A FUNDAMENTAL STUDY ON CROSS-VENTILATION RATE AND AIRFLOW CHARACTERISTIC INSIDE FLOW PATH THROUGH A ROOM
通風時の換気率及び室内を通過する気流性状に関する基礎的研究

Tomohiro KOBAYASHI*1, Hisashi KOTANI*2, Toshio YAMANAKA*3, Kazunobu SAGARA*4, Yoshina MOMOI*5 and Kaori ASAI*6

小林 知広, 甲谷 壽史, 山中 俊夫, 相良 和伸, 桃井 良尚, 浅井 香里

The conventional method to predict flow rate of a room cannot work well for cross-ventilation through large openings, and predicted flow rate becomes smaller than actual one. As a final goal of the work, the authors aim to establish an improved prediction method based on mechanical energy balance inside stream tubes passing around/through a building, i.e., power balance model. Since the cross-ventilation is quite complicated flow phenomenon, a detail of the flow characteristics in flow paths inside/outside the building has not been sufficiently clarified. This paper first presents discrepancy in flow rate between conventional prediction method and actual one by means of wind tunnel experiment. Afterwards, variation in flow quantities inside flow path through a cross-ventilated room is shown in accordance with opening size to understand the phenomenon. Finally, to validate CFD analysis which is essential to analyse the stream tube in the future work, its accuracy is to be shown by comparing experimental and numerical results for the flow path passing through a simplified building model.

Keywords: Cross-Ventilation, Cross-Ventilation Rate, Wind Tunnel Test, CFD, Wind Pressure, Stream Tube
通風、通風量、風洞実験、CFD、風圧、流管

1. Introduction

Cross-ventilation is regarded as a beneficial method to obtain thermal comfort in a hot summertime. To save non-renewable energy sources, such control method of building utilizing renewable energy is important. The cooling energy required for mechanical devices is not significant in residential buildings, whereas the cross-ventilation design is still important from the viewpoint of both thermal comfort and energy performance. In designing a well-functioning cross-ventilation building, various factors in flow characteristics must be taken into account in advance in the designing phase, e.g., flow rate, velocity magnitude, flow path. Since the cross-ventilation is a complicated flow problem, however, these cannot be easily predicted. Even the flow rate that is one of the most fundamental information, there exists no appropriate prediction method of convenience for the time being when the openings are large. This means that the conventional method using wind pressure coefficients and summed-up resistance coefficient of openings cannot work well for large openings. This is mainly because the stream tube from the inlet opening reaches leeward opening before sufficient expansion 4), i.e., overall pressure loss becomes smaller than predicted one.

Ishihara*7 showed that the flow rate is underestimated in cross-ventilation due to “interference” of openings and introduced correction factor called interference coefficient which modifies overall resistance coefficient through the flow path. Kurabuchi and Ohba et al. 5) dealt with the case where wind direction is not perpendicular to the openings because the resistance of an opening is affected by inflow direction, and postulated that the discharge coefficient be determined based on local similarity at the opening. Kotani and Yamanaka*8 showed the iterative calculation method of flow rate considering the effects of both interference and inflow direction. These are interpreted as corrective method of the discharge coefficient or effective opening area, which are supposed to use wind pressure difference obtained from a sealed building model as driving pressure. Considering cross-ventilation phenomena, however, to use the pressure from a sealed building model is also questionable because relatively large dynamic pressure could remain in the flow path in reality.

Sandberg*7 interpreted the prediction of cross-ventilation rate as a catchment problem of the flow and examined variation in flow rate by introducing geometrical parameter named catchment area. Murakami, Kato,

*1 Lecturer, Department of Architecture and Urban Design, College of Science and Engineering, Ritsumeikan University, Dr. Eng.
*2 Assoc. Prof., Division of Global Architecture, Graduate School of Engineering, Osaka University, Dr. Eng.
*3 Prof., Division of Global Architecture, Graduate School of Engineering, Osaka University, Dr. Eng.
*4 Prof., Division of Global Architecture, Graduate School of Engineering, Osaka University, Dr. Eng.
*5 Assistant Prof., Division of Global Architecture, Graduate School of Engineering, Osaka University, Dr. Eng.
*6 Graduate Student, Department of Architecture, Graduate School of Engineering, the University of Tokyo

立命館大学理工学部建築都市デザイン学科
講師・博士（工学）
大阪大学大学院工学研究科地球環境工学専攻
准教授・博士（工学）
大阪大学大学院工学研究科地球環境工学専攻
教授・博士（工学）
大阪大学大学院工学研究科地球環境工学専攻
教授・工博
大阪大学大学院工学研究科地球環境工学専攻
助教・博士（工学）
東京大学大学院工学系研究科建築学専攻
修士課程
Akabayashi et al.\(^{6,7}\) showed an alternative prediction concept considering energy balance inside stream tube (Power Balance Model), which was originally proposed for the flow in the pipe junctions by Guiffl and Fraser.\(^{8}\) Axley and Chung\(^{9}\) formulated airflow network model based on almost the same theory. These prediction methods based on actual phenomena regarding stream tubes passing through/around a building seem to be rational and available for various flow condition in treating cross-ventilation problem. However, the details of the stream tube characteristics have not been sufficiently clarified and consequently, there exists no method to estimate energy loss inside the stream tube at the moment. The objective of this work is to estimate and evaluate the energy loss and energy transportation inside the stream tube quantitatively. This is to be achieved by using CFD, which must be validated based on wind tunnel measurement in advance. This paper focuses on the stream tube especially passing through a room and aims to clarify qualitative characteristics of the cross-ventilated flow field based on wind tunnel tests, and to verify accuracy of the CFD analyses simulating those experiments.

2. Models and Studied Cases

A closed-circuit wind tunnel in Osaka University was used for the measurement, which has a working section of 11.0 m length, 1.8 m width and 1.8 m height. A basic configuration of the model is a building assumed to be composed of nine one-room residences in square array as shown in Fig. 1. Only the central room is provided with open windows on opposite sides. The end walls of the central room, shown as a shaded area in Fig. 1, are made of 0.8 mm thick brass plate to obtain sharp edge opening and others were made of acrylic board of 6.0 mm. As for the studied cases, the side length of the openings (L) was then varied as 15, 30, 45, 60, and 90 mm. To investigate the airflow where two openings interfere each other, the test model was exposed to the approaching flow with its openings perpendicular to the wind direction. To understand fundamental characteristics of the flow, in addition, the test model was located at the centre of the wind tunnel cross-section. This model was exposed to a free flow of 10 m/s to keep Reynolds number sufficiently high at the opening.

3. Wind Pressure Coefficient on the End Walls

To confirm accuracy of the model installation especially regarding wind direction, pressure distribution on the end wall was measured. Here, brass plates were replaced by those provided with pressure taps shown in Fig. 2, and vertical/horizontal pressure distributions were measured. The pressure was measured for 30 seconds with a sampling frequency of 100 Hz by using a manometer (Validyne, MP-45). Obtained pressure is shown in Fig. 3, where pressure is shown as dimensionless value (Cp value) by dividing it by dynamic pressure measured by a pitot tube located 300 mm away from the wall of the wind tunnel and 600 mm upstream from the windward wall. From this result, horizontal and vertical symmetry can be seen and both of them are almost the same. In addition, since the opening exists and stagnation line where stream tube is divided is around the opening, it can be seen that the pressure becomes lower around the edge of the opening.

As basic and specific information of this model, the wind pressure (Cp value) was also measured without any opening on envelop, which would be interpreted as driving pressure for the conventional prediction method when the opening is provided there. For this measurement, the brass plate was replaced by a plate without opening, which is provided with 121 pressure taps instead as shown Fig. 4. Pressure was measured in the same way as that shown above. Obtained pressure distributions for the windward and leeward sides are shown in Fig. 5. On the windward wall, pressure distributes in relatively narrow range. Nevertheless, central region shows high pressure, which indicates the flow stagnates here. On the leeward wall, meanwhile, Cp value is almost between -0.67 and -0.68, which means pressure distribution is almost uniform. The area-weighted average value of these pressures over the area where an opening is expected to be is to be regarded as driving force of the ventilation in the conventional method to predict flow rate, which is discussed in the following section.

4. Flow Rate and Its Underestimation by Conventional Method

To examine fundamental problem of the conventional flow rate prediction method quantitatively, the flow rate was measured by using a hot wire anemometer (KANOMAX JAPAN, 0251R-T5 type φ 0.5 μm tungsten wire). By integrating X-component of velocity measured on the leeward opening,

---

![Fig.1 Studied Model (Dimensions in mm)](image1)

![Fig.2 Measurement point for Wind Pressure Coefficient](image2)

![Fig.3 Wind Pressure Coefficient on the End Walls Provided with Openings](image3)

![Fig.4 Measurement point for Wind Pressure Coefficient](image4)

![Fig.5 Distribution of Wind Pressure Coefficient on the End Walls](image5)
flow rate was estimated. The velocity was measured at the centre of each virtual square over the leeward opening shown in Fig. 6. Measurement was conducted for 30 seconds with 1.0 kHz frequency, and the same procedure was repeated five times on each measurement point to decrease uncertainty of measurement. Table 1 summarize the obtained flow rate in addition to that estimated by the conventional method given by following orifice equation;

\[ Q_{\text{Conventional}} = \frac{1}{\sqrt{\phi}} \cdot \sqrt{C_{P_W} - C_{P_L} \cdot V} \]  

(1)

where, wind pressure coefficient is referred to the previous section, and the resistance coefficient of the opening (\( \zeta \)) is obtained from the chamber method in the preceding work by Furukawa et al.\(^{1,11}\) where almost no difference could be seen among five cases of opening size and determined to be 2.6. Fig. 7 compares the dimensionless flow rates between conventional method and the hot wire measurement, which is divided by opening area and approaching velocity. When the openings are large, in general, discrepancy in flow rate becomes large and predicted rate is approximately 70 % of actual one. Here, it must be noted that the flow rate obtained from hot wire might be overestimated in the cases of \( L=15 \) and \( 30 \) mm due to the paucity of measurement point, where velocity distribution cannot be sufficiently taken into account.

5. Measurement of Flow Quantities along Central Line through Model

5.1 Methodology

The variation of both spatial static pressure and velocity inside the stream tube is fundamental and beneficial information, whereas it has almost never been shown except the authors’ work for simpler model of only one room\(^{12,13}\). With two aims of understanding fundamental characteristics of the stream tube passing through a room and obtaining correct value of the CFD analyzes, static pressure and velocity are measured along a central line through the model where measurement seems to be relatively easy by assuming symmetry vertically and horizontally. The static pressure was measured by hand-made pressure tube provided with four static pressure holes, and velocity was measured by I-type hot wire used in the flow rate measurement. Both probes are shown in Fig. 8. Since the static pressure tube is original, its sensitivity to the wind direction must be known in advance. Fig. 9 shows the variation in pressure reading obtained for every 2 degrees in wind direction. Here reference pressure is that of 0 degree and each pressure is divided by the reference dynamic pressure of approaching flow. Since the dynamic pressure affects the pressure when the flow is separated from the tube, tube pressure reading becomes smaller as wind direction becomes large. Between the range of \( \pm 6 \) degrees, as the extent of this error, discrepancy from static pressure of 0 degree is less than 1.0 % of dynamic pressure of that location. Therefore, this effect must be considered where the flow is highly turbulent. For this flow field along the central line, the flow seems to be highly turbulent and comes from several direction in the wake region generated on the leeward side. In this region, however, velocity is relatively small and so is dynamic pressure consequently according to square of velocity magnitude, and it is believed that this effect to underestimate static pressure is not significant for this measurement.

Fig. 10 depicts the experimental set-up. Test model was located at the centre of the wind tunnel and exposed to a free flow of 10 m/s. The pressure tube or hot wire was attached to the arm provided for the traverser, which could be moved with an accuracy of 0.05 mm. Pressure or velocity was measured every 10 mm between \( X=-150 \) to 1,000 mm in the vicinity of the opening (\( X=-50 \) to 50 mm and \( X=130 \) to 230 mm), the distance of measurement points was amended to be 5.0 mm because abrupt variation seemed to occur. The reference pressure inside the wind tunnel was that measured by a pitot tube. Pressure measurement was conducted for 30 seconds with 100 Hz, and velocity measurement, 30 seconds with 1.0 kHz to estimate the turbulent kinetic energy based on measured velocity fluctuation.

![Fig.6 Measurement Points for X-component of Velocity in Estimating Flow Rate](image)

![Fig.7 Comparison of Flow Rate between Hotwire Measurement and Conventional Prediction Method](image)

![Fig.8 Probe Head of Static Pressure Tube and Hot wire](image)

![Fig.9 Rotation Characteristics of the Static Pressure Tube](image)

![Fig.10 Schematic of Experimental Set-up of Wind Tunnel Test for Flow Quantity along a Central Line (Dimensions in mm)](image)
5.2 Experimental Results and Discussion

Fig. 11 shows the experimental results. Static pressure is normalized by total pressure at the most upstream measurement point. Velocity is also expressed as dimensionless value by dividing it by the velocity of the approaching flow which was measured by the pitot tube simultaneously. Based on velocity fluctuation, turbulent kinetic energy was also estimated \(^{19}\).

The static pressure generally increases in front of the model according to Bernoulli’s principle, where velocity decreases 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 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. The velocity in front of the model measured by hot wire corresponds to this tendency.

In flowing into the room, static pressure decreases and the large opening case tends to show lower pressure. Considering this pressure drop includes both energy loss and conversion into the dynamic pressure, large openings exhibiting large pressure drop generates larger velocity because energy loss seems to become small along the central line close to the opening. Measured dimensionless velocity exceeds 1.0 except the case where L=15 mm. The case of L=90 mm shows pressure recovery inside the room.

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 \(^{11}\) 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 back flow exists on the leeward side. Here, it must be noted that I-type hot wire anemometer cannot output negative velocity but absolute value. Although the existence of the back flow is to be examined in the following work by using PIV, it is believed that the static pressure on the leeward side depends almost only upon whether or not the wake is blown away.

To analyse turbulent statistics is also important for understanding phenomenon and for verifying 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 a sharp peak relatively close to the model after discharged from an opening, the location is shifted in leeward direction as openings become large. The rationale of this production seems to be that the discharged flow collides to the back flow in the wake here. 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 model. From this result, 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 back flow exists on the leeward side. In such a flow field, static pressure on the leeward side is almost independent of opening size and might be replaced by wind pressure obtained from a sealed building model.

6. Accuracy Study of CFD Analysis
6.1 Methodology

To analyse the stream tube and its power transportation, CFD analysis is beneficial because it can determine streamlines easily if compared with experiment. To explore a simulation method having sufficient accuracy, above shown wind tunnel test was simulated by using three kinds of turbulence models; i.e., Standard k-\(\varepsilon\) model (SKE), Reynolds Stress Model (RSM)\(^{12}\), and Large Eddy Simulation (LES). A commercial code Fluent 6.3 was used for the analysis. Summaries of calculations are shown in Table 2 and Table 3. As for RANS calculation, vertical and horizontal symmetry planes were applied as free slip boundaries to decrease computational load. The inlet boundary condition of turbulent statistics for RANS calculations was based on turbulent intensity of the approaching flow (\(I=1.0\%\)) and hydraulic diameter of the working section (\(D=1,800\ mm\)) and given as;

\[
k = \frac{3}{2} \left(\frac{V_{\text{approach}}}{D}\right)^{1.75}
\]

\[
\frac{\overline{u'\overline{v}'}}{k} = \frac{2}{3} k \quad \frac{\overline{u'\overline{v}'}}{u''} = 0 \quad (\text{for Reynolds Stress Model})
\]

\[
\varepsilon = C_\varepsilon \frac{u''}{\Lambda} \quad (4)
\]

where, \(C_\varepsilon\) is a closure coefficient of 0.09 and \(\Lambda\) is the turbulence length scale given as 0.07\(D\). As for the LES calculation, no symmetry assumption was applied because instantaneous flow field was obtained. A constant velocity of 10 m/s was given as the inlet boundary condition because the approaching flow was not highly turbulent and its effect seemed not to be significant. LES calculation was started with the result of SKE as the initial condition. The time interval of the LES calculation was 10\(^{-3}\) second and calculated for

![Fig.11 Experimental Results of Flow Quantities along the Central Line through the Test Model](image-url)
23,000 time steps in total. Fig. 12 shows the fluctuation of static pressure and X-component of velocity at the inlet/outlet opening and on the leeward side of the model along the central line. During approximately 0.4 seconds in the beginning, both fluctuations of pressure and velocity are obviously different from later ones. Here, the results during 0.8 seconds in the beginning were deserted regarding this as transition term from the result of SKE and average value over the rest 1.5 seconds were used for the analysis. The computational domain and the mesh layout are shown in Fig. 13. Relatively long leeward region (length 4,000 mm) was created to enable the comparison of the stream tube in the future work. This also aims to give uniform pressure as outlet boundary condition.

6.2 Results and Discussion

As basic information, the flow rate was calculated by integrating X-component of velocity over the inlet/outlet opening and averaging them for each case. The flow rate obtained by CFD is compared with that of hot wire measurement and conventional prediction method in Fig. 14. Generally, the conventional prediction method underestimates flow rate and tends to be relatively constant dimensionless flow rate. In other four method, flow rate becomes larger as the openings becomes large. The result from SKE is smaller than other two turbulence models and also hot wire measurement. RSM and LES show almost the same flow rate, and these results well agree with those of hot wire measurement when the openings are large. For small openings, especially the case of L=15 mm, both CFD results differ from experimental result. However, this could be due to that the experimental result is overestimated because of lack of the measurement point and the velocity distribution was not sufficiently taken into account. As for the flow rate in the case of small opening case, therefore, it seems difficult to evaluate the accuracy of CFD from this result.

Fig. 15 compares static pressure, X-component of velocity, and turbulent kinetic energy between CFD and measurement. Static pressure and velocity are shown as dimensionless value as well as Fig. 11. The reference pressure of CFD results took the same location as experiment after calculation.

As for the static pressure, all turbulence models show good agreement with experimental result on the windward side of the model. Inside the room, however, SKE shows obvious overestimation of the pressure, especially in the case of large openings. RSM and LES resulted in relatively similar pressure to the measurement. On the leeward side, higher pressure is seen in SKE result. RSM seems to improve this tendency, but distribution is different from experiment. LES generally gives a little higher pressure than experiment, but the distribution is quite similar. In addition, considering that measured pressure might be smaller than actual one on the leeward side because of the effect of dynamic pressure as shown in sensitivity of the static pressure tube, accuracy of the LES calculation seems to be sufficient regarding static pressure along the central line.

As for X-component of velocity, CFD results can evaluate average negative value, while experimental result based on I-type hot wire can output only absolute value. Therefore, numerical and experimental results can be naturally different on the leeward side where the back flow exists. To eliminate this, based on LES, instantaneous velocity assumed to be comparable with hot wire measurement is here defined as;

\[ v_{LES-instant} = \sqrt{v_x^2 + v_z^2} \]  

This assumes that Z-component is ignored from velocity reading by I-type hot wire. The time-average value of this velocity is plotted in Fig. 15, and expressed as the case “LES-Hotwire”. Accordingly, turbulent kinetic energy based on the fluctuation of this instantaneous velocity was also calculated.

On the windward side of the model, X-component of velocity is generally well simulated by CFD. Inside the model, velocity is apparently

---

---
underestimated by SKE, but RSM and LES well agree with experimental result. On the leeward side, experimental result shows positive value and it is regarded as resultant of X and Y component of velocity. In the result of RSM, the locations where velocity starts to decrease and increase afterwards are shifted downstream if compared with experimental result. This indicates that RSM overestimates the size of the wake region on the leeward side. As discussed by Okka et al., this may be due to that the modelling method of Reynolds stress and closure coefficient has been supposed to be applied to relatively simple shear flow and resulted in an inappropriate model for such complicated flow field like a wake. The velocity obtained from LES seems to be similar to experiment where back flow does not exist, and the result of LES-hotwire shows good agreement with experiment though small difference can be seen in the case of large openings.

In the result of turbulent kinetic energy obtained from SKE, excessive production is seen on the windward side and quite different from experiment. This is typical tendency of standard k-ε model based on eddy viscosity model, e.g., see Murakami et al. The profile of the turbulent kinetic energy of RSM is similar to experimental result but is shifted downstream as discussed above for velocity. Although result from LES based on strict definition of turbulent kinetic energy much differs from experiment, LES-hotwire that is arranged to be comparable with measurement could simulate the experimental result well. Therefore, it can be said the numerical result based on LES yielded the most accurate result in three turbulent models studied here at least along the central line through the model. In analysing power transportation within whole flow path, accuracy of this calculation must be verified also around the model in the future work.
7. Conclusions

The authors aim to analyse the power transportation inside the stream tube by using CFD for prediction of cross-ventilation rate of the building. For this objective, this paper presented wind tunnel experiment to verify the discrepancy in flow rate between conventional prediction method and reality. Also by measuring flow quantities and conducting CFD analyses using three turbulence models simulating experiment, the fundamental characteristics of cross-ventilation flow field and accuracy of CFD simulations were studied. The conclusions are summarised as follows.

1) For large openings, the flow rate predicted by the conventional method becomes smaller than reality, and the extent of this underestimation was approximately 70% for the largest opening case studied in this paper.

2) The size of the wake generated on the leeward side including back flow is determined mostly by building shape and independent of opening size.

3) When there exists the wake including back flow, pressure on the leeward side becomes almost the same when opening size is different despite that the velocity retained by discharged flow externally differs, i.e., it is whether or not the wake exists that determines leeward static pressure.

4) The peaks of turbulent kinetic energy can be good indicators of the collision point of discharged and back flow, and the confluence point of external flow passing around the building.

5) SKE could not precisely simulate static pressure, velocity, and turbulent property for the cross-ventilated flow field. RSM showed somewhat improved accuracy if compared with SKE, but tended to overestimate the size of the wake region. LES could predict measured flow quantities best of all turbulence models studied including turbulent statistics. Although small difference could be seen in the case of the largest opening, it was regarded as sufficiently accurate result in analysing the stream tube.

This paper focused on the accuracy of CFD for only the stream tube passing through a room. Since it is also aimed to analyse the energy loss due to the convergence and divergence of the stream tube passing around the building, accuracy of CFD analyses “around” the model must also be verified in the following work, and power transportation is finally to be analysed based on the results of CFD using Large Eddy Simulation.

Acknowledgement

A part of this work was supported by Grant-in-Aid for JSPS Fellows (Representative Tomohiro Kobayashi (20-912)), Japan Society for the Promotion of Science, 2009.

Note

*1) In estimating turbulent kinetic energy based on experimental result using I-type hot wire anemometer, following assumption of isotropic turbulence was applied.

\[ k = \frac{1}{2} \left( \sigma_u^2 + \sigma_v^2 + \sigma_w^2 \right) = \frac{1}{2} \sigma_{ww}^2 \]

Since the hot wire is assumed to measure resultant of two components, turbulent kinetic energy was estimated based on fluctuating velocity as;

\[ \sigma_{ww}^2 = \sigma_u^2 + \sigma_v^2 + 2\sigma_w^2 \]

*2) The Reynolds Stress Model used here basically follows LRR Model \(^{10}\). However, the modelling method of the turbulent diffusion term in the transport equation of Reynolds stress is based on isotropic assumption.

Nomenclature

- \( A \) Area [m²]
- \( C_p \) Wind pressure coefficient [-]
- \( C_m \) Model coefficient [-]
- \( I \) Turbulence intensity [-]
- \( k \) Turbulent kinetic energy [m²/s²]
- \( Q \) Flow rate of the room [m³/s]
- \( V \) Mean velocity [m/s]
- \( \rho \) Reynolds stress [m²/s²]
- \( \nu \) Instantaneous velocity [m/s]
- \( \nu' \) Fluctuating velocity [m/s]
- \( \varepsilon \) Turbulent dissipation rate [m²/s]
- \( \Lambda \) Turbulence length scale [m]
- \( \rho \) Air density [kg/m³]
- \( \zeta \) Resistance coefficient of an opening [-]

References


14) R. Ooka, S. Murakami, A. Mochida : Study on Modeling for \( q_r \), \( \Phi_0 \) and Turbulent Diffusion of \( \langle u'(r) \rangle \), Journal of Architecture, Planning and Environmental Engineering, Transaction of AIJ, No.504, pp.55-61, 1998.2. (In Japanese)


1. はじめに
通風時に換気を予測する際、開口を有しない建物模型から得られる風圧係数とチェンバー法により得られる開口単一の抵抗係数を用いる従来の風呂換気量予測手法では、換気量が過小に評価されると思われており、石原らは通風口間の「干渉」によるものと説明し、換気係数の抵抗係数として干渉係数を定義した。その後、通風時の換気量予測値に関しては種々の手法⑴⑵が提案されているが、大開口を通過する通風時に適用が可能で実用的な予測手法は確立されていない。村上、加藤、赤林ら⑶⑺は通風時の平均値による流速のモデルを示し、流速内のエネルギー損失とエネルギー輸送の半波面式に基づく実現象に即した通風換気予測手法の考え方を示した。同様に、近年ではAxleyら⑾が妥当な換気計算理論を結約しているが、開口内で生じるエネルギー損失等のデータはあくまで、通風時の静圧や風速の詳細な分布も実測的に十分に明らかにされていないのが現状と言える。本論文は、流速内のエネルギー損失等のデータ整備のため、風洞実験により風速及び静圧分布の詳細を明らかにした上で、流速の解析に有効と考えられるCFD解析を行い、静圧・風速・乱流統計に関する解析精度を検証するものである。

2. 実験概要及び実験条件
筆者らはこれまでに単室の簡易な形状の模型を用いて通風時の気流解析を行ったが、より実現的な形状に関し検討を行うため、図1に示す通り9つの単室で構成される建物を想定した模型を用い、中央の一室のみに開口を設けて実験している状況とした。開口サイズはL=15, 30, 45, 60, 90 mmの5条件を設定し、開口の大きさによる流速特性の違いを明らかにする。通風時の基本的特性を明らかにするため、模型の開口を風向に対して正対させ、風洞の中央に設置した。

3. 風上及び風下側風圧係数
主に風向に関する模型の設置精度を確認するため、L=45 mm条件に関して、風上・風下側面の壁面静圧分布を測定した。図3に示す通り、鉛直方向、水平方向に対称性が見出された。従来法により換気量を推定する際の換気動的な開口を有しない場合の風上/風下側面静圧（風圧係数）も模型内の基本情報として測定した（図5）。この結果、風上側は模型中央付近より大きく低い値が存在すること、風下側は概ね一様に風圧が分布していることが確認された。

4. 通風量の測定及び従来の予測手法による風量の過小評価
本章では従来法により予測された通風量と実際の風量の差異を検証する。従来法による風量は、前章で得られた風圧係数分布を風洞入口で面積積分し平均値した値であり、既往の研究⑾より開口単一の抵抗係数から算出した。また、実際の風量は図6に示す測定で熱線風速計により測定した風速により算定した。風量の比較結果（表1及び図7）から、開口条件では従来法の風量が実際の風量の70%程度であること、また、開口部風速の誤差が大きくなることがわかった。また、開口が小さい場合には風速測定点数が少ないため、開口の縁付近で低風速となる風速分布が十分に考慮されないと考えられ、実際の風量の差異はさらに小さいと考えられる。

5. 中心線上の流体量測定
5.1 実験手法
本章では室全体を通過する流量の状況を把握することを目的として、開口部中心線上における静圧と流速の測定を行った。なお、測定は自作の静圧管及び1型熱線風速計（図8）を用いた。静圧管の指向特性は図9に示す通り、風がブローパーに平行でない場合は速度圧の影響により圧力の読み値が小さくなるため、中心線上であっても気流が大きく乱れる後流域では静圧を低く評価する可能性がある。しかし、wakeの中心付近や、静圧測定点自体が小さくなると考えられ、速度圧や流速の2乗に比例して小さくなるため、本実験の測定点では静圧管の指向特性が大きな影響を及ぼすことはないと考えられる。

5.2 結果と考察
図11に示す測定結果から、風上側では気流が減速するため静圧が上昇し、室へ流入する際は嘆流により静圧が低下するというベルヌーイの定理に従った傾向が得られている。後流域では、図12両サイドが変化した際に風速の低下する箇所は異なるが、静圧の分布に大きな差異は見られない。これはどの開口条件でも逆流を含むwakeが生じており、その領域の大きさは開口サイドとよくなる基本形状上により決まるためと考えられる。乱流エネルギーは模型出口直後に急激なピークが見られるが、ここの点で流速変動とwake内の逆流が衝突して乱れが大きくなったと推察される。その後乱流エネルギーは緩やかに上昇し再度極大値を示すが、このように再度乱れが大きくなる可能性として、剥離した気流が合流する点であることが考えられる。

6. CFD解析の精度検証
6.1 解析手法
後に流速を同定し、エネルギーの損失及び輸送の解析を行うことを目的として、前章の風洞実験をCFD解析により再現した。解析は標準k-εモデル（SKE）、応力方程式モデル（RSM）、LESの3種の乱流モデルを用いて行った（表2、表3）。

6.2 結果と考察
図14に風量の比較結果を示す。SKEの結果は風量を大きく過小評価しているが、RSMとLESでは概ね実験値と一致している。また、開口が小さい場合では実験値はCFDと比較して大きな値を示しているが、これは測定点が少ないため、速度分布を十分に考慮していないことが原因と考えられる。図15に実線の結果を示す。SKEでは室への流入以外、静圧の傾斜が実験値と大きく異なる。RSMでは結果は改善されている傾向は観察されるが、後流域のサイズを過大評価している。LESの結果は実験結果と最も良く一致しており、風速と乱流エネルギーに関しては1型熱線風速計が風速の2成分を測定していることを考慮して再現した結果（LES-Hotwire）が実験と良く一致しているため、最も良い結果が得られたと言え、流速の解析を行うための十分な精度を有していると判断した。

7. おわりに
本論文では、風洞実験により従来の風呂換気量予測手法の誤差を定量的評価することとともに、通風時に室内側を通過する流速の基本的特性を明らかにした。また、LESを用いたCFD解析が十分な精度を有することを確認した。今後は本解析結果を用いて室外側流速の解析精度を検証し、流速エネルギーの解析を行う所存である。

（2010年8月10日原稿受理、2011年4月5日採用決定）