 11 July 2018)

ABSTRACT

This study is about flow distribution in ventilation systems used in marine louvers. The flow analysis on a louver installed on the vent of a vessel results in the following conclusions: (a) as the velocity of the fluid entering the louver increases, the pressure drop increases; (b) as the pressure drop increases, it tends to increase following a quadratic function. The velocity was confirmed to decrease at the entrance of the louver. This indicates that as the pressure increases, the velocity decreases, and the velocity of the moving fluid is increasing as it passes through the louver vanes.

Key Words : Ventilation(환기), Louver(루버), CFD(전산유체역학), Polar Ship(극지운항선박)

1. Introduction

A ventilating opening is installed in the ship to discharge internal air to the outside due to the operations of the air-conditioning system and internal ventilation system. A louver installed in the ventilating opening can be divided into a fixed type, in which louver blades are not movable, and an operating type, in which the ventilating opening can be opened or closed when necessary. The operating type that can be opened or closed plays a similar role as a kind of damper. The louver applied in this study is a mode installed by the operating type.

The louver is a kind of air-resistant object when ventilated according to the design of the louver blade, as shown in Fig. 1. Thus, the installation location and the number of louver blades are determined depending on the pressure distribution by the flow or flow pattern. In addition, the louver plays a role in controlling the air flow and flow rate as well as the differential pressure between the inlet and outlet when a forced ventilation system is installed through air blowers.[1,2]

Corresponding Author : chindohun@hanmail.net
Tel: +82-51-629-7737, Fax: +82-55-327-2310

A previously published study[3] described the pressure distribution results produced when air passed through the ventilation louver. However, this study aims to investigate the effect of flow distribution.

2. Flow analysis method

In this study, the flow in the louver blade was assumed to be three-dimensional (3D) turbulent flow at a steady state. The governing equations used to calculate the velocity distribution and pressure were continuous and momentum equations. For the turbulence model, the realizable k-ε model, which has already been verified in engineering fields, was employed. The energy equation was not calculated since heat transfer was not considered[4,5].

This study performed calculations using STAR–CCM +, a commercial thermal fluid program. For the numerical analysis environment to obtain the analysis result, if a residual value reached 10^{-3}or lower, it was considered to have converged for the convergence test of the dependent variables during the iterative calculation at a steady state[6,7].

Fig. 2 shows the boundary conditions used in the numerical analysis. As the boundary condition to perform the numerical analysis of the louver, the flow rate was gradually increased from 0.5 to 5m/s at the inlet, and for the outlet, the atmospheric pressure was applied to conduct the flow analysis. In addition, a uniform flow condition was used whose flow direction was constant in the louver inlet, and the turbulent kinetic energy was assumed to have 5% of the mean flow rate at the inlet.
For the turbulent dissipation rate, the turbulent mixing distance was set to 10%.

Fig. 3 shows the control volume for flow analysis. The louver ventilating window was modeled with a closed volume in the 3D modeling created with CATIA V5 and then imported to STAR-CCM after converting it to an IGS file, which is a standard conversion file format. The imported IGS file was entered as a kind of surface mesh form. Here, the surface mesh size was highly important to create a volume mesh. Thus, the surface mesh was re-created, and the inlet and outlet were extruded to play a role as a wind tunnel of the inlet and outlet, thereby creating a volume mesh by finally applying the aforementioned Hexahedral+Trim mesh.

The louver inlet and outlet were extruded by six and 10 times the hydraulic radius, respectively, to ensure the stability of the flow. The reason for the longer extrusion at the outlet was to ensure a sufficient length at the outlet; the flow was expected to have deviated in a one-sided direction because the vane installed inside the louver had an angle in the middle.

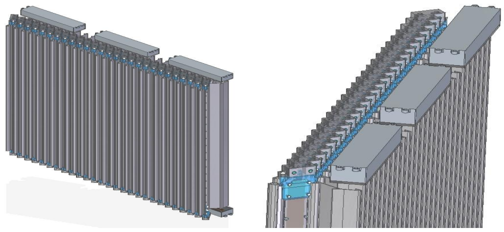


Fig. 1 Louver shape for polar ship

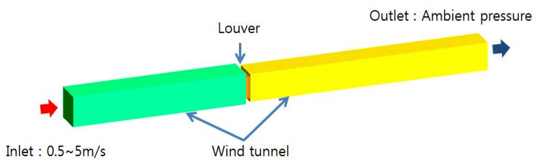


Fig. 2 Boundary conditions of ventilation louver

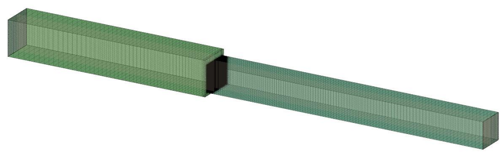


Fig. 3 Control volume of ventilation louver

3. Flow analysis method

Fig. 4 shows the static pressure distribution at the center cross-section of the louver. It verified that the pressure increased at the inlet side of the louver as the flow rate increased from 0.5 m/s at the inlet. This was because the louver played a role as a kind of resistor at the wind tunnel, thereby increasing the pressure at the inlet. The vane's shape installed inside the louver was designed to modify the flow direction in the middle area so that flow stagnation occurred at the inlet side, thereby increasing the pressure.

In contrast, as the flow was discharged to the outlet side, the flow rate increased so that negative pressure occurred, thereby causing a sudden drop.

In addition, it was verified that the pressure was recovered as the flow completely passed through the louver to the outlet of the wind tunnel.

Fig. 5 shows the static pressure distribution at the surface of the louver. It verified a significant increase in pressure at the vane surface in the inlet side of the louver as the flow rate in the inlet increased.

Fig. 6 shows that pressure drop occurred at the inlet and outlet of the louver. The pressure drop occurred significantly more as the flow rate in the inlet increased. In addition, it showed an increasing trend with the quadratic function characteristics as the pressure drop increased.

Fig. 7 shows the velocity distribution with regard to the flow rate at the center cross-section of the louver. It was verified that as the flow rate in the inlet increased, the velocity at the vane installed inside the louver increased, and flow distribution occurred according to the installation angle of the vane. The louver cross-section verified that the flow was deviated to the Vane-A side due to the flow travel direction, as shown in Fig. 8, thereby discharging at a fast rate.

Fig. 9 shows an enlarged view of the louver side, in which the flow direction was changed to the left at the outlet of the louver. This was due to the bending of the flow direction to the left side due to the vane's effect. It also verified that there were a total of three curved portions in the passage inside the louver, and the velocity differed at the curved portions.

Fig. 10 shows the velocity distribution found at the passage through the louver. Fig. 10 shows the sensing position for acquiring the velocity distribution inside the louver.

It shows the velocity distribution according to a change in the flow rate in Sensing position-1, which verifies that the velocity decreased at the inlet of the louver. It confirms that as the pressure increased, the velocity decreased, and the flow rate increased as the flow passed through the 90-mm position of the vane, where the vane’s angle was changed. This indicated that the flow rate increased while passing through the vane, thereby reducing the velocity. As the flow channel was changed, the velocity was reduced, at which the static pressure distribution started to decrease.

It shows the velocity distribution according to a change in the flow rate at Sensing position-2, which is the velocity distribution at the center of the louver. It verifies that at this position, the velocity distribution was slightly higher. In contrast, the change in velocity at the 90-mm position dropped slightly. In addition, the velocity tended to increase again at the outlet position of the louver. This was because the center portion was less affected by the flow resistance than the edge portion. This was because there was a wall in the louver at the edge portion, which incurred a pressure increase as a resistance factor.

It shows the velocity distribution according to a change in the flow rate at Sensing position-3. Note that the velocity tended to be the lowest at the 80-mm and 90-mm positions. This indicates that the velocity distribution was high as it was closer to the outlet side, which was due to the acceleration of the flow since the flow was directed to Sensing position-3 as the vane's angle was changed.

Fig. 12 shows the stream line when the inlet flow rates were 0.5 m/s and 5 m/s. The figure verifies that the flow was introduced in a straight direction from the inlet. Then, the flow was passed through the louver and deviated to the left side. In addition, the figure verified that the flow re-circulation section appeared due to the wall side of the wind tunnel on the right side and velocity difference on the left side.

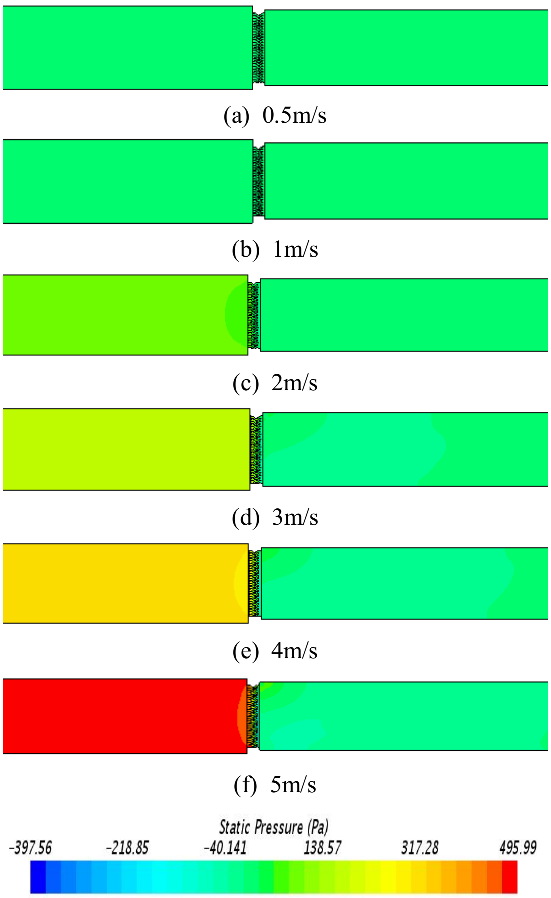


Fig. 4 Comparison of static pressure at center section

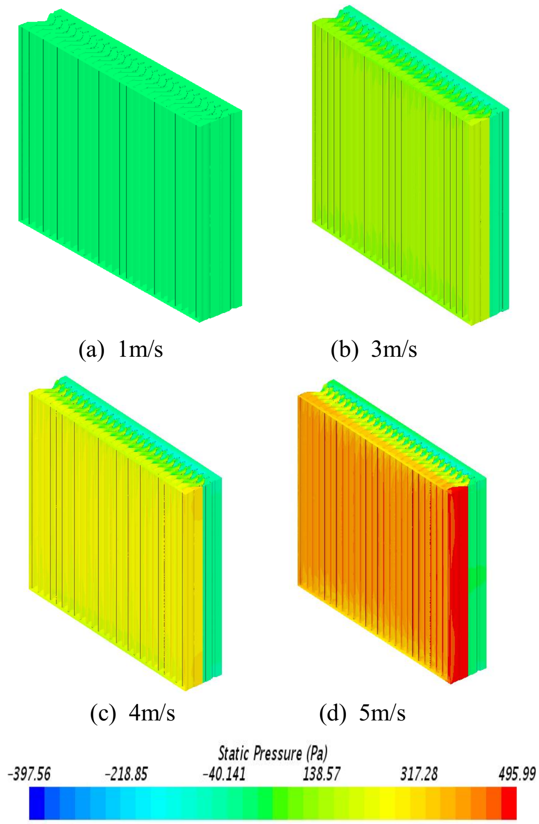


Fig. 5 Comparison of static pressure at ventilation louver surface

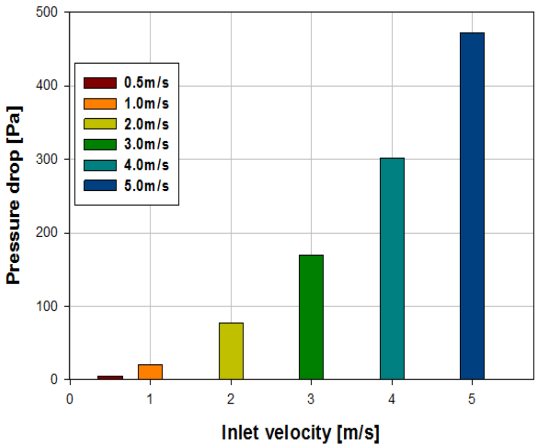
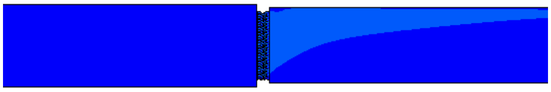
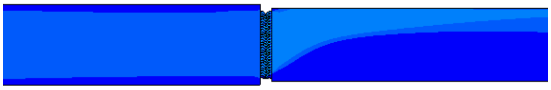


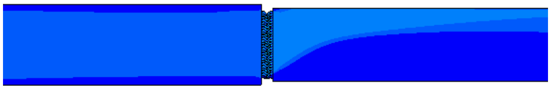
Fig. 6 Result of pressure drop at ventilation louver



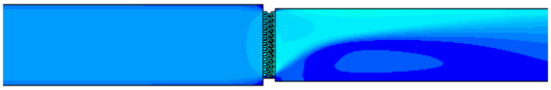
(a) 0.5m/s



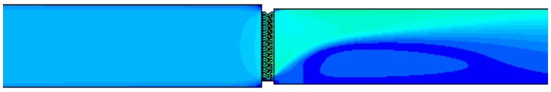
(b) 1m/s



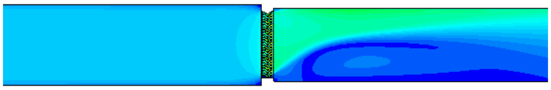
(c) 2m/s



(d) 3m/s



(e) 4m/s



(f) 5m/s

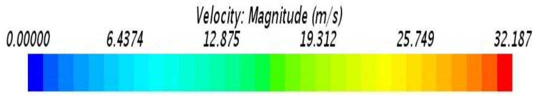


Fig. 7 Comparison of velocity magnitude at center section

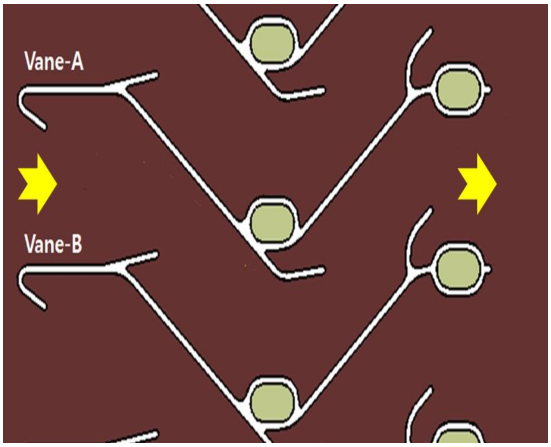
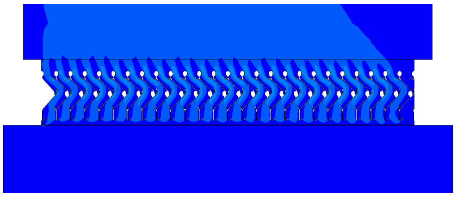
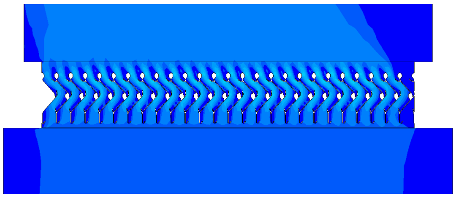


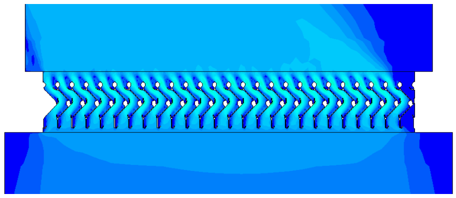
Fig. 8 Shape of louver section



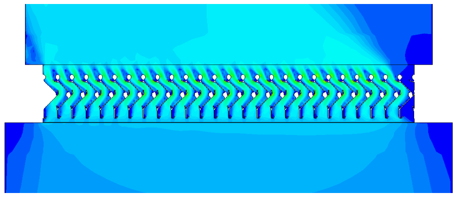
(a) 0.5m/s



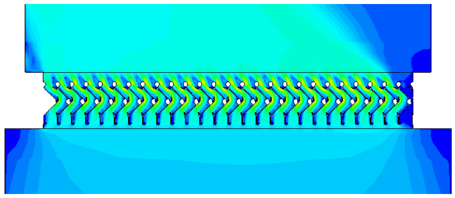
(b) 1m/s



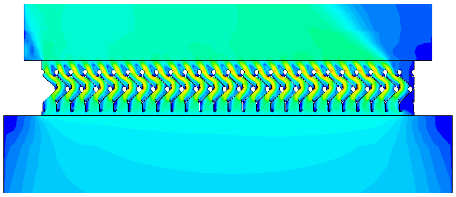
(c) 2m/s



(d) 3m/s



(e) 4m/s



(f) 5m/s

Fig. 9 Comparison of velocity magnitude in louver section

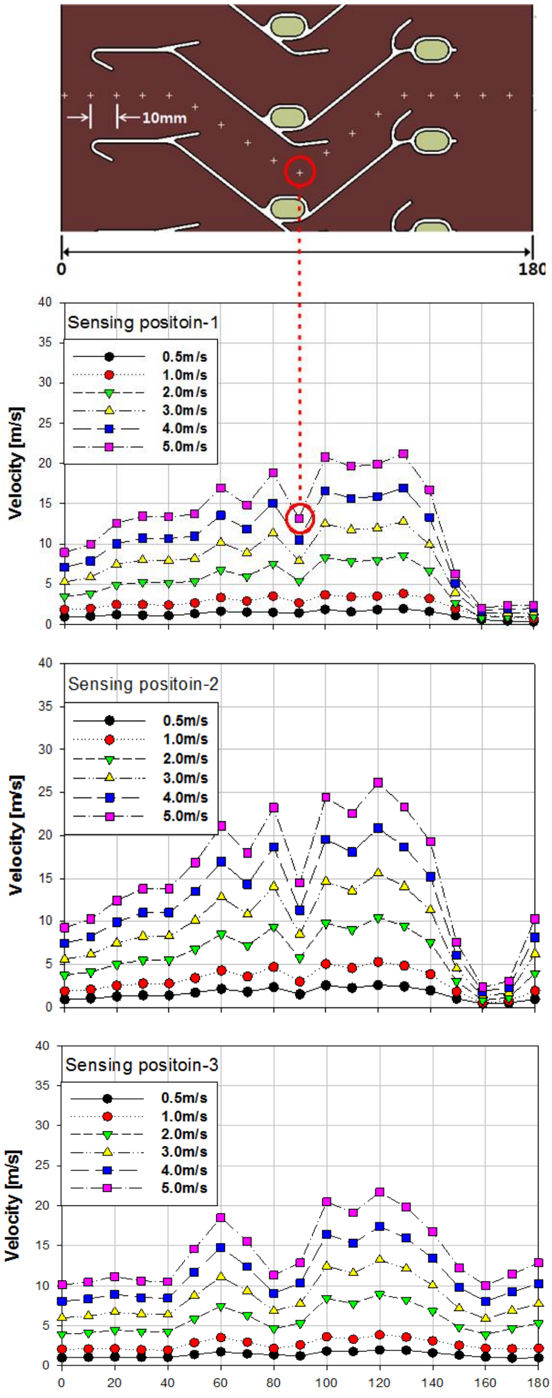


Fig. 10 Comparison of velocity distributions in louver

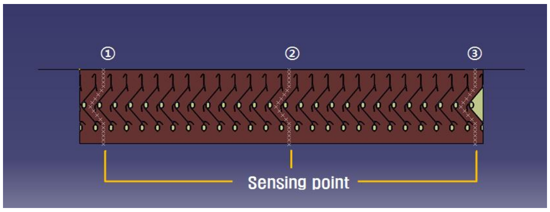
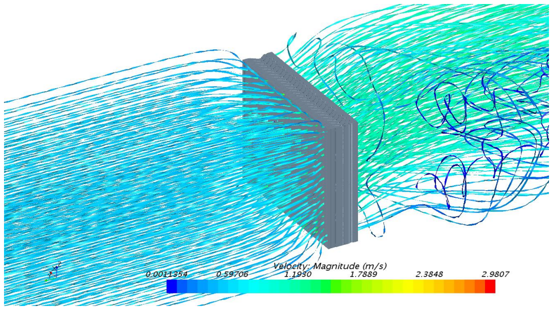
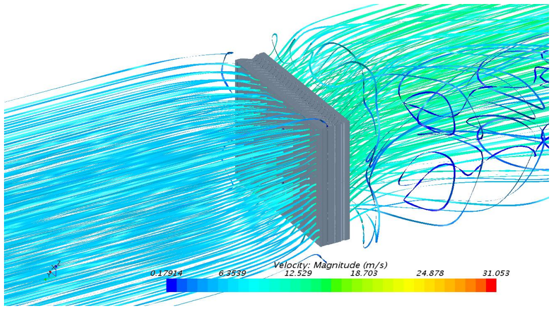


Fig. 11 Sensing positions in louver



(a) 0.5m/s



(b) 5m/s

Fig. 12 Comparison of streamline

4. Conclusions

This study conducted a flow analysis of the louver installed in the ventilation window in a ship, and the following conclusions were derived.

The pressure drop occurred significantly more as the flow rate in the inlet of the louver increased. In addition, it showed an increasing trend with the quadratic function characteristics as the pressure drop increased.

It was verified that the velocity dropped at the inlet of the louver. This was due to the pressure increase, which decreased the velocity, and the flow rate increased while passing through the vane.

Acknowledgment

This study was supported by the 2015 Economic Cooperation Industry Promotion Business (Region-led R&D) (R0004882).

REFERENCES

1. Seo, J. H., Kim, B. T., Chin, D. H., Yoon, M. C. Kwak. J. S., "Comparison of the Contact Characteristics for Sealing strips of the Tsunami Damper", Journal of the Korean Society of Manufacturing Process Engineers, Vol. 14, No. 1, pp. 21-28, 2015.
2. Jang, S. C., Jung, W. B., Yi, C. S., "A Study on Performance Improvement of Gear Type Vane Damper in Marine/Offshore FD Fan", Journal of the Korean Society of Manufacturing Process Engineers, Vol. 14, No. 2, pp. 7-13, 2015.
3. Yi, C. S., Chin. D. H., “Numerical Analysis on the Development of a Ventilation Opening Louver for Marine”, Journal of the Korean Society of Manufacturing Process Engineers, Vol. 16, No. 1, pp. 30-35, 2017.
4. 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" Journal of KSME B, Vol. 30, No. 2, pp. 177-185, 2006.
5. Yi, C. S., Jang, S. C., Choi, J. H., "Numerical Analysis on Hood Shape Improvement of Local Ventilation System", Journal of ACRE, Vol. 21, No. 4, pp. 260-265, 2009.
6. Park, J. Y., Yi, C. S., Chin, D. H., “Numerical Analysis on the Development of Shut off Damperfor Tsunami at Nuclear Plant”, Jorunal of KSMTE, Vol. 23 No. 5, pp. 471-477, 2014.
7. Yi, C. S., "Numerical Analysis of the Kitchen Hood Ventilation System for Marine Environment", Journal of the Korean Society of Manufacturing Process Engineers, Vol. 14, No. 5, pp. 96-101, 2015.