Experimental Investigation and Accuracy Study of CFD Analysis for Airflow around Cross-Ventilated Building

K. Asai [1], H. Kotani [2], T. Kobayashi [3], T. Yamanaka [2], K. Sagara [2] and Y. Momoi [2]

[1] Department of Architecture, The University of Tokyo, Japan
[2] Department of Architectural Engineering, Osaka University, Japan
[3] Department of Architecture and Urban Design, Ritsumeikan University, Japan

Abstract

In predicting flow rate of a building ventilated by wind, the orifice equation is usually used. This conventional method cannot work for the building provided with large openings. Therefore, the final goal of this study is to establish a new prediction method of the cross-ventilation rate, which is based on energy balance inside the stream tubes passing through/around a building. In determining stream tubes, it is beneficial to use CFD. In this study, the accuracy of CFD analysis is studied. Three turbulence models are used in the simulation; i.e. Standard k-ε Model Reynolds Stress Model, and Large Eddy Simulation. As a result, we can state the accuracy of Large Eddy Simulation is better than other CFD models.

Keywords: Cross-Ventilation, CFD analysis, Wind Tunnel Test

Introduction

In designing a building ventilated naturally by wind, the flow rate of a room under the wind must be known. The flow late of a cross- ventilated building is usually predicted by using so-called orifice equation based on Bernoulli's principle. In this method, the discharge coefficients obtained from the chamber method, and the wind pressure coefficients from a sealed building model are usually used. Ishihara (1969) showed this method could underestimate the flow rate of the building if the openings are large. Several studies on this problem have been conducted for a long time.

Murakami et al. (1991), Kato (2004) proposed “Power Balance Model”. In this method, flow rate is predicted based on energy balance inside stream tube. They proposed a model of the stream tube passing through/around a building and divided it into the control volumes. The final goal of this work is to establish a new prediction method based on “Power Balance Model”.

Kobayashi et al. (2008) have analyzed the stream tubes passing through/around the room using a rectangular model. To study more realistic condition, a multiple room building was used as a test object in this paper. In analyzing stream tubes, CFD seems to be effective. To analyze the stream tubes in the future work, the previous wind tunnel experiments are simulated by CFD analysis. By comparing measurement and CFD results using several turbulence models, the accuracy of the CFD simulation is to be studied. This paper focuses on the airflow around a cross-ventilated building, which includes such complicated flow field as separation and wake region.

CFD Analysis Method

Based on the previous wind tunnel test (Kotani et al. (2010)), CFD analyses were conducted. A test model shown in **Figure 1** was used for the analyses. The test model has a configuration of the building assumed to be composed of nine one-room residences of square array. The model is exposed to a free wind of 10 m/s. Five cases were simulated varying the parameters of the side length of the openings (L). The condition of the opening size was L=15, 30, 45, 60, and 90 mm. (For all cases, model has the side walls whose thickness is 6.0 mm, and 0.8 mm thick end walls were provided for the central residence to obtain sharped edge opening.) **Figure 2** gives all the cases, which are the same as those of wind tunnel test. In this study, two RANS models (Reynolds Averaged Navier-Stokes Equation) and LES (Large Eddy Simulation) were used. The summary of CFD analysis is shown in the following section.

Summary of CFD analysis

Standard k-ε Model (SKE) and Reynolds Stress Model (RSM) were used as RANS turbulence models. In the calculation of RSM, results of SKE were used for the initial condition. Assuming vertical and horizontal symmetry to reduce the calculation load, the computational domain was a quarter part of the working section of the wind tunnel. The mesh layout is shown in **Figure 3**. LES simulation using Smagorinsky-Lilly model was also conducted. The results of SKE were used for initial condition in LES simulation. The time step was set at 10^{-5} seconds. The calculated results during 0.8 seconds (8,000 time steps) from the beginning have been deserted regarding this term as the period of transition. As main calculation, unsteady flow was calculated for 15,000 time steps (1.5 seconds); i.e. 2.3 seconds in total. The calculation domain was equal to that of RANS model. Since it was unsteady analysis, however, the whole space of the wind tunnel was calculated. Summary of CFD analyses by RANS and LES are shown in **Table 1**.

Results and Discussion

Flow Field inside/around Model

From **Figure 4** to **Figure 6** shows the velocity distributions on the central cross section (Y=0) obtained from PIV measurement and CFD analysis. Here, only the cases of L=15, 45, and 90 mm are shown. As for LES, the time-average velocity was calculated based on the computational results for 1.5 seconds of main calculation.

In the result of PIV, the velocity vectors inside the model could not be estimated accurately. This is because some part of the laser sheet was reflected by the side wall of the model, which was made of acrylic board. For all the cases, the measurement shows a large circulating flow around the separation region and a wake generated on the leeward side. This tendency has been well simulated in the results of LES simulation. As for the SKE, in the cases of smaller openings, the tendency is simulated generally better, but in larger opening case, is not simulated well. In addition, SKE tends to underestimate the size of the separation region. RSM could not predict the circulating flow in the separation region. The measurement shows that the airflow traveling in positive direction and negative direction collide with each other on the leeward side. RSM makes a prediction that the positions of collision is shifted more downstream, which means RSM has a tendency to overestimate the size of the wake region. In LES results, a circulating flow in the separation region is well simulated. The flow circulation on the leeward side of the model could also be well simulated. Although the negative flow on the leeward side was not simulated only in the case of L=90 mm, the results in LES seems to be much better than other CFD results using RANS model.

Velocity Profiles around Model

Figure 7 shows the profiles of velocity magnitude around the side wall of the model obtained from PIV measurement and CFD analysis. Since the profiles are almost the same, in all cases, only the case of L=45 mm is shown.

In the line of X=0, 30 mm, PIV measurement shows the moderate velocity gradient if compared with the results of CFD. In this study, Recursive Cross-correlation Method was used in PIV algorithm. Using this method might underestimate velocity gradient because information in the target interrogation window was influenced by the surrounding interrogation. The result of SKE shows poor agreement in all turbulence models and tends to underestimate the size of the wake region. On the other hand, RSM and LES show relatively good agreement with the experiment.

Velocity and Pressure along a Center Line

Figure 8 shows static pressure, X-component of velocity, and turbulent kinetic energy along a central line through the model obtained from the measurement by I-type hot wire anemometer and CFD analysis. The horizontal axis shows dimensionless X-coordinate divided by the model length. In the result of velocity, the vertical axis is normalized by a reference velocity of 10 m/s, which is the velocity far upstream of the model. In the results of X-component of velocity and turbulent kinetic energy, the LES results simulating the measurement by I-type hot wire anemometer (described as “LES-simulated I-type hot wire”) were shown together. This is because I-type hot wire measures resultant value of X-component and Y-component of velocity as an absolute value.

X-component of instantaneous velocity was assuming as follows;

$$v_{Hotwire} = \sqrt{u^2 + v^2} \qquad (1)$$

To estimate turbulent kinetic energy, each component of fluctuating velocity was assumed to be the same as;

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

Assuming that I-type hot wire was measuring two component of velocity, turbulent kinetic energy was estimated as;

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

Static Pressure

The vertical axis indicates the dimensionless static pressure. Measured static pressure is divided by the reference dynamic pressure measured simultaneously at the point of pitot tube in order to ignore the effect of temperature fluctuation. Then, the dimensionless pressure was normalized by divided by the dimensionless total pressure at the most windward point.

For all cases, all turbulence models simulate the static pressure on the windward side of the model accurately. SKE has a tendency to overestimate static pressure particularly inside the model and on the leeward side. RSM overestimates the static pressure on the leeward side compared with the result of the measurement. LES result agrees with the measurement better than the results of other two models.

X-component of Velocity

X-component of velocity on the windward side was well simulated by RANS and LES. On the leeward side, the results of CFD analysis were quite different from the experimental results. This is because I-type hot wire cannot evaluate the negative velocity. The result of SKE disagrees with experimental result in the model. As for RSM, the larger openings are, the more out of agreement from the experimental result is. It is believed that RSM tends to overestimate the size of the wake region. The LES-simulated I-type hot wire predicts that the position of velocity weakening exists more windward in the case of L=90 mm, but it could be said that velocity field was well simulated. Therefore, it would appear that the result of LES simulation is relatively precise.

Turbulent Kinetic Energy

In the experimental result, turbulent kinetic energy was produced on the leeward side of the model. On the other hand, SKE shows the maximum turbulent kinetic energy at the windward opening. It is generally known SKE overestimates turbulence kinetic energy at the collision point, the region of great velocity gradient (see Murakami et al. (1990)). The result of the LES-simulated I-type hot wire was much similar in the tendency to the experimental result, though underestimation of the turbulence kinetic energy is seen in the case of larger openings. Actually, it is expected turbulent kinetic energy is higher on the leeward side.

Conclusions

In this paper, the previous wind tunnel test using PIV measurement was simulated by CFD analysis. By comparing measurement and CFD results using several turbulence models, the accuracy of the CFD simulation was studied. The conclusions are listed as follows:

(1) Standard k-ε Model (SKE) could not simulate the velocity and pressure field well.
(2) In the results of Reynolds Stress Model (RSM), velocity profile around the model was well simulated. However, RSM has a tendency to overestimate the size of the wake region.
(3) LES showed the best agreement with the experimental results.

In the future work, the stream tube passing through/around the building should be analyzed based on LES simulation shown as highly accurate model.

Acknowledgments

This research was partially supported by the Ministry of Education, Science, Sports and Culture, 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)

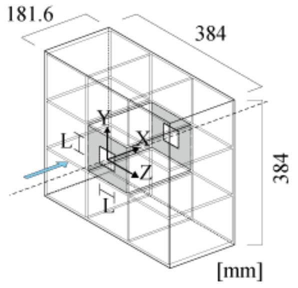
2. Murakami S., Kato S., Akabayashi S., Mizutani K., and Kim Y. D., “ Wind Tunnel Test on Velocity Pressure Field of Cross-Ventilation with Open Windows”, ASHRAE Transactions, Vol.97, Part 1, pp.525-538, 1991.

3. 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.

4. 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.

5. Kotani H., Kobayashi T., Asai K., Tanaka Y., and Shiozaki Y., “Velocity Measurement Inside and Outside a Cross-Ventilated Building by Means of PIV”, *Submitted to the 31st AIVC Conference*, Seoul, Korea, 2010

6. Murakami S., Mochida A., and Hayashi Y., “Examining the k-ε Model by Means of a Wind Tunnel Test and Large-Eddy Simulation of the Turbulence Structure around a Cube”, *Journal of Wind Engineering and Industrial Aerodynamics*, Vol.35, pp.87-100, 1990.



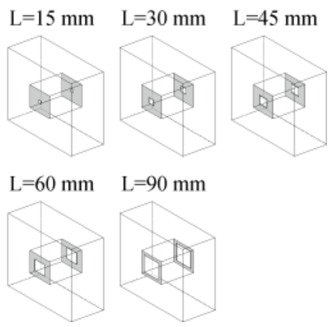


Figure 1 Test Model **Figure 2** Analysis cases

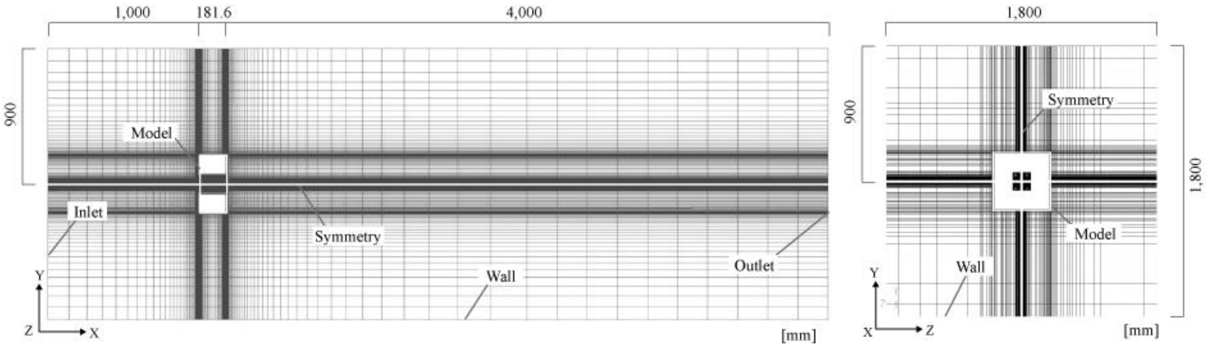


Figure 3 Mesh Layout

Table 1 Summary of CFD Analysis

RANS (Reynolds Averaged Navier-Stokes Equation)		
CFD Code		Fluent 6.3
Turbulence Model		Standard k-ε Model (SKE) Reynolds Stress Model (RSM)
Algolithm		Implicit Method (SIMPLE)
Discretization Scheme for Advection Term		QUICK
Boundary Condition	Inlet	Volocity : 10m/s Turbulent Intensity : 1% Turbulent Length Scale : 126mm
	Outlet	Gauge Pressure : 0 [Pa]
	Walls	Walls : Generalized Log Low Symmetry : free slip
Total Number of Cells		L=15mm : 1,612,128
		L=30mm : 1,612,648
		L=45mm : 1,685,808
		L=60mm : 1,685,808
		L=90mm : 1,619,688

LES (Large Eddy Simulation)		
CFD Code		Fluent 6.3
Turbulence Model		Large Eddy Simulation (Smagorinsky-Lilly Model)
Algolithm		Implicit Method (SIMPLE)
Discretization Scheme for Advection Term		Central Difference
Time Step		0.0001s (10kHz)
Transition Term		8,000 time step (0.8sec.)
Boundary Condition	Inlet	Velocity : 10m/s (Constant)
	Outlet	Gauge Pressure : 0 [Pa]
	Walls	Warner & Wengle linear power law
Smagorinsky Coefficient		0.1
Total Number of Cells		L=15mm : 1,893,060
		L=30mm : 2,050,240
		L=45mm : 2,024,400
		L=60mm : 2,026,160
		L=90mm : 2,028,240

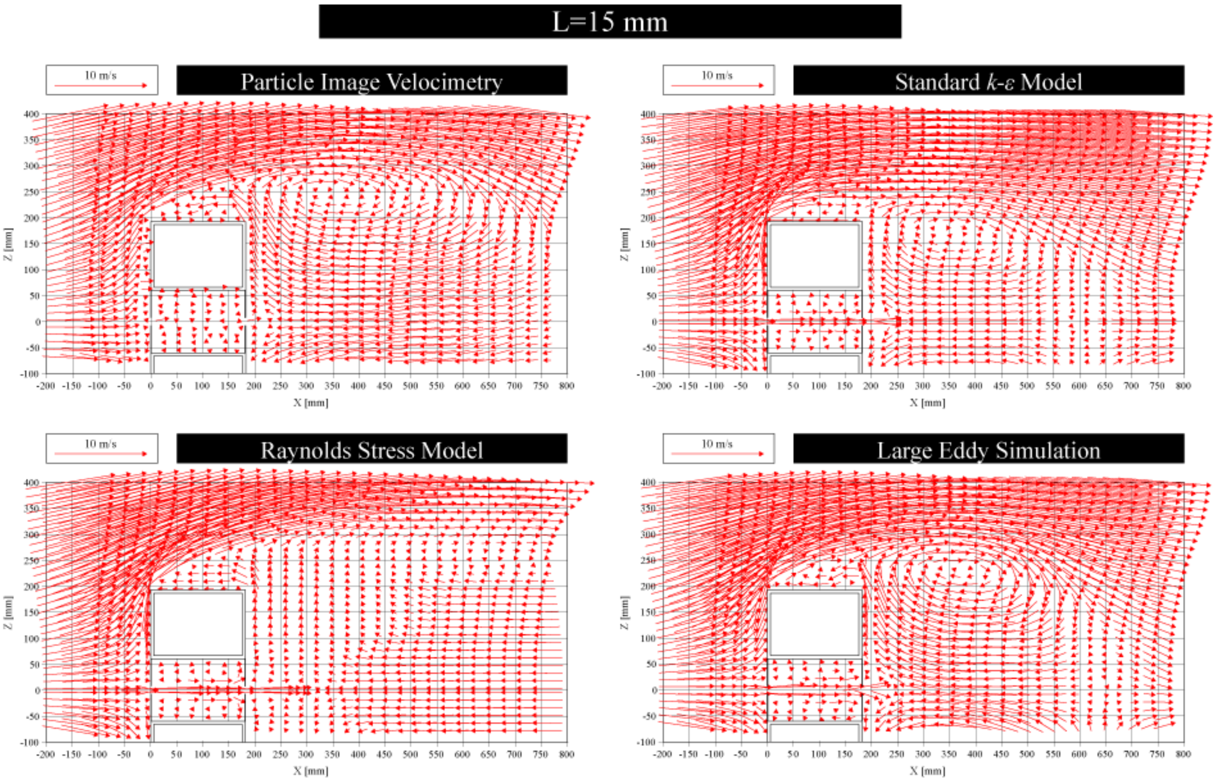


Figure 4 Velocity Vector for L=15mm

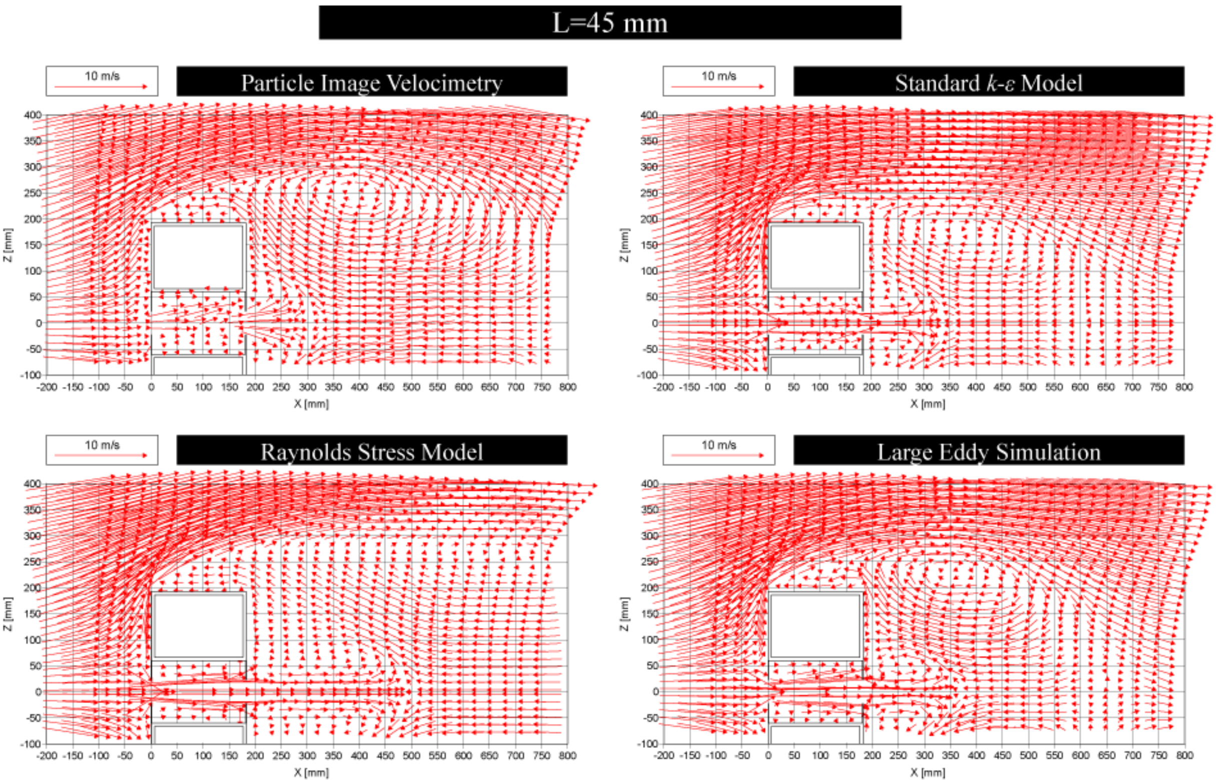


Figure 5 Velocity Vector for L=45mm

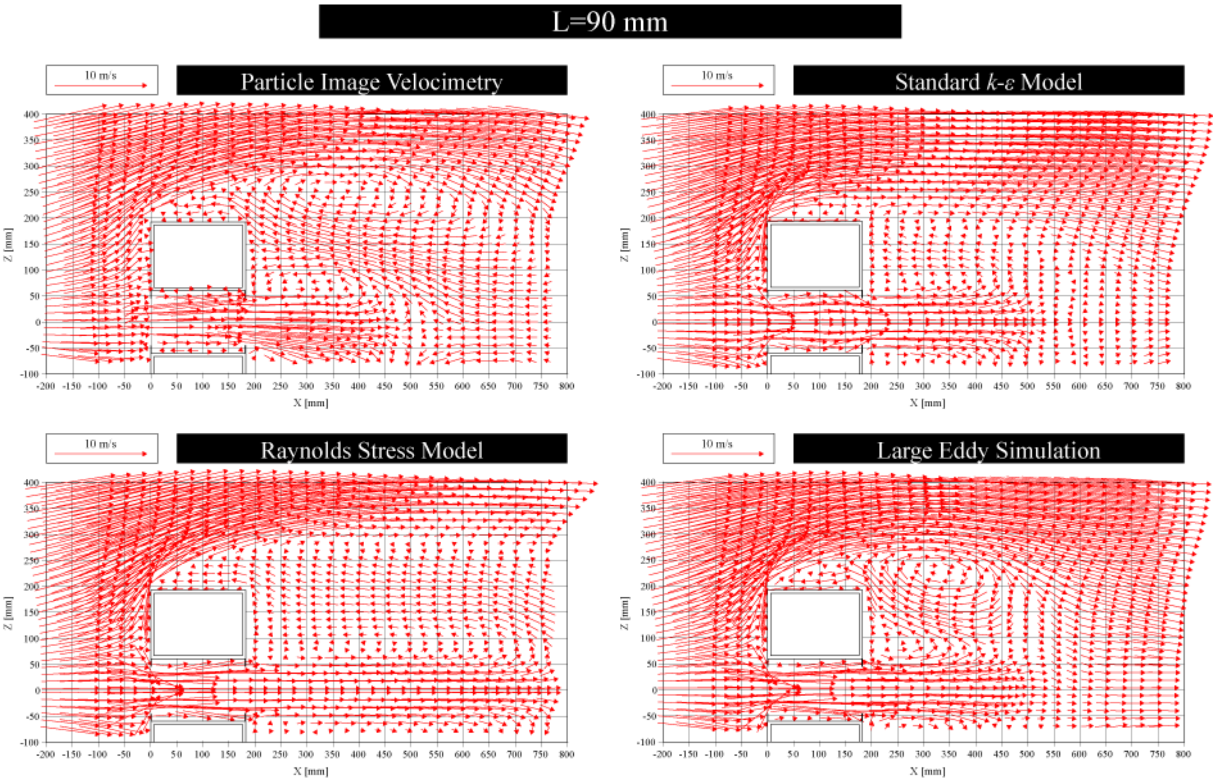


Figure 6 Velocity Vector for L=90mm

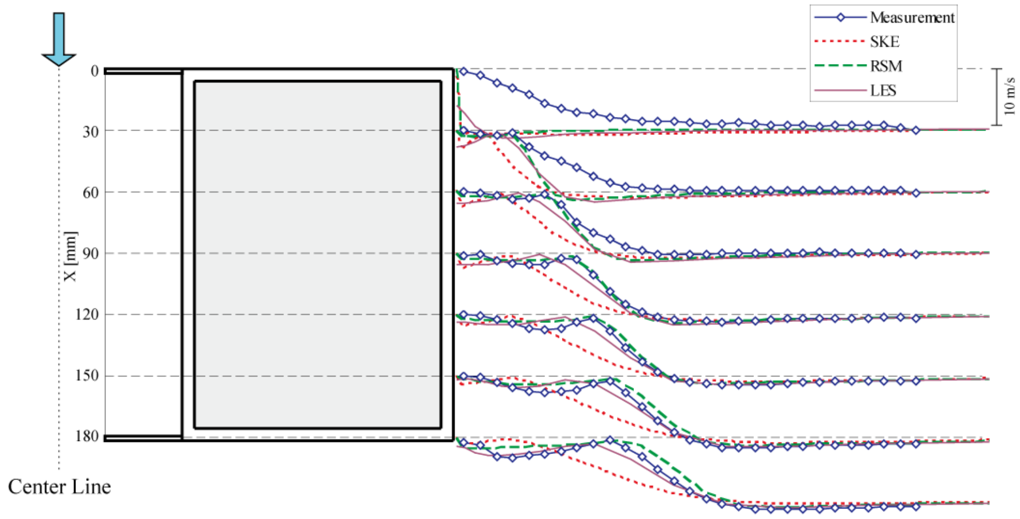


Figure 7 Profile of Velocity Magnitude

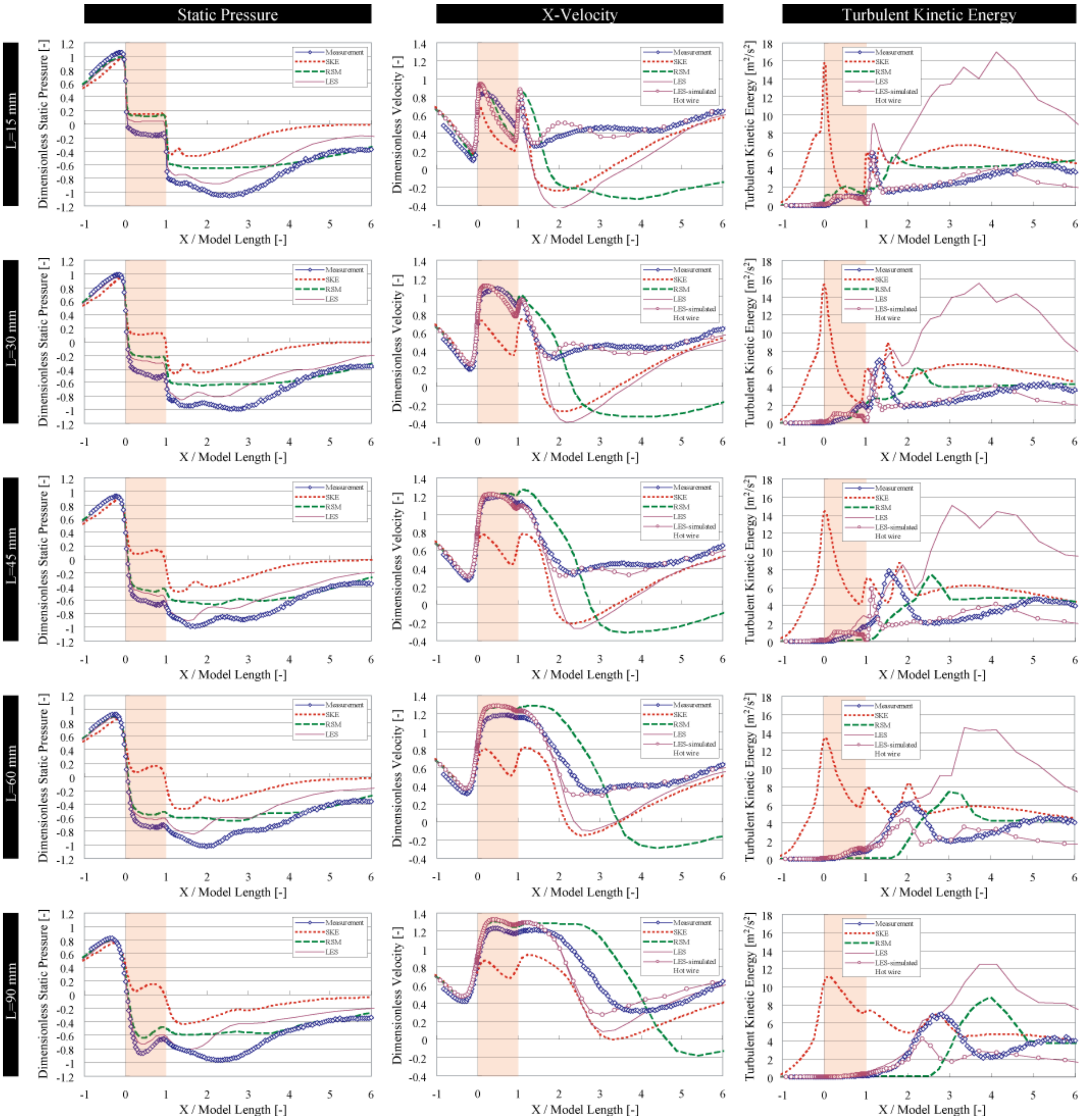


Figure 8 Static Pressure, X-component of Velocity, and Turbulent Kinetic Energy along a Central Line through the Model