Numerical Method for Simulating Thermal Convection in a Long Channel Standing Vertically

Tetuya Kawamura
Graduate school of humanities and sciences, Ochanomizu University

(Received October 8, 2008)
(Revised Novenber 19, 2008)

Abstract

A new numerical method that is suitable for the calculation of thermal convection with main flow in a very long region is proposed. The flow variables are expressed as sum of those of the main flow caused by the thermal convection and their variations. Using this method, two-dimensional flow in a long channel is simulated. It is found that the steady flow of upward direction with high temperature is maintained. In order to verify the results, the method based on the stream function and the vorticity is also examined.

1. Introduction

Thermal convection is one of the most interesting fluid motion occurring in our daily life. In this study, we focus on the flow induced by the thermal convection in a long channel standing vertically. This kind of flow is important both for the purpose of engineering and for prevention against disaster.

The example of the prevention of disaster is the analysis of fire in a tunnel. One of the most tragic disaster is that occurring inside the tunnel near the entrance caused by the fire of the cable car in Austria in November 2000. The main objective of the present study is to develop a numerical method suitable for simulating this kind of flow numerically.

In the simulation of the convective flow in a long region, difficulty due to the high aspect ratio (length/diameter) takes place. It is quite difficult to satisfy the conservation of mass especially in the case that the dominant flow exits. In a vertical channel, upward flow is induced due to heat sources so that the air is sank at the entrance (lower boundary) and flows out at the exit (upper boundary). However, if we use conventional methods based on the primitive variables such as MAC method [1], the flow tends to stop somewhere in the long channel due to the accumulation of numerical error. It is widely known that the method based on the stream-function and the vorticity ($\psi - \omega$ method) can satisfy the conservation of mass exactly [2]. However, this method is restricted to computation of two-dimensional and axis-symmetrical flow.

In our previous study [3], we have developed new numerical method that is suitable for the computation of incompressible flow where one dominant flow exists such as a channel or a pipe flow caused by the pressure gradient. In the method, the flow variables are expressed as sum of those of the main flow and their variations. The former is obtained by solving one-dimensional equation analytically. To apply this method to convective flow, it should be modified since it is difficult to define dominant flow in thermal convection. In this paper, we use the averaged velocity near the heat source as the velocity of main flow and examine the effectiveness of the method mentioned above.

2. Numerical Method

The computational region is a long channel standing vertically. The heat source locates on the both sides of the wall as shown in fig.1 (the figure is rotated 90 degree to save the space). The convective flow is induced by this heat source. From this geometrical configuration, the flow can be assumed nearly two-dimensional. Therefore, we suppose that the flow is two-dimensional as the first approximation in this study although it is not difficult to extend the method to three-dimensional computations.

The basic equation for incompressible two dimensional flow under the Boussinesq approximation is as follows:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \tag{1}$$

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = -\frac{\partial p}{\partial x} + \frac{1}{Re}\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) \tag{2}$$

$$\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} = -\frac{\partial p}{\partial y} + \frac{1}{Re}\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) + \frac{Gr}{Re^2}T \tag{3}$$

$$\frac{\partial T}{\partial t} + u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y} = \frac{1}{Re \cdot Pr}\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}\right) \tag{4}$$

where T is the temperature, x and y are horizontal and vertical direction, parameters Re, Gr and Pr(=0.71) are Reynolds, Grashof and Prandtl numbers respectively. If the computational region is very long in vertical direction, the vertical flow is dominant and is nearly one-dimensional. In other words, we can assume

$$v = v(y,t), \quad p = p(y,t), \quad u = 0, \quad \partial/\partial x = 0 \tag{5}$$

as the first approximation. We also neglect the buoyancy force. Substituting equation (5) into equations (1) and (3) with T=0, we obtain

$$\frac{\partial v}{\partial y} = 0 \tag{6}$$

$$\frac{\partial v}{\partial t} = -\frac{\partial p}{\partial y} \tag{7}$$

(Equation (2) becomes 0=0). From equation (6),

$$v = f(t). \tag{8}$$

In our previous study, $f(t)$ is determined from boundary condition at the entrance of the long region. In this case, we determine it from the averaged velocity in the region surrounded by two heat sources. If we substitute equation (8) into (7), we obtain

$$f'(t) = -\frac{\partial p}{\partial y} \quad i.e. \;\; p = -f'(t)y + C \quad (C : constant). \tag{9}$$

From equations (8) and (9), we can define the variation of the velocity and pressure as follows:

$$v = f(t) + \tilde{v}(x,y,t), \quad p(x,y,t) = -f'(t)y + C + \tilde{p}(x,y,t) \tag{10}$$

After the substitution of equation (10) into original equations (1)-(4), we obtain the basic equation in this study

$$\frac{\partial u}{\partial x} + \frac{\partial \tilde{v}}{\partial y} = 0 \tag{11}$$

$$\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+(f+\tilde{v})\frac{\partial u}{\partial y}=-\frac{\partial \tilde{p}}{\partial x}+\frac{1}{Re}\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right) \tag{12}$$

$$\frac{\partial \tilde{v}}{\partial t}+u\frac{\partial \tilde{v}}{\partial x}+(f+\tilde{v})\frac{\partial \tilde{v}}{\partial y}=-\frac{\partial \tilde{p}}{\partial y}+\frac{1}{Re}\left(\frac{\partial^2 \tilde{v}}{\partial x^2}+\frac{\partial^2 \tilde{v}}{\partial y^2}\right)+\frac{Gr}{Re^2}T \tag{13}$$

$$\frac{\partial T}{\partial t}+u\frac{\partial T}{\partial x}+(f+\tilde{v})\frac{\partial T}{\partial y}=\frac{1}{Re\cdot Pr}\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}\right). \tag{14}$$

These equations are nearly the same as the original equations (1)-(4) although unknowns are different. These equations can be solved by the standard method such as MAC method, SMAC method [4] and so on. Note that the boundary condition on the wall (no-slip) becomes

$$\tilde{v}_{wall}=-f(t) \tag{15}$$

from its definition.

3. Results

In order to examine the effectiveness of the numerical method mentioned above, we simulate the flow in the channel with two heat sources shown in fig. 1. The aspect ratio of the channel is 25. The number of the grid points is 250×20 in vertical and horizontal direction. Note that the direction of the gravity is right to left in figures 2,3,5,6,9,11,12,14 and they are also shortened 1/2.5 to save the space. The Reynolds number based on the width of the channel is 50 and the Grashoff number is 5000. The function $f(t)$ appearing in equation (8) is determined by averaging the velocity v over the region surrounded by two heat sources.

Figures 2 and 3 are the velocity vectors and isotherms at non-dimensional time of T=2. Figure 4 is the flow rate along the vertical wall. Figures 5,6 and 7 are the results when the flow reaches nearly steady state (T=30). These figures correspond to figures 2,3 and 4 respectively. Figure 8 is the time history of the flow rate. It is shown the the flow rate increase gradually and reaches about 0.7.

In order to verify the results, the same flow is simulated by ψ-ω method. Since the flow rate is unknown a priori, the value of stream function on the wall is determined as follows.

At first, the velocity is calculated from the stream function instantaneously. We compute the flow rate by the velocity in each cross section between two heat sources. The average value of flow rate mentioned above is set to the value of the stream function of one wall at that moment while that of another wall is set to 0.

Figure 9 is isotherms obtained by the $\psi - \omega$ method at T=30. The agreement to the figure 6 is well. Figure 10 is the time history of flow rate. Although it increases slowly compared with figure 8 and reaches slightly large value (=0.8), the tendency is nearly the same. These differences are due to the boundary conditions at the entrance (uniform flow is imposed at the entrance in this case).

Figure 11-16 shows the results obtained by the standard MAC method. Figures 11-13 are the results at T=2 and figures 14 and 15 are those at T=30. Unlike to the results obtained by the present method and $\psi - \omega$ method, the flow in the channel is very weak and local circulation near the heat source is observed. Moreover,the flow rate is not conserved well as shown in fig 13.

4. Concluding Remarks

It is known that the numerical method for incompressible flow based on solving the Poisson equation of pressure such as the MAC method has a weakness that it is difficult to satisfy the equation of continuity precisely. This shortcoming becomes more serious when the computational region is very long or it is required to obtain unsteady solutions.

Thermal convection in a long channel studied in this paper is just the case. In our previous study, we proposed the new method that is effective for the calculations of incompressible flow in a long region. The idea of the method is that the flow variables are expressed as sum of the main flow and their variations. Therefore, the main flow should be defined by the boundary conditions at the entrance. On the other hand, in the present case, the velocity of main flow is determined by the flow itself during the computation through the effect of heat source.

In this study, we define the velocity of the main flow by averaging the velocity over the region surrounded by two heat sources and apply our previous method. Using this method, two-dimensional flow in a long channel is simulated. It is found that the steady flow of upward direction with high temperature is maintained. This result agrees well with that obtained by the $\psi - \omega$ method. However, this result is not given by the standard MAC method.

References

[1] F.H. Harlow and J.E. Welch: Numerical Calculation of Time-Dependent Viscous Incompressible Flow of Fluid with Free Surface. Phys. Fluids, Vol.8 pp.2182-2189

[2] See textbook of CFD such as Computational Fluid Dynamics and Heat Transfer written by D.A. Anderson, J.C. Tannehill and R.H. Pletcher, Hemisphere Publishing, 1984

[3] Tetuya Kawamura and Kazuko Miyashita: A Numerical Method for Calculations of Flow in Long Region with Branch, Natural Science Report of the Ochanomizu University, Vol.58, No.1 pp.45-55, 2007

[4] A.A. Amsden and F.H. Harlow: The SMAC Method: A Numerical Technique for Calculating Incompressible Fluid Flows, LOS Alamos Scientific Laboratory Report LA-4370, 1970

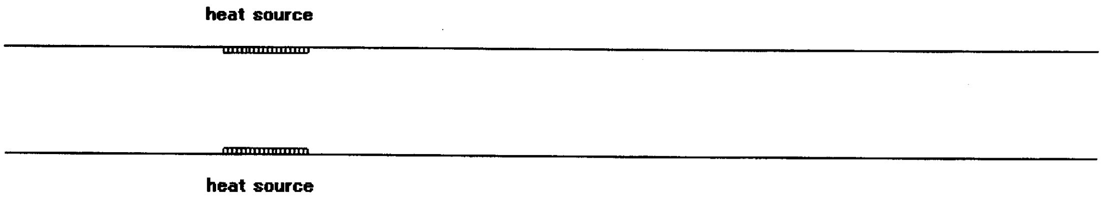


Fig.1 Computational domain and location of heat sources (All figures are Rotated 90 degree i.e. gravity acts left direction and are shortened 1/2.5 i.e. aspect ratio is 25)

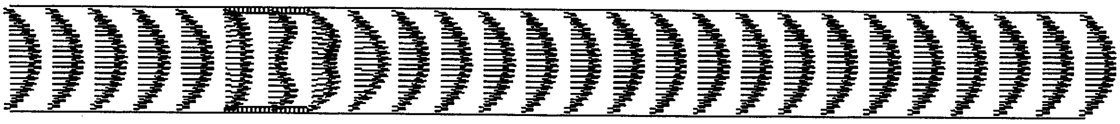


Fig.2 Velocity vector at T (non-dimensional time) =2

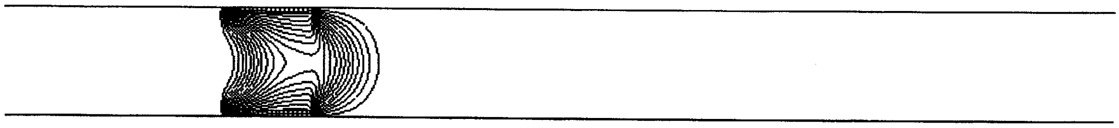


Fig.3 Isotherms at T=2

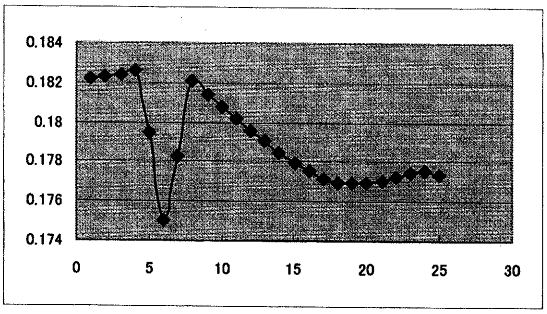


Fig.4 Flow rate along the wall at T=2

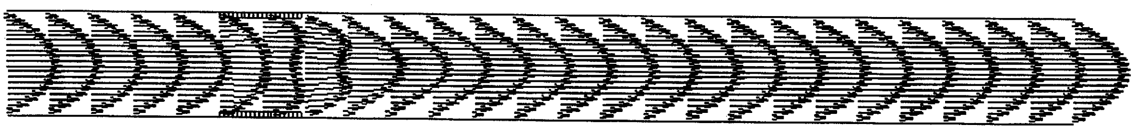


Fig.5 Velocity vectors at T=30 (nearly steady state)

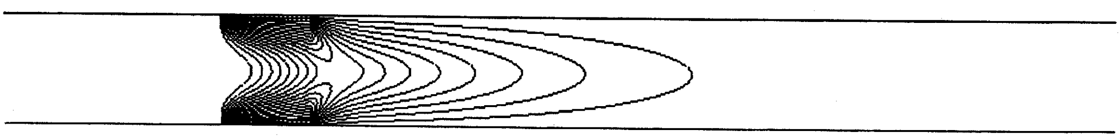


Fig.6 Isotherms at T=30

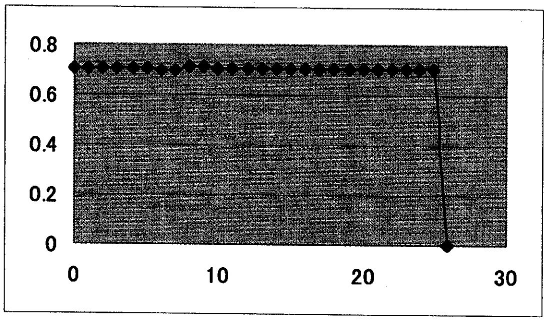


Fig.7 Flow rate along the wall at T=30

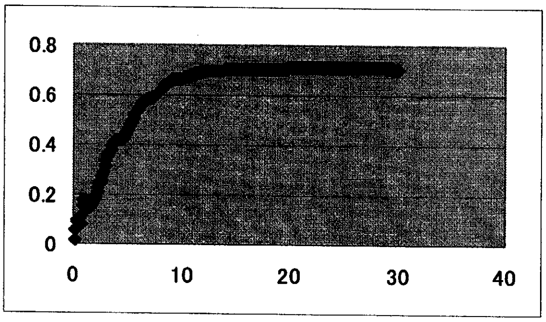


Fig.8 Time history of flow rate

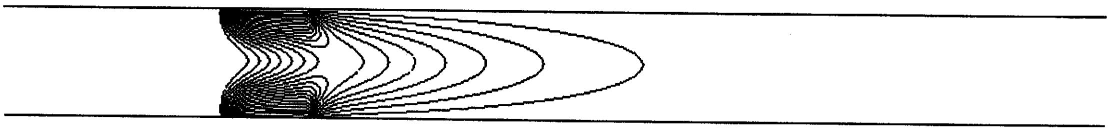


Fig.9 Isotherms obtained by stream function and vorticity method (T=30)

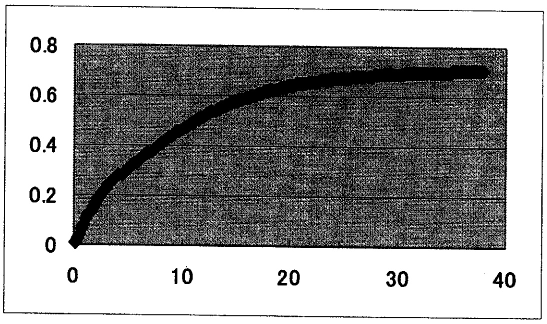


Fig.10 Time history of flow rate obtained by stream function and vorticity method

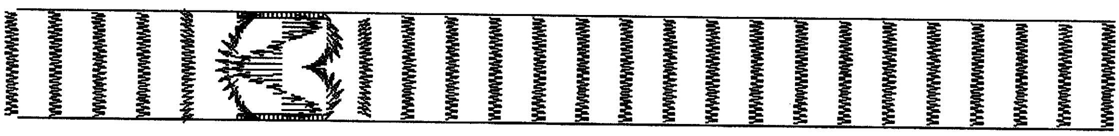


Fig.11 Velocity vectors at T=2 obtained by standard MAC method

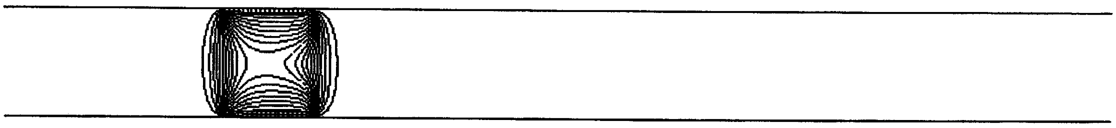


Fig.12 Isotherms at T=2 obtained by the standard MAC method

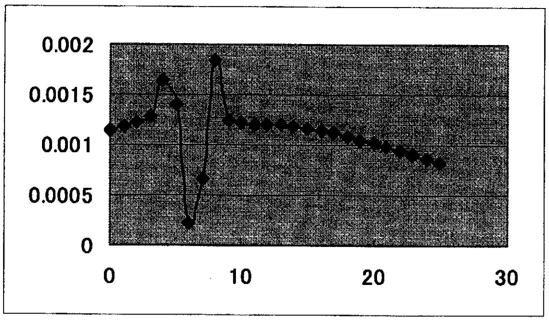


Fig.13 Flow rate along the wall at T=2 obtained by standard MAC method

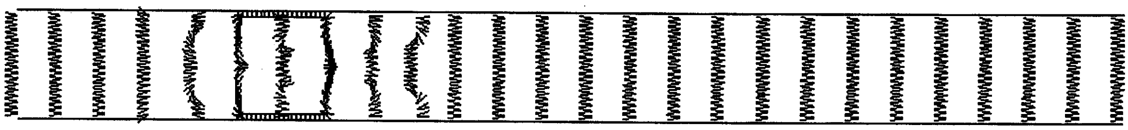


Fig.14 Velocity vectors at T=2 obtained by the standard MAC method

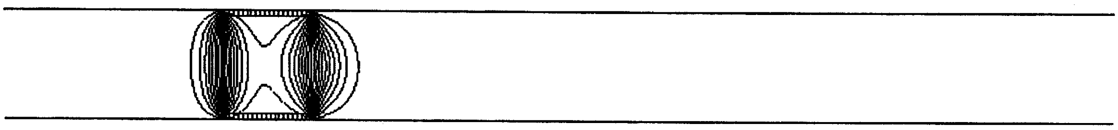


Fig.15 Isotherms at T=30 obtained by the standard MAC method

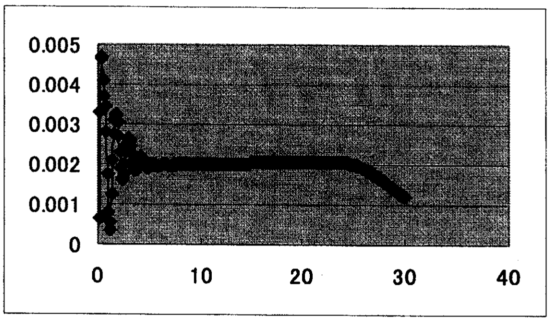
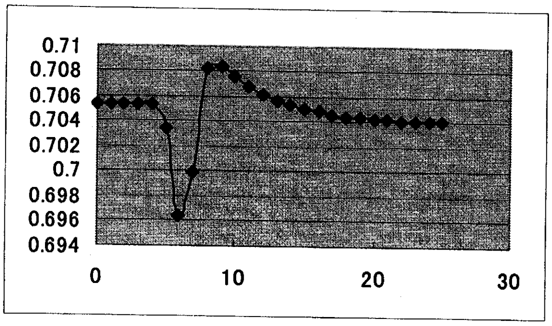
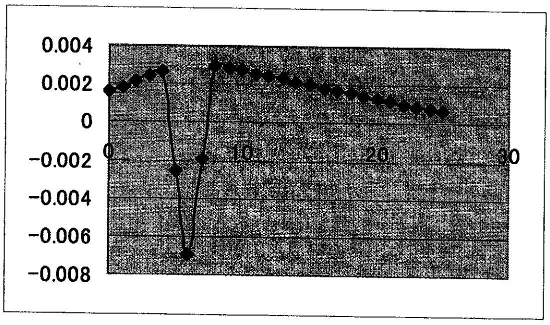


Fig.16 Time history of flow rate obtained by the standard MAC method


0.71
0.708
0.706
0.704
0.702
0.7
0.698
0.696
0.694
0
10
20
30


0.004
0.002
0
-0.002
-0.004
-0.006
-0.008
0
10
20
30