Velocity Measurement Inside and Outside a Cross-Ventilated Building by Means of PIV

Hisashi KOTANI [1], Tomohiro KOBAYASHI [2], Kaori ASAI [3], Yasue TANAKA [4], and Yasuhiro SHIOZAKI [4]

[1] Department of Architectural Engineering, Osaka University, Suita 565-0871 Osaka, Japan
[2] Dept. of Architecture and Urban Design, Ritsumeikan University, Kusatsu 525-8577 Shiga, Japan
[3] Department of Architecture, The University of Tokyo, 153-8505 Tokyo, Japan
[4] Kanomax Japan, Suita 565-0805 Osaka, Japan

Abstract

Cross-ventilation is regarded to be beneficial control method to obtain thermal comfort in a hot summer without using mechanical devices. Since it is complicated flow phenomenon, details of flow characteristics have not been sufficiently known. The final goal of this work is to establish a new prediction method of flow rate based on energy balance within the stream tube passing through or around the building. To validate numerical results obtained by CFD, they need to be compared with experimental results. This paper presents wind tunnel measurement of flow characteristics inside and outside a cross-ventilated model. Velocity and pressure were measured along the central line of the stream tube for internal flow, and velocity distribution of the external flow was measured by Particle Image Velocimetry (PIV).

Keywords: Cross-Ventilation, Wind Tunnel Test, Particle Image Velocimetry (PIV)

Introduction

The use of renewable energy sources to control internal environment has been attracting practical and academic attentions. Wind-induced cross-ventilation has been beneficial method to moderate hot and humid environment in summer time. To design a building where cross-ventilation functions well, flow characteristics such as flow rate, flow pattern and flow movement must be considered in advance. As Ishihara [1] pointed out, the conventional prediction method of the flow rate based on the orifice equation cannot work for the large opening. This is due to that, for cross-ventilation phenomena, the resistance coefficient of an opening cannot be precisely estimated by the chamber method and the driving pressure cannot be altered by wind pressure coefficient either (see Kobayashi et al. [2]). Ishihara [1] explained this problem as *Interference* of openings and tried to evaluate the overall resistance by introducing *Interference coefficient*. Sandberg [3] introduced geometrical parameter named Catchment area, which can also be regarded as dimensionless flow rate, and showed the relationship with porosity defined as opening area divided by wall area. Kotani and Yamanaka [4] followed Ishihara′s interference coefficient and also proposed a prediction method where vectors normal to the opening and parallel to the wall are added to obtain overall resistance coefficient. Murakami et al. [5], Kato [6] showed the concept of an alternative prediction method based on energy balance inside the selected stream tube passing through/around the building (i.e. power balance model). Axley et al. [7] also showed almost the same principle and formulated the multi-zone flow network model. These prediction concepts seem to be rational because of considering actual phenomenon. However, details of the flow characteristics in the stream tube have not sufficiently been clarified. The final goal of this work is to establish a new prediction method based on energy balance inside the stream tube. To analyze stream tube Computational Fluid Dynamic (CFD) seems to be useful.

The authors have studied power transportation within stream tube for a single room model (see Kobayashi et al. [8]). In this paper, a building model supposed to be composed of nine single-room residences is analyzed. Given the validation of the numerical results that will be obtained by CFD, the flow quantities must be known in advance by means of wind tunnel experiment as the correct value of the calculation. Nishimoto et al. [9] investigated the flow characteristics of a single cross-ventilated room model by means of Particle Image Velocimetry (PIV) system and CFD. This paper pays attention to the airflow around a cross-ventilated building, which includes the complicated flow field as separation and relatively large wake region. It focuses on both internal and external flows of the building model. As for the former, velocity and static pressure are measured along the central line and turbulent kinetic energy is also estimated by anemometer and pressure tube, and the latter, 2-D velocity distribution are obtained using PIV system. In addition to providing qualitative information of the cross-ventilation flow field, these results are to be compared with CFD results that have been obtained in the following work presented by Asai et al. [10].

Method

A closed-circuit wind tunnel in Osaka University whose length, depth, height of the working section was 9.5m, 1.8m, 1.8m respectively was used for the measurement. Fig.1 shows the studied building model which is supposed to be composed of nine one-room residences where central one is provided with two openings on opposite sides. Here, shaded area indicates the attachable orifice plates provided with an opening of which resistance coefficient based on the chamber method is already known (see Kotani and Yamanaka [4]). As studied cases, the side length of the openings (L) was changed as 15, 30, 45, 60, and 90 mm. To analyze fundamental cross-ventilation flow and to see its variation corresponding to the opening size, the test model was located at the center of the wind tunnel with the wind direction perpendicular to the opening. The test model was exposed to a free flow of 10 m/s to keep Reynolds number sufficiently high, where turbulent intensity was approximately 1.0 %. As the reference static pressure in the wind tunnel, that of a Pitot tube installed at the location of 300 mm away from the wall of the wind tunnel was adopted.

As an analysis of the stream tube passing through the room, velocity and static pressure were measured along the central line of the openings. The velocity measurement was conducted by using I-type hot wire anemometer with its sampling frequency and sampling time 1,000 kHz and 30 seconds respectively considering estimation of the turbulent statistics. As for the static pressure measurement, a handmade static pressure tube was used with its sampling frequency 100 Hz for 30 seconds. Both probes are depicted in Fig.2. The probe was attached to an arm provided with a support pole fixed to a 1-D traverser. Velocity and static pressure were measured between the points of X=-150 and X=1100 mm basically at every 10 mm. In the vicinity of two openings, however, this interval was changed to be 5.0 mm assuming the abrupt variation in pressure and velocity. Fig.3 shows the experimental set-up of the measurement.

Analysis of the external stream tube passing around the building was done using PIV system. PIV is a measurement technique obtaining an instantaneous velocity by detecting the length of spatial shift of seeding particles between two photographs taken within a known time interval. Fig.4 illustrates the experimental set-up. A smoke generator was used and the seeding particles were injected into the wind tunnel through the nozzle. A double-pulse Nd : YAG laser was located outside of the wind tunnel and a laser sheet was radiated to the central section of the model horizontally. A CCD camera was installed above the model. Time interval of two pulses were set at 200 μs with a frequency of 4.0 Hz for 25 seconds. Thus, 100 pairs of photographs were taken to obtain time-average velocity by exactly synchronizing with laser pulses under the control of PIV program operated by PC. Since the wake region was supposed to be large due to the relatively large façade area, both model vicinity and the wake regions were analyzed separately by taking photograph for each region as shown in Fig.5. In estimating velocity, FFT cross-correlation method (Willert [11]) was applied. In addition, to improve the statistical reliability and spatial resolution, the recursive correlation method (Hart [12]) was also adopted. Table 1 summarizes the setting of PIV measurement.

Results and Discussions

Flow quantities along central line

Fig.6 shows the static pressure, X-component of the velocity, and turbulent kinetic energy along the central line of the openings. Shaded area indicates the model and X-axis is the distance from inlet opening divided by the model length. Static pressure was expressed as dimensionless pressure by divided by the total pressure far upstream of the model. The velocity is also dimensionless value where reference velocity is obtained by Pitot tube assumed to be the same as approaching flow. To estimate turbulent kinetic energy, each component of fluctuating velocity was assumed to be the same as;

$$k = \frac{1}{2}(\overline{u'^2} + \overline{v'^2} + \overline{w'^2}) = \frac{3}{2}\overline{u'^2}$$

Assuming that I-type hot wire was measuring two components of velocity, turbulent kinetic energy was estimated based on instantaneous velocity reading ($v'_{hotwire}$) as;

$$k = \frac{3}{2}(\frac{1}{2}\overline{v'^2_{hotwire}}) = \frac{3}{4}\overline{v'^2_{hotwire}}$$

In general, the static pressure increases in front of the model according to the Bernoulli's principle, where velocity becomes small due to the impingement on the windward wall. In the case of smallest opening, dimensionless static pressure becomes almost 1.0, which means a large part of dynamic pressure of the approaching flow is converted into static pressure without any energy loss. In the case of L=90 mm, meantime, dimensionless static pressure is approximately 0.8, which indicates that the dynamic pressure remains to some extent at the inlet opening. Correspondingly, the lowest velocity value obtained by hotwire differs according to the opening size. After flowing into the room, lower static pressure can be seen inside the room, and large opening case tends to show lower pressure. Considering this pressure drop is regarded as energy loss and conversion into the dynamic pressure, large openings exhibiting large pressure drop generates larger velocity because energy loss seems to become small. Measured dimensionless velocity also exceeds 1.0 except the case of L=15 mm. The case of L=90 mm shows pressure recovery inside the room. Whereas small openings generate a “jet” flow type, this tendency indicates that the dominant stream tube is formed and its cross-sectional area gradually varies. Moving to the leeward side of the model, any significant difference cannot be seen in static pressure among all cases. On the other hand, importantly, velocity extremely differs. The authors have shown that the negative static pressure could become small on the leeward side when the openings are large, where the wake was blown away and no backflow exists on the leeward (see Kobayashi et al. [2][13]). Here, it must be noted that I-type hotwire anemometer cannot measure negative velocity but absolute value. As referred to hereinafter in discussing PIV results, a backflow in the wake can be seen in all the cases studied here. From these results, it can be said that the static pressure on the leeward depends only upon whether or not the wake is blown away.

To analyze turbulent statistics is also important for understanding phenomenon and for studying accuracy of CFD analyses simulating these experiments. Almost no turbulent kinetic energy is produced on the windward side and no significant difference is seen inside the model either. However, there exists notable difference on the leeward side. Although all results show the sharp peak relatively close to the model after discharged from opening, the location is shifted in leeward direction as openings become large. As is seen in velocity vector plots obtained from PIV, the discharged flow collides to the backflow in the wake here, and consequently turbulent kinetic energy is produced. This peak is followed by a moderate increase, and in all cases, the second peak is seen downstream where X-axis is approximately 5.0. A conceivable phenomenon that caused this correspondence of location is confluence of the external flow passing around the test model. From these outcomes, it is believed that the second peak indicates an end of the leeward wake whose size is determined mostly by façade area regardless of the strength of the discharged flow as far as backflow exists on the leeward side. In such a flow field, static pressure on the leeward side is independent of opening size.

External flow field obtained from PIV measurement

Fig.7 shows the velocity vector plots obtained from PIV measurement, where vector plots are omitted to show only one eighth of them are shown. Although obtained velocity inside a room is not reliable due to the light reflection and transmission, this measurement was conducted to analyze external flow; mainly separation and the wake that could have complicated flow characteristics. The angle of separation around the windward corner of the model seems to be almost the same among all cases, and consequently flow pattern in the external flow becomes almost the same. This is due to that the façade area is sufficiently large if compared with opening area. Therefore, the location of the confluence point which seems to exist further downstream is believed to be almost the same.

As for the discharged flow, the collision to the backflow is clearly seen in the wake, as is not confirmed by hotwire measurement. It can also be confirmed that the X-axis of this collision points corresponds to that of the sharp peak in turbulent kinetic energy. Throughout these wind tunnel tests, the qualitative flow characteristics have clarified except static pressure distribution around the model which is also important for power transportation inside the stream tube. This must also be measured by another method. In order to analyze the stream tube passing through/around the building numerical study seems to be essential because it can determine streamlines and provide spatial distribution of velocity and pressure. The wind tunnel experiments introduced in this paper are also simulated by CFD analysis and those results are compared with experimental results presented in this paper to validate CFD in analyzing the stream tube in future work (see Asai et al. [10]).

Conclusions

The authors aim to establish an improved prediction method of the flow rate in cross-ventilation based on energy balance inside the stream tube passing through/around the building based on CFD analyses. This paper presented the wind tunnel experiments analyzing internal and external flow; i.e. measurement of flow quantities along the central line and PIV measurement around the building model, of which purpose is to obtain correct results to be compared with CFD and to understand fundamental flow characteristics in cross-ventilation. From the viewpoint of latter purpose, the following conclusions have been acquired.

1. The size of the wake generated on the leeward side including backflow is determined mostly by building shape (e.g. aspect ratio of length based on façade area to building depth) and independent of opening size.
2. When there exists the wake including backflow, static pressure on the leeward side becomes almost the same when opening size is different despite that the velocity retained by discharged flow can extremely differs; i.e. it is whether or not the wake exists that determines leeward static pressure.

3. The peaks of turbulent kinetic energy can be good indicators of the collision point of discharged flow and backflow, and confluence point of external flow passing around the building.
4. In the case of large openings, quite large static pressure drop occurs in flowing into a room and the velocity becomes large inside. For such internal flow, dominative stream tube is formed and the conventional prediction of flow rate seems not to work appropriately.

This paper presented fundamental flow characteristics in cross-ventilation phenomenon by means of wind tunnel test. Given the analysis of the power transportation inside the stream tube for a new prediction method, CFD analyses of these experiments were also conducted. As a future prospect of this work, after studying the accuracy of the calculation (Asai et al. [10]), the stream tube is to be determined, and the transported power on the sections of stream tube and energy loss inside the control volume are to be analyzed based on CFD.

Acknowledgment

This work was partly supported by Grant-in-Aid for JSPS Fellows (representative T. Kobayashi (20-912)), Japan Society for the Promotion of Science, 2009.

References

1. Ishihara M., "Building Ventilation Design", Asakura shoten, 1969. (In Japanese)

2. Kobayashi T., Sagara K., Yamanaka T., Kotani H., Takeda S., and Sandberg M., "Stream Tube based Analysis of Problems in Prediction of Cross-Ventilation Rate", *The International Journal of Ventilation,* Vol.7, No.4, pp.321~334, 2009.

3. Sandberg M., "Airflow through large openings – A catchment problem?" *Proceedings of ROOMVENT 2002,* pp.541~544, Copenhagen, Denmark, 2002.

4. Kotani H. and Yamanaka T., "Interference coefficient for discharge coefficient in prediction of wind-driven cross ventilation rate through large openings", *Proceedings of the 29th AIVC Conference,* Vol.3, pp.27-32, Kyoto, Japan, 2008.

5. Murakami S., Kato S., Akabayashi S., Mizutani K., Kim Y. D., " Wind Tunnel Test on Velocity Pressure Field of Cross-Ventilation with Open Windows", *ASHRAE Transactions,* Vol.97, Part 1, pp.525-538, 1991.

6. Kato S., "Flow Network Model based on Power Balance as Applied to Cross-Ventilation", *The International Journal of Ventilation*, Vol.2, No.4, pp.395-408, 2004.

7. Axley J.W., and Chung D. H., "POWBAM0 Mechanical Power Balances for Multi-zone Building Airflow Analysis", *The International Journal of Ventilation,* Vol.4, No.3, pp. 95-112. 2005

8. Kobayashi T., Sagara K., Yamanaka T., Kotani H., Takeda S., and Nishimoto N., "Power Transportation Analysis inside Stream Tube of Cross-Ventilated Building", *Proceedings of the 29th AIVC Conference*, Vol.1, pp.249-254, Kyoto, Japan, 2008.

9. Nishimoto N., Sagara K., Yamanaka T., Kotani H., Kobayashi T., and Takeda S., "Measurement of total pressure, static pressure and wind velocity around cross-ventilated building", *Proceedings of the 29th AIVC Conference,* Vol.1, pp.237-242, Kyoto, Japan, 2008.

10. Asai K., Kotani H., Kobayashi T., Yamanaka T., and Momoi Y., "Experimental Investigation and Accuracy Study of CFD Analysis for Airflow around Cross-Ventilated Building", *Submitted to the 31st AIVC Conference,* Seoul, Korea, 2010.

11. Willert C. E. and Adrain R. J., "Digital Particle Image Velocimetry", *Experiments in Fluids,* Vol.10, No.4, pp.191-215, 1991.

12. Hart D. P. “Super-Resolution PIV by Recursive Local-Correlation”, *Journal of Visualization*, Vol.3, No.2, pp.187-194, 2000.

13. Kobayashi T., Sagara K., Yamanaka T., Kotani H., and Sandberg M., “Wind Driven Flow through Openings – Analysis of the Stream Tube”, *The International Journal of Ventilation*, Vol.4, No.4, pp.323-336, 2006.

Table 1 Setting of PIV

Camera Frame Size	625 mm × 471 mm 1,376 pixel × 1,040 pixel
Program	Davis 7.2
Algorithm	FFT Cross-Correlation Method (Recursive Correlation Method)
Interrogation Window Size	Pass 1 : 64 pixel × 64 pixel Pass 2 : 32 pixel × 32 pixel Pass 3 : 32 pixel × 32 pixel
Overlap	50 %
Total Numver of Vectors	5,590 (86×65)
Time Interval of Pulse	200 µs
Sampling Frequency	4.0 Hz
Sampling Time	25 seconds
Laser Output	50 mJ / Pulse
Seeding Size	0.3 - 1.0 µm

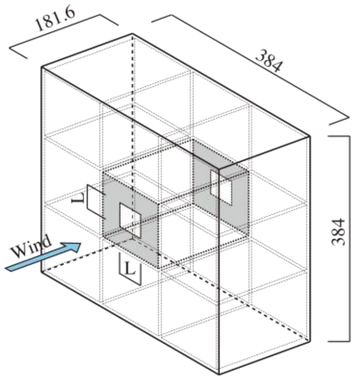
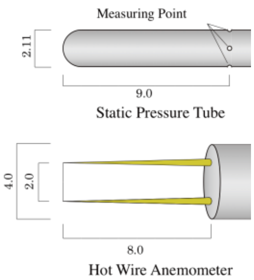


Fig.1 Model configuration



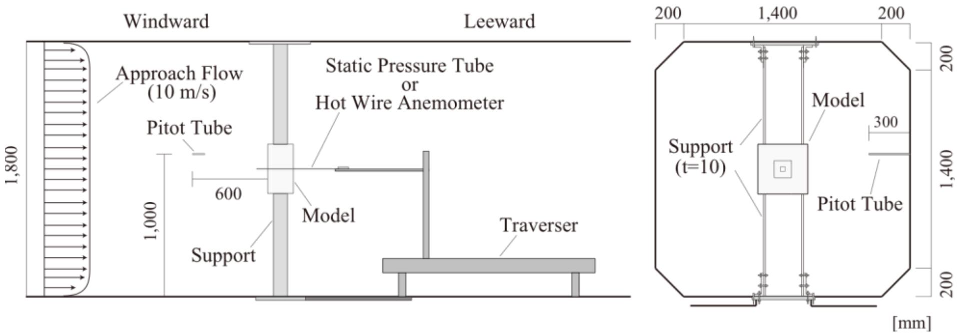
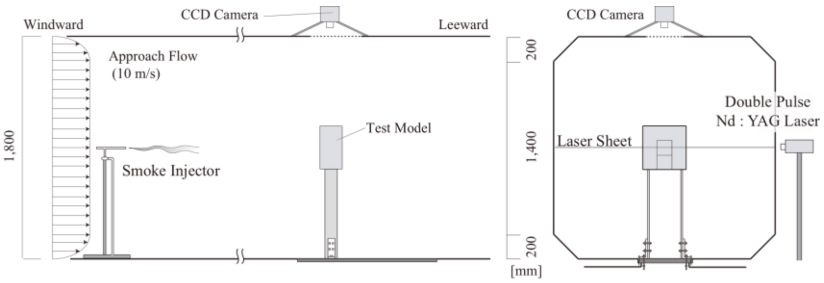


Fig.2 Probe head **Fig.3** Experimental set-up of the measurement along the central line



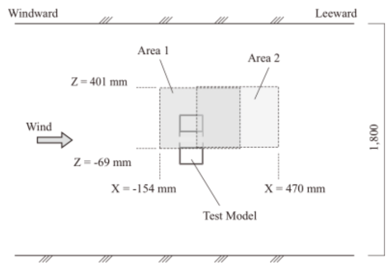


Fig.4 Experimental set-up for PIV measurement **Fig.5** Measured regions in PIV

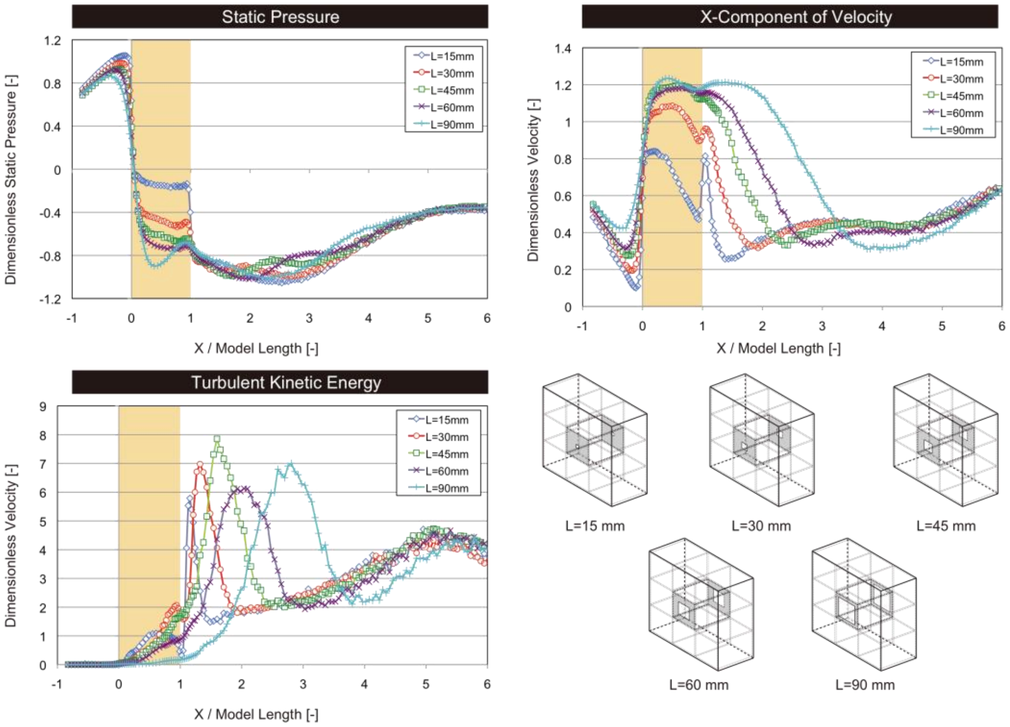


Fig.6 Flow quantities along the central line of the openings

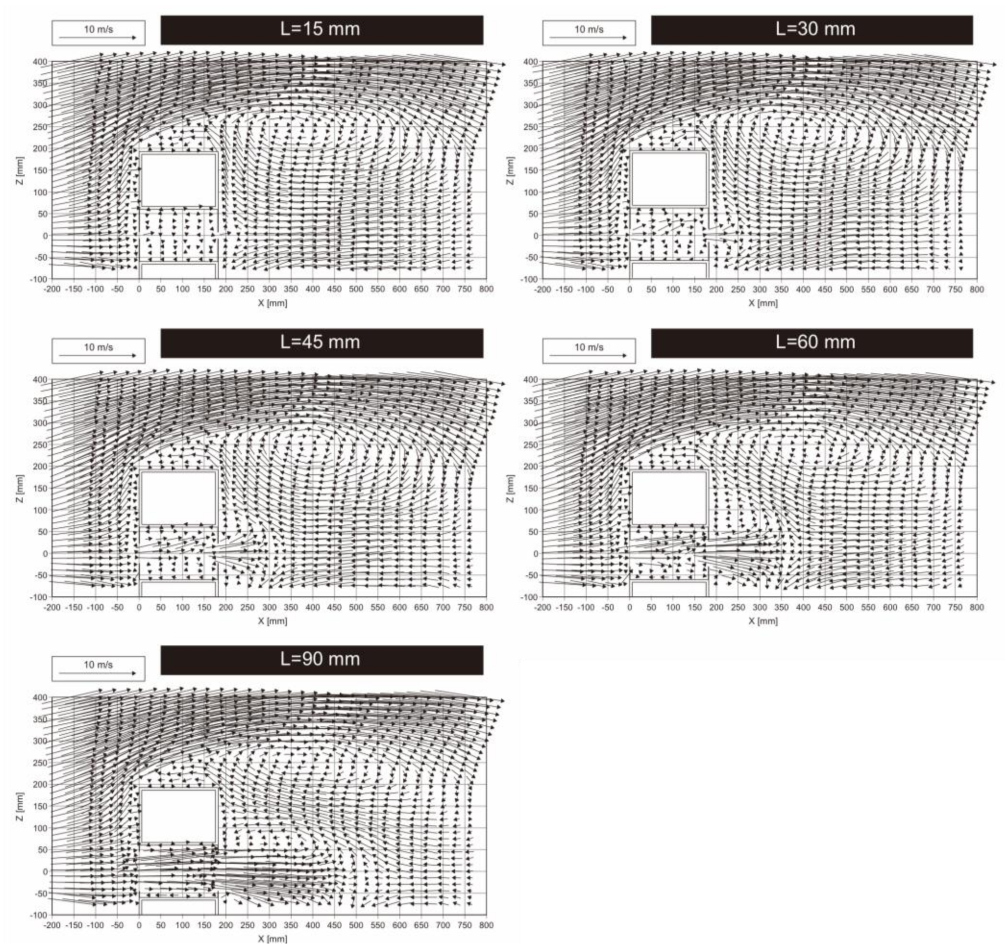


Fig.7. Velocity Vector Plots obtained from PIV