Technical Paper

Investigation of Air Flow passing through Louvers

Toshikazu Nakanishi
Tamotsu Nakamura
Youichirou Watanabe
Katsumasa Handou
Takahiro Kiwata

The structure of louvers installed in the air intake and exhaust unit in the exteriors of the engine compartment of the vehicles is used for design, safety, noise reduction and other purpose. However, on the other hand, the louvers obstruct the flow of cooling air required by the radiator and other components. To know pressure loss characteristics as a parameter of the louver angle, wind tunnel measurement was performed in an effort to obtain design direction for louver shapes that minimize pressure loss when air passes through louvers while satisfying necessary functions. CFD (Computational Fluid Dynamics) was applied after verifying the measurement results. Additionally, the mechanism of pressure loss characteristic was studied by visualizing the air flow using LDV (Laser Doppler Velocimetry) and smoke wire method. As a result, it was found that CFD could have good agreement with measurement of pressure loss characteristic against louver angle variations. Visualization of air flows showed separation vortexes between louvers, indicating that narrowing the flow path caused by a separation was one of the reasons for pressure loss.

Key Words: *Louver, CFD, ventilation resistance, LDV, velocity vector, smoke wire, joint research*

1. Introduction

The louvers are installed in air inlets in the exteriors of vehicles, screening fences of buildings, air conditioner air outlets and other places and are an array of 2D fins that are used to shut out foreign matter and lights, to attenuate noise, to deflect flows and for other purposes while allowing air to pass (**Fig. 1**). Louvers used in the exteriors of outside air inlets of vehicles are varied in shape and serve the purposes of letting in air and protection of the inside.[1] Air that is introduced through louvers of vehicles cools the radiator that acts as a heat exchanger. The cooling efficiency of the heat exchanger is affected by airflow introduced through the louvers. A low pressure loss of the louvers will allow a reduction in fan revolution speed, leading to low noise and energy saving.

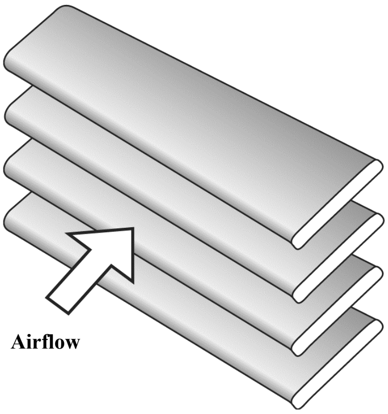


Fig. 1 Ordinary shape of louver

In this research, a 2D model was fabricated for the model of the louvers used in the air intake and exhaust unit in the exteriors of the engine room of the vehicles and studied the characteristics of airflow passing through the louvers by using wind tunnel. In the experiments, pressure losses as an indicator of ease of airflow when passing through the louvers, visualization of airflows around the louvers using the smoke wire method, separation of airflow around the louvers and other phenomena were measured by LDV (Laser Doppler Velocimetry). Additionally, analysis by CFD (computational fluid dynamics) of flow fields of the entire louvers was performed using CFD++, a CFD code sold on the market, under the same conditions as those in the experiment. CFD allows determination of the behaviors of entire flow fields and can easily analyze conditions that are difficult to measure in experiments and can easily change the model shape. This research captured the behaviors of airflows while air passed through the louvers and analyzed the generation mechanism of pressure losses, to obtain design guidelines for a louver shape that would reduce pressure losses during the passing of air, to optimize the louver shape. Numerical forecasting accuracies were verified by comparing results of the experiment and CFD.

2. Experiment Apparatus and Method

2.1 Wind tunnel system

A wind-tee type, low-turbulence wind tunnel (Model No. S1803-2 manufactured by Kanomax Japan Inc.) was used in this research. The cross section of the measuring part was 1200 x 300mm, non-uniformity of velocity distribution at the wind velocity of 10m/s was within ±1% and turbulence intensity was 0.3% or less. **Figure 2** shows a photo of the LDV and measuring part when the louvers are installed. A schematic diagram of a louver model in a flow path is shown in **Fig. 3**. "X" marks in the illustration show the positions where pressure was measured. Six positions were provided on the upstream side of the louvers and ten positions, on the downstream side. Pressure differences in front and rear of the louver model were measured by a Gettingen manometer.

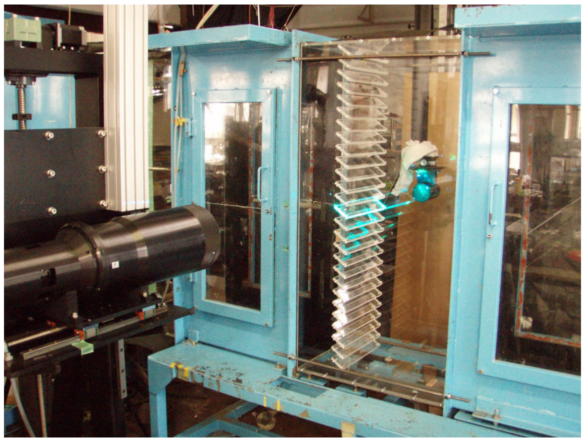


Fig. 2 Measuring section of wind tunnel system

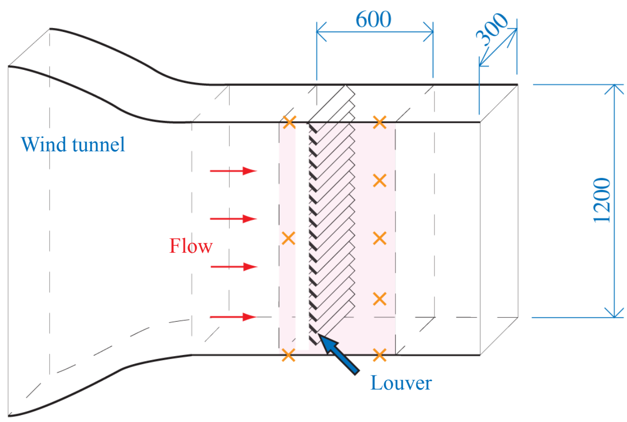


Fig. 3 Sketch of wind tunnel system

2.2 Sample model

A planar louver board was used in the experiment as a louver model. Three types of this louver board were used, changing the louver angle to 45, 35 and 25° as shown in **Fig. 4**. At these angles, the projection width to the bottom side was common, 50mm. The louvers were placed so as to create no gaps relative to the projection to the fore, making the rear invisible when viewed from the front. As the louver angle decreases as illustrated below, the spaces between the louvers become small and the total number of louvers increases.

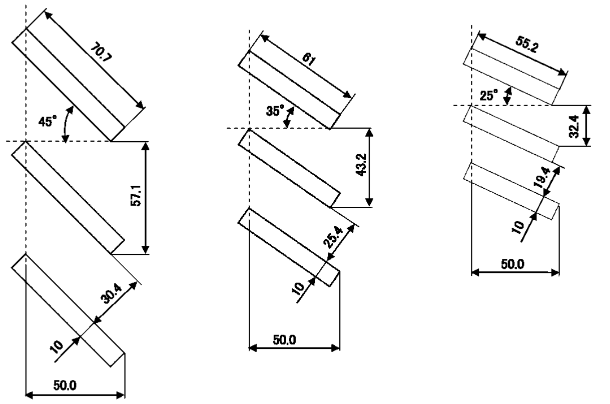


Fig. 4 Louver models

2.3 LDV (Laser Doppler Velocimetry) system

Velocity measuring means are grouped generally into contact and non-contact types. The contact type that is widely used is pitot tubes, the wind mill type called Biram's vane anemometer and hot wire type. LDV (Laser Doppler Velocimetry) is known as the non-contact type. Recently, PIV (Particle Image Velocimetry) is also commercialized. Among these wind anemometers, meters that can measure 3D velocity vectors are limited. The contact type includes porous pitot tubes[2] and the type that rotates a hot wire.[3] Multiple laser beams are emitted in LDV of the non-contact type to measure 3D velocity vectors.

In this research, it was difficult to insert a contact-type probe into a tilted space of about 50 x 20mm to measure velocity vectors between louvers and the probe itself was feared to affect the airflow even if a probe could be inserted. Under the circumstances, measurement was performed by a non-contact 3D LDV system.

The LDV used in the research telemeters the flow velocity using an argon laser. A three-dimensional flow velocity (velocity vector) in the focal position of the lens in the probe can be measured. Beams are emitted to the same focal point from the probe, laser beams reflect on micro particles emitted by the tracer and velocity is measured by the Doppler effect using particle velocity. The probe can be moved in parallel in three dimensions by the traverse system. The measuring system is shown in **Fig. 5**.

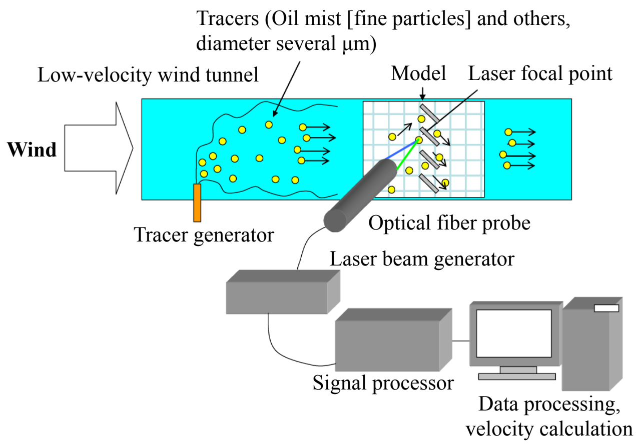


Fig. 5 LDV measuring system

(Measurement Principle)

As illustrated in **Fig. 6**, two laser beams are crossed to generate a fringe pattern with a contrast. Scattered lights generated when particles pass through the fringe pattern are detected by a position sensitive device (PSD) inside the probe. Two laser beams perform 1D measurement and six laser beams in three colors of different frequencies (green, blue and purple) are used in 3D measurement.[4)-6)] The particle sizes are relatively large and forward scattering becomes intense. Measurement sensitivity is anticipated to enhance by detecting forward scattering. A corner cube prism is installed on the opposite side of the probe and forward scattering is measured by the probe using reflected lights of the prism.

The fringe spacing (d_f) is determined by the wavelength of a light and half of the crossed axes angle and is expressed by the following formula.

$$d_f = \frac{\lambda}{2\sin(\kappa)} \qquad (1)$$

d_f: fringe spacing
λ: wavelength
k: half of crossed axes angle

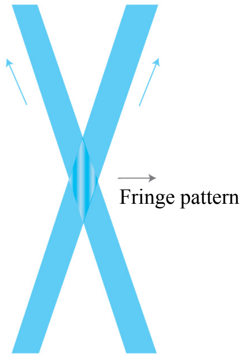


Fig. 6 Fringe pattern by beam crossing

The Doppler frequency (f_d) at this time can be calculated as follows relative to the particle velocity U.

$$f_d = \frac{U}{d_f} \tag{2}$$

(Measurement of positive and negative velocities by frequency shift)

A velocity can be calculated when a fringe spacing is calculated and a Doppler frequency relative to the fringe is measured. However, this calculation calculates only the absolute value of a velocity and its positiveness or negativeness cannot be calculated. For this reason, in measurement by LDV, one beam is "shifted" using an acoustooptic modulator called a Bragg cell and the frequency is varied slightly.[4) - 6)] This enables measurement of a reverse flow. **Figure 7** illustrates the concept of this shift. By shifting the frequency, the Doppler frequency corresponds to a velocity of 1:1 even in a negative velocity region of a certain range. This shift portion is subtracted in the signal processing stage. The relationship between shifting and a fringe pattern is shown in **Fig. 8**. By shifting, the fringe pattern moves from a shifted beam to a beam, which is not shifted, at the frequency of the shift portion, adding the Doppler frequency for the shift relative to the particle velocity. In other words, measurement within the range that does not exceed the shift frequency will be possible even if particles flow in the reverse direction.

In this research, the shift and measurement frequency range were set taking into consideration the main flow velocity and reverse flow velocity in three directions of u, v and w.

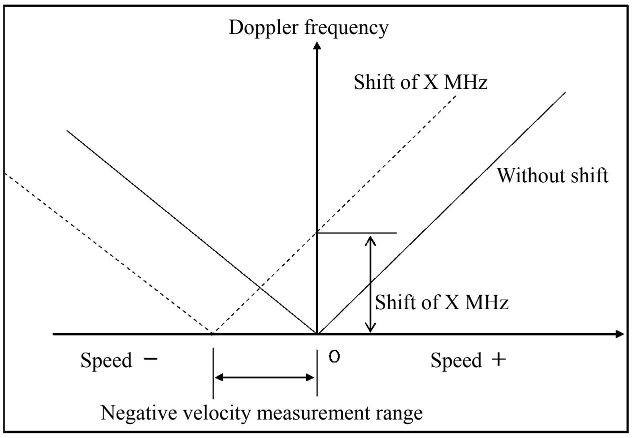


Fig. 7 Concept of shift

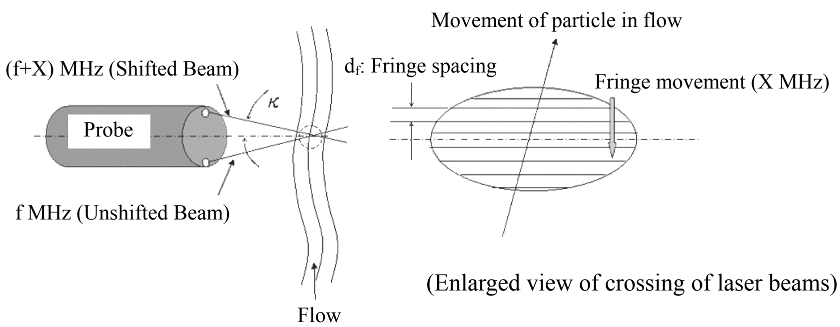


Fig. 8 Movement of fringe pattern

(Rationalization of amount of tracers)

An important factor in measurement is effective feeding of fine oil and fat particles called tracers that generate scattered lights. The tracer generator that was used was a Ruskin-type fumigator and settings were made to obtain adequate particles in the measuring parts by optimizing pressure, nozzle installation position and other factors. **Figure 9** shows converging of laser beams in a laser measuring part. Laser beams can be viewed by the naked eye as shown in the photo if there are sufficient tracers inside the measuring part.

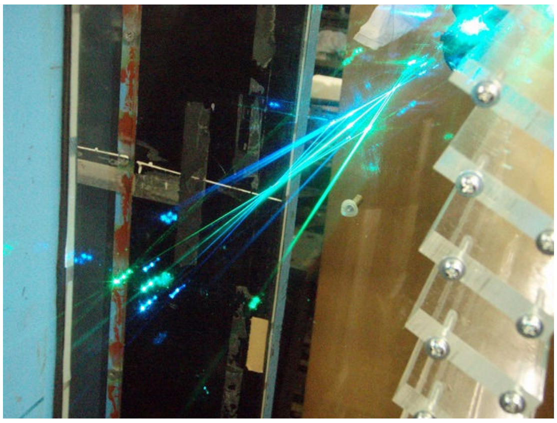


Fig. 9 Beam crossing of LDV laser

3. Analysis by Simulation (CFD)

3.1 CFD model

A 2D analysis model was used in CFD because of the 2D configuration of the louvers and flow in the wind tunnel. This model offered large advantages in shortening the calculation time and allowing ease of mesh control. The constitutive equation that was used was a compressible N-S equation and a k-ε2 equation model was used as the turbulence model. "CFD++," a CFD software package sold on the market, was used as analysis software.[7)]

The CFD model used in the analysis is outlined in **Fig. 10**. The vertical dimension was 1200mm, which was the height of the wind tunnel, to reproduce wind tunnel conditions in 2D. Therefore, the CFD model had the same number of louvers as in the experiment. As boundary conditions, the inlet was velocity input and outlet, pressure condition. The top, bottom and louvered part had the condition of no-slip walls.

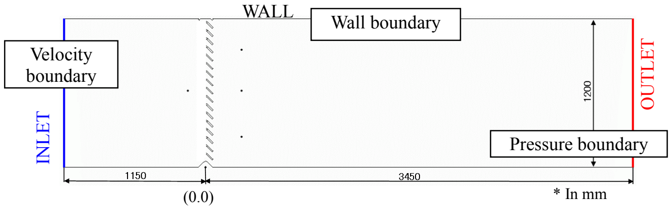


Fig. 10 2D CFD Model

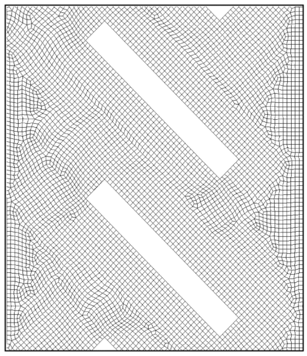


Fig. 11 Meshes in louvered part

The meshes of the louvered part are shown in **Fig. 11**. In the generation of meshes between the louvers, quad (square) meshes were used considering the impact of flows between the louvers on the resistance. Quad meshes allowed a layout of meshes on louver surfaces along airflows. Quad meshes are expected to have a high accuracy compared with triangle meshes.

One problem with mesh size in this case is the size near the walls. Fluid flowing dynamically near a wall is classified into a viscous bottom layer and logarithmic region in accordance with the viscosity effect. Originally, a mesh to reproduce these phenomena must be provided in CFD also. In this case, however, the mesh will be to the order of 10^{-6}m when trying to gain the resolution of viscous bottom layer, and this will not be practical analytically. In CFD, therefore, by putting first mesh in logarithmic region in use of wall rules, a technique to express a flow is used without directly solving the flow of a viscous bottom layer. A dimensionless distance y^+ is used as an indicator for this and the criterion will be to make this a logarithmic region.[8)] A mesh size that will make y^+ 40 or less was set in the experiment and was about 1.5mm near the louvers. Taking the generation of a vortex also into consideration, roughly the same mesh was set as the mesh between the louvers.

4. Experiment, Calculation Results and Consideration

4.1 Result of pressure measurement

The results of pressure measurement and of CFD are shown in **Fig. 12**. The pressure in front of the louvers and differential pressure in an upper measurement point in the rear were most prominent as pressures and these pressures were compared as representative pressures. The axis of abscissas plots the wind velocity in front of the louvers and the axis of ordinates plots the differential pressure made dimensionless by the reference pressure.

The measurement results show that the differential pressure is almost a quadratic function of the velocity and that flows passing through the louvers also have a resistance characteristic of ordinary flows. When the angle becomes small, that is, when the angle to flows flowing through the louvers becomes small, the pressure differences tend to decrease. The louvers are installed to eliminate projected gaps in front and the area occupied by the louvers increases in the cross section when the angle becomes small, while the flow path becomes narrow (See **Fig. 4**). Notwithstanding this, the pressure differential becomes significantly small. **Figure 13** compares the ratios of pressure differences of the louvers based on a pressure difference at 45° and 5m/s as a reference. The pressure difference lowers to 43% at 25°.

The results of CFD also show a similar trend as in the measurement. The difference between CFD and measurement at the absolute value of pressure differences shown in **Fig. 12** is about 13% at 5m/s. As shown in **Fig. 13**, in a ratio that makes 45° as a reference, the difference is about 2% (62% and 64%), indicating that the recent CFD faithfully reproduced differences in characteristics between the louvers.

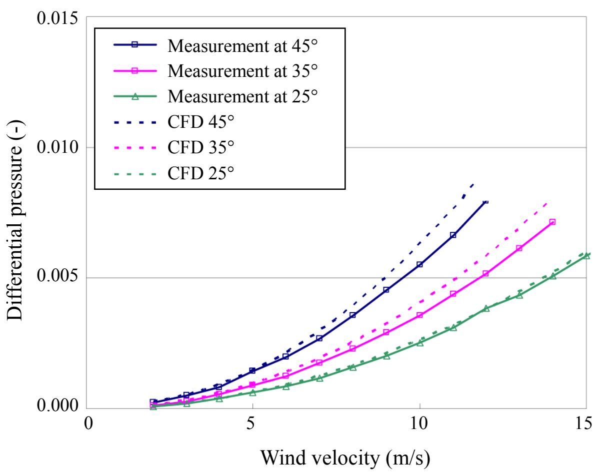


Fig. 12 Pressure loss of each louver

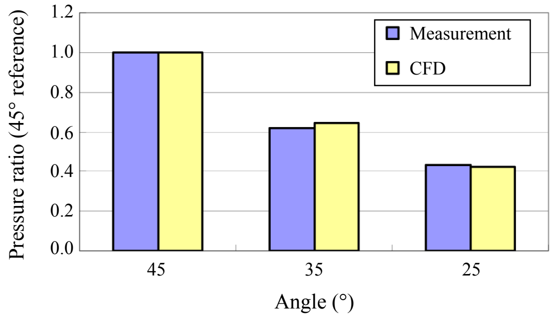


Fig. 13 Proportion of pressure loss lowering

4.2 Results of visualization

Flows around the louvers were examined by the measurement of velocity vectors by LDV and through visualization of flows at a very low wind velocity by the smoke wire method to study the phenomena of flows to pressure losses by the louvers. Additionally, the velocity vectors were indicated by CFD.

1) Visualization by velocity vector measurement by LDV

Figures 14 and **15** show velocity vectors by LDV. The wind velocity was 5m/s on a louver in about the center of the louver board. The diagrams clearly show separation from the tips of the louvers and behavior of the vortex after the separation. The diagrams also show that the vortexes become large, the larger the angle is. It can safely be concluded that the velocity between the louvers increases as the flow path is narrowed by vortexes caused by separation when the angle to the current increases, thereby increasing the pressure differences due to the pressure loss caused by the increase in the velocity.

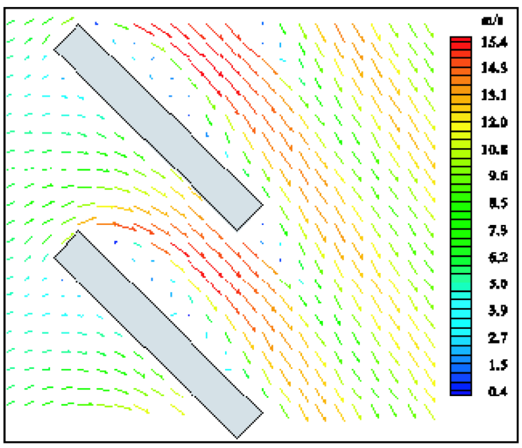


Fig. 14 Velocity vector (45°)

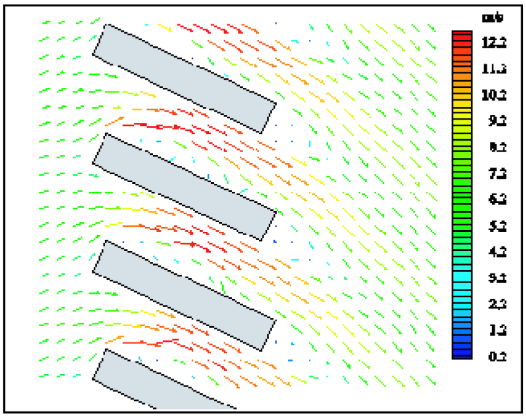


Fig. 15 Velocity vector (25°)

2) Visualization by smoke wire

Measurement of the entire flow field by a velocity meter is time consuming and separation, turbulence, periodicity of after vortexes and other phenomena were checked by visualization observation using the smoke wire method. Flows were visualized by impressing voltage onto a stainless steel wire 0.2mm in diameter stretched in a flow path and by photographing the streaks of smoke generated from liquid paraffin coated on a wire which was heated by the voltage. In this experiment, visualization was performed to determine the condition of a qualitative flow at about 0.2m/s, at which smoke hardly dispersed. **Figures 16** and **17** show the results of visualization.

The visualization shows that separation is taking place from the tips of the louvers at a very low velocity as in the measurement results of LDV and that separation is large at 45°.

Fig. 16 Smoke wire image (45°)

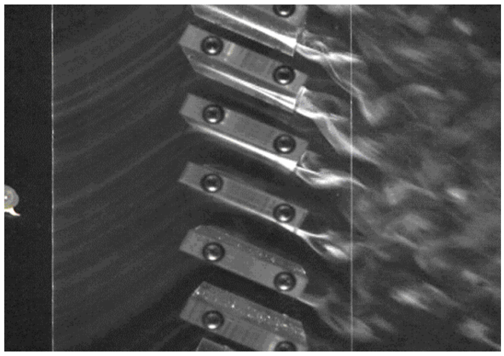


Fig. 17 Smoke wire image (25°)

As one of the features of the smoke wire method, the behaviors of flow fields in a wide area can be determined. **Figure 18** shows the condition in the downstream area that includes the louvers. After passing the louvers, airflows prominently deflect downward, while the upper side flows in reverse.

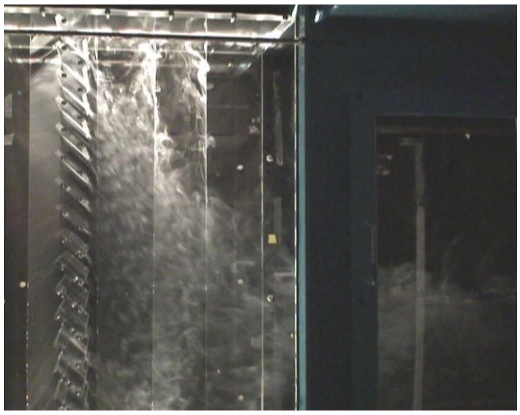


Fig. 18 Entire smoke wire image (45°)

3) Visualization by CFD

Velocity vectors by CFD are indicated by equally dividing the space. As shown in **Figs. 19** and **20**, airflows are separated at the tips of the louvers and vortexes are generated thereafter, which was the phenomenon observed with LDV and with the smoke wire method. As observed in the measurement, vortexes tend to become small, the smaller the angle is.

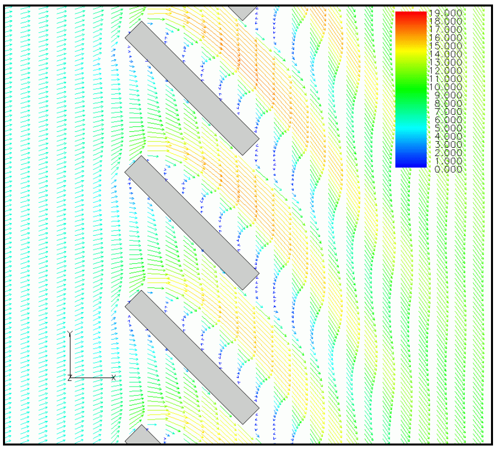


Fig. 19 Velocity vector by CFD (45°)

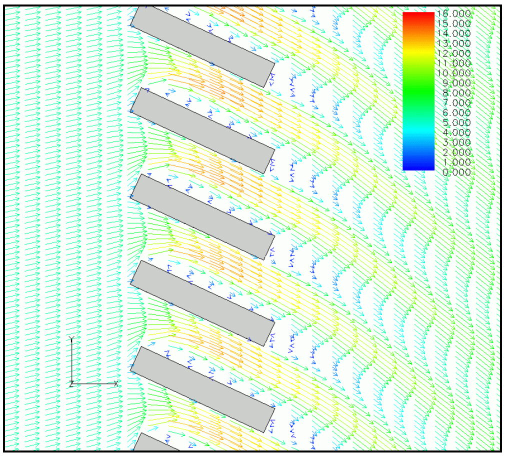


Fig. 20 Velocity vector by CFD (25°)

The behavior of the entire flow is shown in **Fig. 21**. As in the visualization images by the smoke wire method shown in **Fig. 18**, airflows after passing the louvers deflect downward and macroscopic flow fields of the entire experiment system are reproduced by CFD.

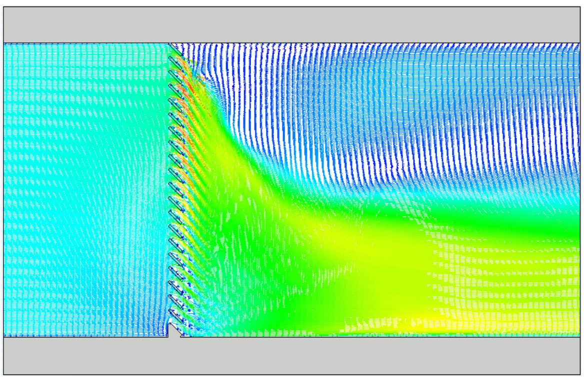


Fig. 21 Entire velocity vectors by CFD (45°)

5. Conclusion

The following knowledge could be gained through this research on airflows that pass the louvers.

1) Pressure loss characteristic varies significantly relative to the louver angle. In this research, pressure losses could be kept to less than half by varying the louver angle 20°.
2) Through visualization experiments by LDV and other means, variations of channel resistance by the louver angle can be attributed to the narrowing of flow paths between the louvers by separation vortexes from the tips of the louvers.
3) CFD allows reliable forecasting of flow behaviors in shapes of louvers as in this research and its availability as a design tool has been verified.

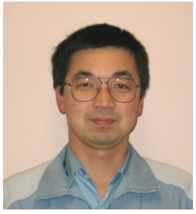
6. Acknowledgment

The authors thank Mr. Takashi Tanaka, a former student at Kanazawa University, for his cooperation in the experiments, data analysis and other functions, as well as Ms. Sakura Nakai of MEITEC Fielders, Inc. for her cooperation in creating the CFD model, in connection with this research.

References:

1) Mitsuo Yabe and Kuniaki Nakata, "Noise Reduction of Hydraulic Excavator," KOMATSU TECHNICAL REPORT Vol. 45, No. 2, pp.12 - 23 (1999).
2) Masakazu Kobayashi and Naoto Hisanaga, "Flow Velocity Measurement Method by Porous Pitot Tube," KOMATSU TECHNICAL REPORT Vol. 37, No. 127, pp.3 - 14 (1991).
3) Toshikazu Nakanishi, "Analysis of Flow Inside Engine Room - Report 2," KOMATSU TECHNICAL REPORT Vol. 45, No. 2, pp.33 - 45 (1999).
4) TSI, Inc., FIND for Windows Version 1.4, Reference Manual, TSI Incorporated, St. Paul, MN, USA, 2000.
5) TSI, Inc., Model 9230 COLORLINK Plus Multicolor Receiver, Instruction Manual, TSI Incorporated, St. Paul, MN, USA, 1999.
6) TSI, Inc., Model 0833 5-Beam Fiberoptic Probe, Instruction Manual, TSI Incorporated, St. Paul, MN, USA, 2001.
7) Atsushi Itoh, Toshikazu Nakanishi, et al., "High-speed Computational Fluid Dynamics (CFD) by Parallel Computer," KOMATSU TECHNICAL REPORT Vol. 51, No. 156, pp.8 - 14 (2005).
8) Chuichi Arakawa, Computational Fluid Dynamics, Tokyo University Press.

Introduction of the writers



Toshikazu Nakanishi

Doctor of Engineering.
Entered Komatsu in 1986.
Currently assigned to the Research and Development Center, Defense System Division.

Tamotsu Nakamura

Entered Komatsu in 1975.
Currently assigned to the Research and Development Center, Defense System Division.

Youichirou Watanabe

Entered Komatsu in 1982.
Currently assigned to the Corporate Audit Office.

Katsumasa Handou

Entered Komatsu in 1970.
Currently assigned to Komatsu General Services Ltd.

Takahiro Kiwata

Associate professor (Doctor of Engineering)
Currently assigned to Fluid Dynamics Research Laboratory, Department of Mechanical Systems Engineering, Faculty of Engineering, Kanazawa University.

[A few words from the writers]

This research was undertaken jointly with Kanazawa University as a companywide industry-academia collaboration project. The research could be completed efficiently through whole support of the university and Komatsu. Aside from research problems, those at the university and at Komatsu were plagued with administrative and other problems that had to be solved. Nevertheless, meaningful time was spent in interfacing with the professor and students of Kanazawa University in the free and candid atmosphere of the university. It will be good for development engineers and designers of companies to occasionally engage in work with people in outside research institutions such as universities, moving away from company work, to not only gain knowledge, but also to enhance flexible conception and motivation. It is hoped that more joint research of this kind will be undertaken actively.