A New Numerical Method for Some Region of Very Large Aspect Ratio with Free Surface like River

Kanako IKEDA, Anna KUWANA, and Tetuya KAWAMURA

(Received December 10, 2012)

Abstract
A new numerical method that is suitable for the calculation of flows in a very long region is proposed. The original flow is expressed as sum of the "main flow" obtained by solving one-dimensional Navier-Stokes equation analytically and their variations. Applying this method to incompressible flows in a very long pipe, we obtained good results. The method is now extended to the flow with free surface such as river flows. In this study, flows with free surface in very long and not flat wall regions are calculated and are examined for the effectiveness of the proposed method.

1. Introduction

We focus on the flow with free surface in a very long region like a river. This kind of flow exist in reality and important for the prevention against disaster. The main problem to calculate the flow in such long region is that it is quite difficult to satisfy the equation of continuity precisely. If we use the method based on the stream-function and vorticity, we are free from problem mentioned above. However this method works only on the two dimensional or axi-symmetric flows. Moreover, if the region involves a branch or confluence, it is difficult to determine the flow rate there. Therefore the MAC method[1] and its variations[2]-[4] are preferred to treat three-dimensional flow and the flow in regions of complicated shapes. On the other hand, the conservation of mass is only approximately satisfied by the method based on the Poisson equation of pressure appearing in the MAC method that is difficult to converge by using the iterative method. This disadvantage is emphasized in the case of the region such as a river flow.

We have developed a new numerical method that is suitable for the computation of incompressible flow where one dominant flow exists such as a channel or a pipe flows[5]-[7]. In this method, the original flow is expressed as sum of the "main flow" and the variation from the main flow. The former is obtained by solving a one-dimensional Navier-Stokes equation analytically. In this paper, we apply this method to a flow with the free surface and examine the effectiveness of the method mentioned above.

2. Numerical Method

If the computational region is very long in x-direction, the flow can be treated as nearly one dimensional time-dependent flow. In other words, we can assume

$$u = u(x,t), \quad p = p(x,t), \quad v = w = 0, \quad \frac{\partial}{\partial y} = \frac{\partial}{\partial z} = 0 \tag{1}$$

as the first approximation. Substituting equation (1) into Navier-Stokes Equations, we obtain

$$\frac{\partial u}{\partial x} = 0 \tag{2}$$

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} = -\frac{\partial p}{\partial x} + \frac{1}{\mathrm{Re}}\frac{\partial^2 u}{\partial x^2} \tag{3}$$

where Re is Reynolds number. We neglect the gravity term here, because we need a simple one-dimensional flow. From equation (2), we obtain

$$u = f(t) \tag{4}$$

as a "main flow" or "dominant flow". The function $f(t)$ is determined by the boundary condition at the entrance of the long region (i.e. inflow). If we substitute equation (4) and the first one of equation (1) into equation (3), we obtain

$$f'(t) = -\frac{\partial p}{\partial x} \qquad i.e. \qquad p = -f'(t)x + C(t) \qquad (C(t)\text{:Arbitrary function}) \tag{5}$$

From equations (4), (5), we can express the original flow as sum of the "main flow" and the "variation of the velocity and pressure $(\tilde{u}, \tilde{p})$" as follows:

$$u = f(t) + \tilde{u}, \quad p = -f'(t)x + C(t) + \tilde{p} \tag{6}$$

After the substitution of equation (6) into Navier-Stokes equation, we obtain the basic equation as follows:

$$\frac{\partial \tilde{u}}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \tag{7}$$

$$\frac{\partial \tilde{u}}{\partial t} + (f + \tilde{u})\frac{\partial \tilde{u}}{\partial x} + v\frac{\partial \tilde{u}}{\partial y} + w\frac{\partial \tilde{u}}{\partial z} = -\frac{\partial \tilde{p}}{\partial x} + \frac{1}{\mathrm{Re}}\left(\frac{\partial^2 \tilde{u}}{\partial x^2} + \frac{\partial^2 \tilde{u}}{\partial y^2} + \frac{\partial^2 \tilde{u}}{\partial z^2}\right) \tag{8}$$

$$\frac{\partial v}{\partial t} + (f + \tilde{u})\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z} = -\frac{\partial \tilde{p}}{\partial y} + \frac{1}{\mathrm{Re}}\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right) \tag{9}$$

$$\frac{\partial w}{\partial t} + (f + \tilde{u})\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z} = -\frac{\partial \tilde{p}}{\partial z} + \frac{1}{\mathrm{Re}}\left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2}\right) - g \tag{10}$$

where g is the gravity. These equations are nearly the same as the original Navier-Stokes equation although unknowns are different. These equations can be solved by the standard method such as the MAC method and so on. Note that the boundary condition on the wall (no-slip: $u = 0$) becomes $\tilde{u} = -f(t)$.

Movement of grid points on the free surface is shown in figure 1. After the decision of shape of free surface at $t + \Delta t$, internal grid points are reallocated in a vertical direction (along z-axis).

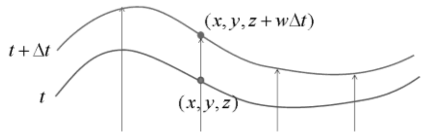
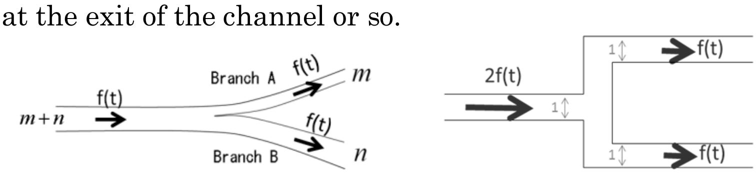


Fig. 1 Move of free surface.

If the channel has branches, the "main flows" in each branch is adjusted so that flow rates through the channel become the same value. For example, in a channel with two branches whose width is m and n respectively, if the width of channel before the branches is $m+n$, as is shown in figure 2(a), the value of the "main flow" is the same in whole channel (on the other hand, if sum of width of two branches is twice as it of before the branches, the "main flows" in branches is 1/2 of it in before the branches as is shown in figure 2(b)).

The velocities of actual flow in the branches do not $m:n$ etc. because of viscosity or the boundary conditions at the exit of the channel or so. But this method is based on an idea of "main flow" plus "their variations", so the "variations" obtained from calculation involve the effect of viscosity or the boundary conditions at the exit of the channel or so.



(a) Same width. (b) Different width.

Fig. 2 Schematic figures of flow channel with branches and before them.

Figure 3 shows that the shape of a typical computational domain in this study. The aspect ratio (length / diameter) is about 25. Pulsatile flow comes in at the entrance of the channel (the left hand side of figure 3). The exit of the channel (the right hand side of figure 3) is the free outflow, both sides of the channel is no-slip. The gradient from the entrance to the exit of the channel is about 1/1000, the Froude number is about 0.6, and the Reynolds number is set to 1000.

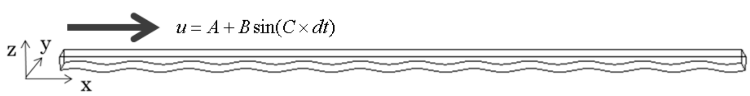
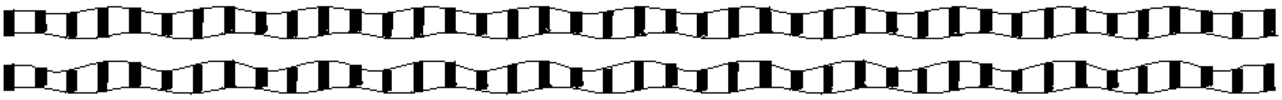


Fig. 3 Typical computational domain in this study with pulsatile inflow.
(A, B, C: reasonable constants)

3. Results

3.1 Channel with Concave-Convex Undersurface

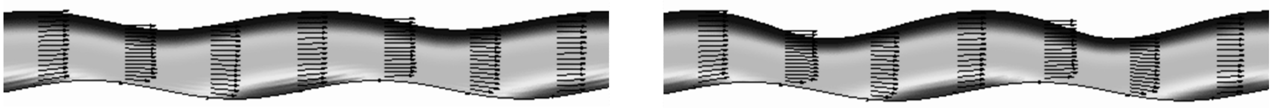
First, calculations are carried out in the channel whose undersurface is concave-convex with pulsatile inflow as shown in figure 3. Since the grid along y-axis is uniform, two-dimensional general coordinate transformation is used for two other directions. Figures show flow fields on x-z surface at the center of the channel. Figure 4 shows the velocity vectors. Pulsatile flow comes in from the left hand side. Because of the free surface, shape of the top surface is varying with time. Figure 5 is close up near middle of the channel. The color shading shows the pressure field. The front surfaces of the convex of undersurface are subject to high pressure. Figure 6 shows the time history of variation of the flux ($\sum(u \times dydz)$) of the channel at the entrance (inflow) and exit (outflow) of the channel. Figure 6(a) shows that flux at the entrance is kept whole of the channel by the proposed method. On the other hand, figure 6(b) shows that flux at the entrance is not kept by the conventional MAC method whose computational cost is in the same range with proposed method of figure 6(a), where the flux decreases with distance from the entrance of the channel. Flux calculated by the conventional method can be kept if the conservation of mass is satisfied very well by the method based on the Poisson equation of pressure, of course.



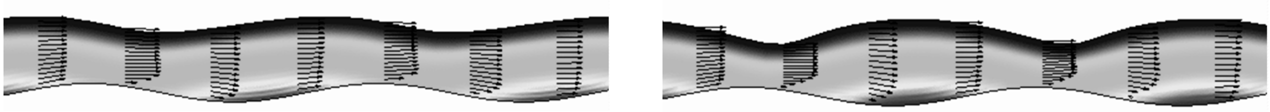
(a) Proposed method.

(b) Conventional method, same computational cost with (a).

Fig. 4 Velocity vector of the channel (at Timestep=5000, 10000 respectively).

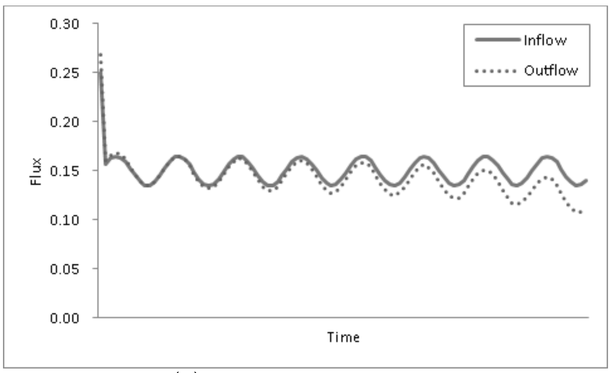


(a) Proposed method.

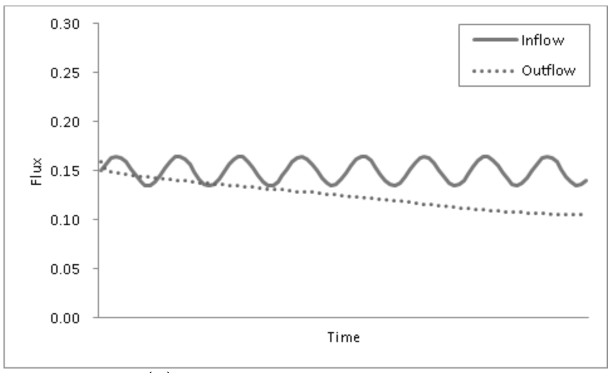


(b) Conventional method, same computational cost with (a).

Fig. 5 Velocity vector and pressure field of the channel (at Timestep=5000, 10000 respectively).



(a) Proposed method.



(b) Conventional method, same computational cost with (a).

Fig. 6 Time history of variation of the flux of the channel.

3.2 Channel with Concave-Convex Undersurface and One of the Side Surface

Second, calculation is carried out in the channel whose undersurface and one of the side surface are concave-convex with pulsatile inflow as shown in figure 7. Three-dimensional general coordinate transformation must be used in this case. Figure 8 is close up near middle of the channel. Figure 9 shows the time history of variation of the flux at the entrance and exit of the channel. The same results as **3.1** are confirmed.

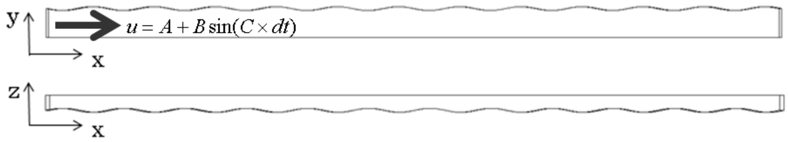
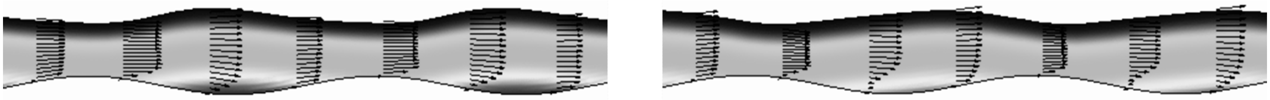


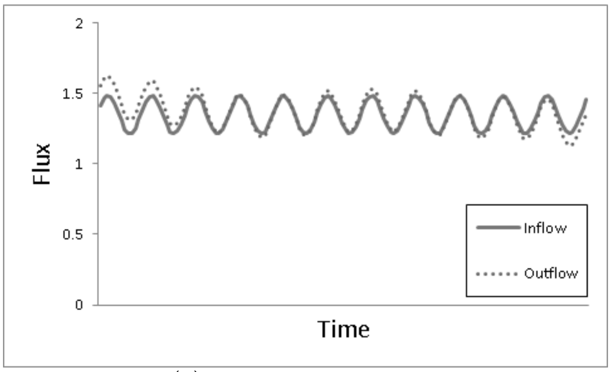
Fig. 7 Computational domain of concave-convex undersurface and one of the side surface.



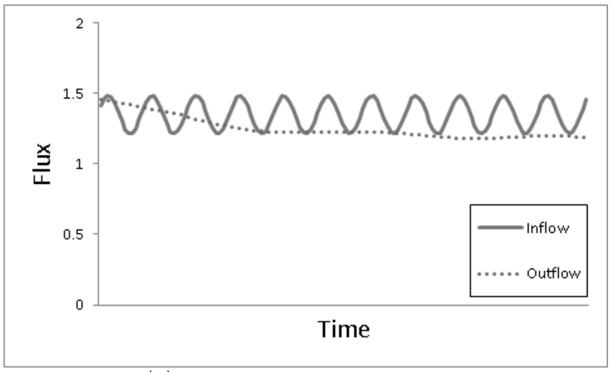
(a) Proposed method.

(b) Conventional method, same computational cost with (a).

Fig. 8 Velocity vector and pressure field of the channel (at Timestep=5000, 10000 respectively).



(a) Proposed method.



(b) Conventional method, same computational cost with (a).

Fig. 9 Time history of variation of the flux of the channel.

3.3 Channel with branches

Calculation is carried out in the channel with two branches with pulsatile inflow as shown in figure 10. Cross sectional areas of the channel before branches and branch A and B are 3: 1: 2 respectively. Since the cross sectional area of channel before the branches is the same as the sum of two branches, the value of the “main flow” is the same in whole channel.

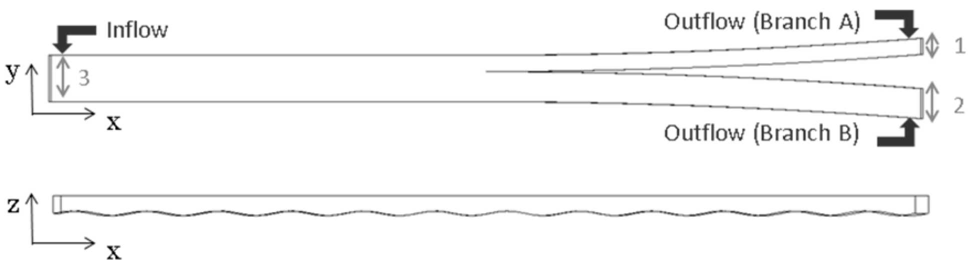
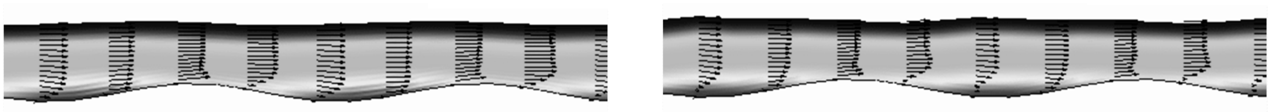
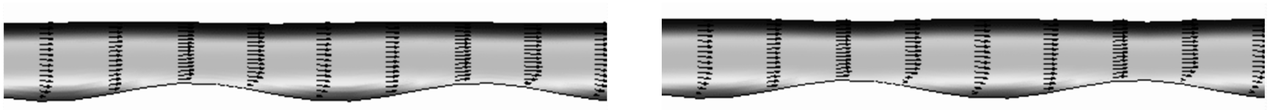


Fig. 10 Computational domain of a channel with branches.

Figure 11 is close up near middle of the branch B. The position of drawing area is shown in figure 12. Figure 13 shows the time history of variation of the flux at the entrance and exit of two branches, and sum of the flux of two branches. Flux at the entrance and sum of the flux at the exit of two branches in figure 13(a) shows that flux at the entrance is kept whole of the channel by the proposed method, on the other hand, figure 13(b) shows that flux at the entrance is not kept by the conventional MAC method whose computational cost is in the same range with proposed method of figure 13(a). Therefore the same results as **3.1** are confirmed.



(a) Proposed method.



(b) Conventional method, same computational cost with (a).

Fig.11 Velocity vector and pressure field of the channel (at Timestep=5000, 10000 respectively).

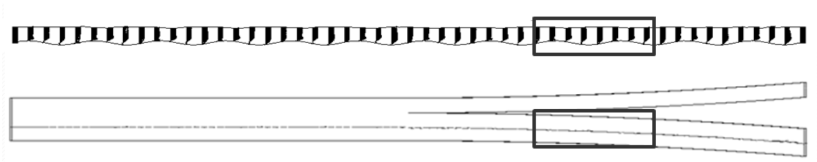
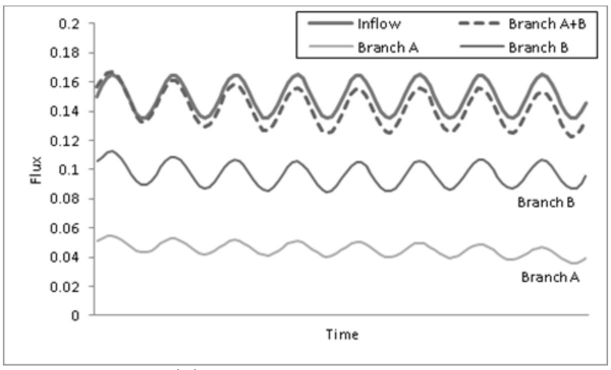
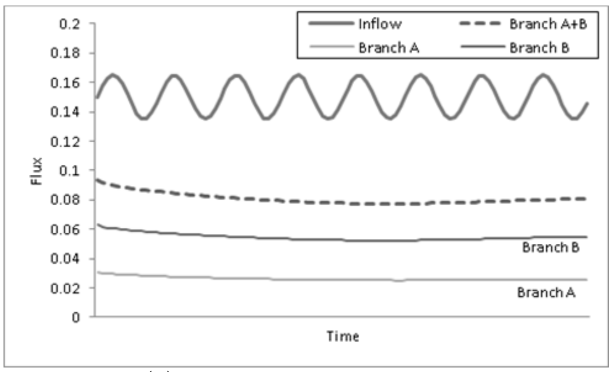


Fig. 12 The position of drawing area of figure 11.



(a) Proposed method.



(b) Conventional method,
same computational cost with (a).

Fig. 13 Time history of variation of the flux of the channel.

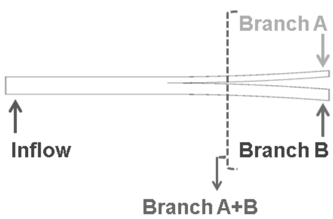


Fig. 14 The position of measurement points of figure 13.

4. Conclusion

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.

In our previous study, the method that is effective for the calculations of incompressible flow in a long region is proposed. The idea of the method is that the original flow is expressed as sum of the "main flow" and the variation from the main flow.

In this study, we apply this method to flow with free surface with concave-convex walls and branches. Three-dimensional flow with pulsatile inflow is simulated using this method. It is found that the conservations of mass of the fluid is well satisfied, while it is not precisely by the calculation based on the conventional MAC method whose computational cost is the same range with proposed method. Although, it is well satisfied by the conventional MAC method if the conservation of mass is satisfied very well by the method based on the Poisson equation of pressure, the computational cost becomes high in such case.

Acknowlegment

This work was supported by KAKENHI (Grant-in-Aid for Scientific Research(C): 22560157).

References

[1] F.H. Harlow and J.E. Welch, "Numerical calculation of time-dependent viscous incompressible flow of fluid with free surface" *The Physics of Fluids*. **8** (1965), pp.2182-2189.
[2] A.A. Amsden and F.H. Harlow, "A simplifie MAC technique for incompressible fluid flow calculations" *J. Comp. Phys.*, **6**(2), (1970), pp.322-325.
[3] A.J.Chorin, "Numerical solution of the Navier-Stokes equations", *Math. Comp.*, **22**, (1968), pp. 745-762.
[4] N.N.Yanenko, "The method of fractional steps", Springer-Verlag, New York, (1971)
[5] T. Kawamura, F. Oono and Y. Nakamura, "A numerical method for calculations of flow in long region with branch" *CFD2006,* (2006), C3-5.
[6] T. Kawamura and K. Miyashita, "A numerical method for calculations of flow in long region with branch" *Natural Science Report, Ochanomizu University,* **58** (2007), pp.45-55.
[7] Y. Sato and T. Kawamura, "New numerical method for calculation of flow in a long region with free surface", *Theoretical and Applied Mechanics Japan*, **Volume 60-** 60th Japan National Congress for Theoretical and Applied Mechanics, (2011), pp.155-162.

Kanako IKEDA
Graduate School of Humanities and Sciences, Advanced Sciences, Computer Science, Ochanomizu University
Otsuka 2-1-1, Bunkyo-ku, Tokyo 112-8610, Japan
E-mail: ikeda.kanako@is.ocha.ac.jp

Anna KUWANA
Center for Simulation Sciences, Ochanomizu University
E-mail: kuwana.anna@ocha.ac.jp

Tetuya KAWAMURA
Graduate School of Humanities and Sciences, Ochanomizu University
E-mail: kawamura@is.ocha.ac.jp