NUMERICAL CALCULATION OF ROOM AIR MOVEMENT
—ISOTHERMAL TURBULENT TWO-DIMENSIONAL CASE—
(PART II)

By Takao TSUCHIYA*, Member of A.I.J.

PREFACE

In the preceding paper (PART I), the precise procedure for the numerical calculation of room air movement was presented. Especially three parameters $r$, $C_i$, and $C_z$ were proposed in order to make the numerical calculation practicable.

In this paper, those parameters are determined by comparing the computed velocity distributions with experiments in a square and a rectangular cross sectional rooms with a low side wall outlet and a high side wall inlet. It is confirmed that these parameters are valid even for a complicated case in which an air curtain is affected by a side blow by comparing the computed flow patterns and velocity distributions with experiments. In an experimental setup, not only the measurements of air velocity but also the flow visualization are attempted.

3. Cases and Conditions of Calculation

Cases and conditions of calculation are summarized in Table 1. Precise descriptions for each case would be found in PART I.

4. Experiment

Measurements of air velocity and flow visualizations were carried out to verify the calculations for each case.

<table>
<thead>
<tr>
<th>Case NO.</th>
<th>Arrangement of Outlet and Inlet</th>
<th>Combination of $r$, $C_2$ and $t$</th>
<th>Outlet Air Velocity</th>
<th>Smallest Mesh Spacing</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td></td>
<td>$r$</td>
<td>$C_2$</td>
<td>$t$</td>
</tr>
<tr>
<td>2</td>
<td></td>
<td>$r$</td>
<td>$C_2$</td>
<td>$t$</td>
</tr>
<tr>
<td>3</td>
<td></td>
<td>$r$</td>
<td>$C_2$</td>
<td>$t$</td>
</tr>
<tr>
<td>4</td>
<td></td>
<td>$r$</td>
<td>$C_2$</td>
<td>$t$</td>
</tr>
</tbody>
</table>

PREFACE

In the preceding paper (PART I), the precise procedure for the numerical calculation of room air movement was presented. Especially three parameters $r$, $C_i$, and $C_z$ were proposed in order to make the numerical calculation practicable.

In this paper, those parameters are determined by comparing the computed velocity distributions with experiments in a square and a rectangular cross sectional rooms with a low side wall outlet and a high side wall inlet. It is confirmed that these parameters are valid even for a complicated case in which an air curtain is affected by a side blow by comparing the computed flow patterns and velocity distributions with experiments. In an experimental setup, not only the measurements of air velocity but also the flow visualization are attempted.

3. Cases and Conditions of Calculation

Cases and conditions of calculation are summarized in Table 1. Precise descriptions for each case would be found in PART I.

4. Experiment

Measurements of air velocity and flow visualizations were carried out to verify the calculations for each case.
4-1 Test Rooms

Two basic test rooms were used. One was a square cross section, the other was a rectangular cross section, respectively. The former dimensions were 1.5×1.5×1.5 m, the latter were 3.3×1.5×1.5 m. The front wall and a part of the side wall were made of acrylic plate so that the observation of air flow was possible. Other walls were composed of veneer plate. The slot type with 1.5×0.05 m outlet and inlet were located at low and high sidewall. Especially, in a rectangular test room, there had another pair of outlet and inlet in the central area of ceiling and floor. These were used for an air curtain.

4-2 Assembly of Air Supply and Return System

The schematic view of supply and return system was shown in Fig. 7. The supplied air from the slot traveled through a test room, return air chamber, a packaged conditioner and a supply air chamber.

4-3 Method of Air Velocity Measurements

The air velocity at each measuring station was measured by moving the hot wire anemometers along a vertical and a horizontal guide pipes. For the measurement of air velocity higher than 0.2 m/s, Anemomaster Model 24-3111 of KANOMAX (Japan) was used and for lower than 0.2 m/s, Type 5580/81 Low Velocity Anemometer of DISA (Denmark) was used.

4-4 Flow Visualization

The arrangement of an experimental apparatus for the flow visualization is shown in Fig. 8. It was composed of the 1/5 scale model of the test room, 0.6 m×0.3 m in plan 0.3 m high, two couples of outlet and inlet air chambers, the illumination box and the tracer generator boxes. They were connected by flexible ducts with two blowers.

The scale model was made of 12 mm veneer plate except the front wall and the part of the floor where 3 mm glass was used.

Particles of metaldehyde\(^{23}\) were used as visible tracers. The substance sublimes at about 100~120°C and then reverts to the solid phase to form light weight particles. These particles were continuously generated by heating the powder state metaldehyde feded from outside.

---

![Diagram](image-url)

**Case 1**

**Fig. 7** Schematic View of Supply and Return Air Systems.
Fig. 8 Arrangement of Apparatus for Visualization of Two-Dimensional Room Air Movements.

The particles were illuminated by four pieces of a high beam lamp (500 W) set in an array mounted in the illumination box. The light was projected through the glass part with 1 cm width of the floor to take the pictures of two-dimensional air movement in the central plane. The time-exposure photographs were taken with different kinds of exposure time.

5. Results
5-1 Value of \( \tau \)

The computations with different value of \( \tau \) were carried out and then the following result was deduced. The value of \( \tau \) strongly affected the computational stability, that is, when \( \tau \) was 0.1 or 0.4, the computation was quite instable and the steady state solution was not obtained. On the contrary, when \( \tau \) was 1.0 or 4.0 the steady state was attained in rather earlier stage of time steps.

The large discrepancy between two calculated velocity distributions in Fig. 10 might be mainly dependent on the difference of \( \tau \) value. In case of \( \tau = 4 \), the diffusion term took a considerable role and the excessive damping was encountered. Therefore, \( \tau = 1 \) appears to be the most proper value.

5-2 Mixing Length

We assumed two types of distribution for mixing length \( l \). Type I was based on the assumption that the boundary layer was extended to the center of a room from the wall, and Type II was on the assumption that the influence of a wall was limited to the area within the point whose velocity took the maximum value and the flow was free turbulent within the area from this point to the center of a room.

In Fig. 10, the broken line represents the velocity distribution when Type I is used, whereas the solid line does when Type II is used. The former indicates a rather moderate curve at the region quite near a wall, on the contrary the latter indicates a steep variation. It is observed that the latter is more close to the experimental velocity distributions. In latter case, we assumed also non-isotropic model of turbulence. As it should be acceptable that the diffusion in the flow direction is quite small as compared with the advection in a turbulent boundary layer, the velocity distribution might be little affected by the assumption of isotropic or non-isotropic. Therefore, it seems to be dependent on the difference of \( l \) distribution that the solid line reveals a steeper curve than the broken line.

5-3 Value of \( C_s \)

It can be seen from Fig. 9 that the velocity distribution near a wall is strongly affected by the value of \( C_s \). In case of \( C_s = 1.0 \), the variation of velocity over a wall indicates a moderate curve and
the maximum velocity appears at the point quite far from a wall. On the other hand, in case of \( C_2 = 0.1 \), the maximum velocity appears at the point close to a wall and the overall velocity distribution agrees very well with the experimental result within a small deviation. It can be concluded from the sections 5-1 5-3 that the best fitted values of \( \gamma \) and \( C_2 \) are 1.0 and 0.1, respectively, for the numerical calculation of two dimensional room air motion with box model type finite differencing scheme, providing that the coefficient \( C_3 \) can be estimated by the knowledge of smooth plate flow. In the later section, the comparison will be made between the experiments and the calculations using the values \( \gamma = 1 \), \( C_2 = 0.1 \).

5-4 Flow Pattern

The calculated stream patterns for Case 1 and 2 are illustrated in Fig. 9 (b) and Fig. 10 (b). They are corresponding to the solid line in Fig. 9 (c) and Fig. 10 (c). The agreement between the calculated and the photos taken by flow visualization technique is remarkable.

5-5 Application to Other Cases

The numerical calculations were applied to other two cases. One is the case for the room air motion with the jet injected downward at the center of the ceiling which is called as an air curtain. The other is for the air motion where the air curtain is affected by the creeping flow along the floor. In

Fig. 9 Comparison of Flow Pattern and Velocity Distributions between Computed and Experiment in Case 1.

Fig. 10 Comparison of Flow Pattern and Velocity Distributions between Computed and Experiment in Case 2.
both cases, the selected values of $r=1$ and $C_r=0.1$ were used. Comparisons between the calculations and experiments were indicated in Fig. 11 and 12. In both cases, the calculated velocities agree quite well with the experiments except the jet region. Furthermore, it can be observed that flow patterns also agree well.

6. Conclusions

The computer programming of the numerical calculations for two-dimensional room air movements was developed by means of a box model finite differencing scheme for advection terms and the leapfrog time marching scheme. Where three parameters were proposed. One was related to the eddy kinematic viscosity and the others were to correct the velocity and the vorticity gradients on a wall represented by a finite scheme. Two types of mixing length distribution were also examined in connection with isotropic or non-isotropic turbulence model. Several kinds of these parameters were attempted in order to find the best fit values as compared with experiments.

The following conclusions are presented:

1) The most appropriate value of $r$ is 1.0, which is an empirical constant when an eddy kinematic viscosity is expressed in the product of the cube of mixing length and the absolute value of vorticity gradient, $K_e = r^2 \left| \frac{\partial^2 u}{\partial y^2} \right|$. When $r$ was smaller than 1.0, the computed quantities indicated an irregular oscillation. That was believed to resolve a part of turbulence. On the contrary, for $r=1$, a relatively large damping took place and the unrealistic feature of flow pattern was disappeared.
(2) The Type II together with non-isotropic is recommended as a distribution of mixing length. That is derived on the assumption that a room air can be divided into two regions by the surface composed of a maximum velocity point, the wall boundary region and the free turbulent region.

(3) The suitable value is believed to be 0.1 or less as $C_z$ which corrects the discrepancy between an actual vorticity gradient on a wall and that represented by finite difference approximation, providing that $C_z$, which corrects a velocity gradient on a wall, can be estimated by the manner similar to the smooth plate flow.

(4) The validity of these parameters was confirmed by comparing the calculated velocity profiles with experiments in applications of the numerical procedure described in this report the room air movement with symmetrical air curtain and that affected by a side blow.

ACKNOWLEDGEMENTS

The author would like to acknowledge the assistance of Mr. H. Seto of the Environmental Engineering Section, Building Physics and Fire Division. His help in carrying out the experiments was greatly appreciated. Also, wishes are extended to all students of Kogakuen University who helped in both calculations and experiments.

REFERENCES

12) T. Tsuchiya : "Approximate Solution of Room Thermal Convection", Ibid, 4206
13) M. Enai and N. Aratani : "Fundamental Consideration for Analysis of Room Thermal Convection", Ibid, 4204
室内気流の実用的数値計算法（梗概）
（第2報）
正会員土屋亜雄

室内気流は一般的に乱流を含むものと考えられ、それをいかにして数値計算法に取り入れるかが問題である。近年乱流の解析法に関する研究の進展は著しく、その成果は各分野で応用されており、室内空気分布に関しても道路等により2方程式モデルによる乱流解析が試みられ良好な結果が報告された。いよいよ室内空気分布の数値計算も実用の段階にさしかかってきたといえよう。

ここに報告するのは、吹出・吸込を伴う二次元等温の室内気流を対象とした、比較的規模の狭小な計算機によっても可能な数値計算の提案である。その特徴は次の諸点であり、可能な限り単純明瞭な取扱いに徹することを第一義としたものである。

i）基本方程式
（1）計算を出来る限り簡単なうえのために、渦度をNavier-Stokes方程式を採用している。
（2）乱流解析には、最も簡単な直線的な代数モデルにとっている。
（3）基本方程式に渦度を用いているので、一般的に行われている渦度特性係数の速度勾配とその結びつけをとこなわず、渦度勾配と結びついている。

ii）数値計算法
（1）基本方程式を差分近似する場合、非線形項による“noodling”あるいは、“aliasing error”的生成増大に伴う計算不安定を避ける意味で、渦度の2乗の平均値が保存されることが望ましい。そのためのスキームはいくつか開発されているが、スキームの単純さおよび次に述べる不規則メッシュの採用にも保存性が保たれるという点からBoxModelを用いている。
（2）一般に室内気流では吹出口および壁面近傍で速度変化が急激であり、室中部の変化は緩和である。壁面近傍の急激な速度変化を計算で捉えるためには非常に細かいメッシュが必要であり、室全体をこのメッシュ分割をおこなうと計算機の記憶容量および解析時間の点で実用的であるとはいかえない。したがって、速度変化の大きい部分は細かく、また変化の緩和な部分は粗くした不規則メッシュ分割が合理的であると考え、BoxModelによる非線形項の差分スキームと組合せている。

（3）時間差分としてはLeapfrogスキーム（中央差分）を使用している。一般に、3つ以上の時間レベルを使ったスキームは必ず物理的モデルの他に計算的モデルが存在するが、Explicit差分スキームの中では、中央差分のみが条件つきでこれら2つのモードに一番近い、また実際の現場と違わない強い運動エネルギーの減衰を生じさせることがないので望ましいスキームと言える。もちろん中央差分にも計算モードによる解の不安定性(time splitting)が存在するが、その増大をおさえる方法が考案されている。

iii）壁面極近傍の係数による近似
さきにも述べたとおり、室内気流の特徴は壁面近傍で速度変化が急激に変化することである。通常その内壁面から数センチメートルのオーダーであり、そのなかにはするかに、数ミリメートルのオーダーの極度の層流境界層が壁面に接して存在しているものと考えられる。したがって、これらの速度変化をとらえるには、数ミリのオーダーのメッシュ間隔が必要となり、たとえば400規則メッシュを用いたとしても、時間スキームの計算安定条件の制約から計算時間が莫大なものとなり実用上不可能にいうをえない。そこで、ここでは緩和型方程式を基礎として使用している関係上壁面における渦度および壁面からの流速の拡散を検討するが必要があるが、その計算を行う場合に補正係数を導入し、実際の速度分布に近づくように配置している。

以上の内容に沿って計算を実行する場合、次の3つの係数を決定しなければならない。すなわち、

\[ \tau = \frac{r_0}{K} \]

の如く渦動粘性係数（K）を混合距離（L）の2乗と渦度勾配との積で表現する場合の係数（r）を、上記の上流を差分近似で求める場合の補正係数（C_r）および上流からの渦度の拡散を求める場合の壁面に沿った方向の渦度勾配の差分近似に対する補正係数（C_s）である。

本論文は、これら3つの係数のうち（C_s）に関して平板上の流れの急激な変化を得る為の仮定のもと、他の2つは平均的な断面形状の室および断面形状が2.2の数値解析の室における実測値との比較から決定し、それらを吹出し方針の異なる2つの室内気流に適用して各々の実験値を比較し、その妥当性を検討したものであり、第1報および第2報に分かれている。
第1報では、渦度基本方程式のBox Modelによる差分近似の展開および3つの係数$r, C_i, C_z$の提案を行った。

本報告（第2報）では、これらの係数を決定するための実験条件・実験方法およびそれに対応した計算条件等を明らかにし、正方形および矩形の2つの断面形状の室を対象として気流速度分布の理論値と実験値との比較により最もふさわしい係数値を求めていいる。

また、これらの係数を用いてエアカーテン使用時の室内気流およびそれに側圧が作用した場合の計算を行なく、気流速度および可視化による流れのパターンの測定結果との比較から係数値の有用性を確かめている。

最後に結論として、十分発達した乱流で且つ流れのパターンがある程度推定でき、混合距離の分布形の概要が予測できる場合には本計算法が適用でき、3つの係数$r, C_i, C_z$の最適値は、

\[ r = 1.0 \]
\[ C_i = (y_1/y_2)^{0.7} \]
\[ C_z = 0.1 \]

ここに、

\[ y_1: \] 壁面に接するBox中央の壁面からの距離
\[ y_2: \] 層流底層の厚さ

であることが導かれた。