Numerical Simulation of Incompressible Flows with Heat Transfer using Seamless Immersed Boundary Method*

Kyohei TAJIRI**, Hidetoshi NISHIDA*** and Mitsuru TANAKA***
** Department of Mechanical and System Engineering, Kyoto Institute of Technology
Matsugasaki, Sakyo-ku, Kyoto-shi, Kyoto, Japan
E-mail: d2821002@edu.kit.ac.jp
*** Department of Mechanical and System Engineering, Kyoto Institute of Technology
Matsugasaki, Sakyo-ku, Kyoto-shi, Kyoto, Japan

Abstract
In this paper, the numerical simulations of incompressible flow with heat transfer are presented by using the seamless immersed boundary method on the Cartesian grid. In the seamless immersed boundary method, the forcing term is added not only on the grid points near the boundary but also on the grid points inside the boundary (solid region) in order to satisfy the velocity boundary condition. Then, the seamless physical quantities, e.g., the pressure, can be obtained, so that the characteristic quantities on the boundary can be estimated precisely. The present seamless immersed boundary method is applied to the energy equation. Then, the temperature satisfying the boundary condition can be easily obtained as well as velocity on the Cartesian grid. The present method is applied to flows around an object with the moving boundary and heat transfer.

Key words: Computational Fluid Dynamics, Computational Method, Heat Transfer, Immersed Boundary Method

1. Introduction
In recent years, the flow problem handled by numerical simulation becomes more broadly and complicated. Although the boundary fitted coordinates are usually adopted for the flow around the object with complex shape, the grid generation may sometimes consume huge time in the practical flow simulation. Therefore, the Cartesian coordinates are spotlighted again, because of its easy grid generation and high computational efficiency. Today, the Cartesian grid approach is adopted in the many flow simulations around the single or multiple objects with complex shape. Also, this computational trend is expected to continue in the flow simulation.

In the Cartesian grid approach, the immersed boundary method (IBM)1 is usually used. Also, the improved method of IBM, i.e., the seamless immersed boundary method, is proposed2. In order to satisfy the velocity and temperature conditions on the (virtual) boundary points, IBM requires the external forcing term and heat flux term added to the momentum equations and energy equation.

The seamless IBM is applied to the flow simulation with moving boundary3 and with heat transfer4 successfully. In the moving boundary problem with heat transfer, however, the seamless IBM has not validated yet. In this paper, the seamless IBM is validated for the incompressible flow around a moving object with heat transfer. First, in order to validate the present seamless IBM, the numerical simulations of incompressible flows around a 2D heated circular cylinder and a 2D oscillating heated circular cylinder with isothermal boundary condition are considered. Finally, the numerical simulations of flow around a swimming fish model with isothermal boundary condition are carried out by using the present seamless IBM.
2. Seamless Immersed Boundary Method

2.1. Governing equations

The incompressible viscous flows with heat transfer are governed by the following continuity equation, incompressible Navier-Stokes equations and energy equation.

\[
\frac{\partial u_i}{\partial x_i} = 0 \tag{1}
\]

\[
\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{1}{Re} \frac{\partial^2 u_i}{\partial x_i \partial x_j} + G_i \tag{2}
\]

\[
\frac{\partial T}{\partial t} + u_j \frac{\partial T}{\partial x_j} = \frac{1}{Pr \cdot Re} \frac{\partial^2 T}{\partial x_i \partial x_j} + Q \tag{3}
\]

Where, \( Re \) denotes the Reynolds number defined by \( Re = \frac{L_0 U_0}{\nu_0} \) and \( Pr \) denotes the Prandtl number defined by \( Pr = \frac{\nu_0}{\alpha_0} \). \( U_0, L_0, \nu_0 \) and \( \alpha_0 \) are the reference velocity, the reference length, the kinematic viscosity and the thermal diffusivity, respectively. The last term in Eq.(2), (3), \( G_i \) and \( Q \) denote the additional forcing term and heat flux term in the seamless IBM.

2.2. Forcing term and heat flux term estimations

In order to estimate the additional forcing term in the governing equations, \( G_i \), there are mainly two ways, that is, the feedback\(^{(6),(7)}\) and direct\(^{(5)}\) forcing term estimations. In this paper, the direct forcing term estimation shown in Fig.1 is adopted.

For the forward Euler time integration, the forcing term can be determined by

\[
G_i = \left[ u_j \frac{\partial u_i}{\partial x_j} - \frac{\partial p}{\partial x_i} + \frac{1}{Re} \frac{\partial^2 u_i}{\partial x_i \partial x_j} \right] + \frac{\bar{U}_i^{n+1} - u_i^n}{\Delta t}, \tag{4}
\]

where \( \bar{U}_i^{n+1} \) denotes the interpolated velocity by the linear interpolation. Namely, the external force is specified as the velocity components at next time step satisfy the relation, \( U_i^{n+1} = \bar{U}_i^{n+1} \). \( U_{ib} \) denotes the specified velocity on the virtual boundary point. When the velocity on the virtual boundary point is stationary, it is defined by \( U_{ib} = 0 \), and it is defined by \( U_{ib} = V_{move} \) for the moving boundary. \( V_{move} \) is the moving speed of the virtual boundary point.

Next, the estimation of the additional heat flux term in the energy equation is explained. In the case of isothermal condition, the additional heat flux term can be determined by the same procedure as velocity. For the forward Euler time integration, the heat flux term can be determined by

\[
Q = \left[ u_j \frac{\partial T}{\partial x_j} - \frac{1}{Pr \cdot Re} \frac{\partial^2 T}{\partial x_i \partial x_j} \right] + \frac{\bar{T}^{n+1} - T^n}{\Delta t}, \tag{5}
\]

where \( \bar{T}^{n+1} \) is determined by the linear interpolation.

![Fig. 1 Direct forcing term estimation.](image-url)
2.3. Seamless immersed boundary method

In conventional IBM by using aforementioned forcing term estimation, the grid points added forcing term are restricted near the boundary. Then, the pressure distributions near the boundary show the unphysical oscillations because of the pressure jump. Also, the non-negligible velocity appears inside the boundary. In order to remove the unphysical oscillations near the boundary, the seamless IBM was proposed(2). In the seamless IBM, the forcing term is added not only near the boundary but also in the region inside the boundary shown in Fig.2.

In the region inside the boundary, the forcing term is determined by satisfying the relation, \( \bar{U}_i^{n+1} = U_b \), where \( U_b \) is the specified velocity, e.g., \( U_b = 0 \) in the stationary solid media. For the heat flux term, similarly, when \( T_b \) is the specified temperature (e.g., \( T_b = 1 \) in the case of isothermal condition), the heat flux term can be determined by satisfying the relation \( \bar{T}_n^{n+1} = T_b \). Then, the hydrodynamic force can be estimated more correctly.

![Fig. 2  Grid points added forcing and heat flux terms.](image)

2.4. Numerical method

The incompressible Navier-Stokes equations (2) are solved by the second order finite difference method on the collocated grid arrangement. The convective terms are discretized by the second order fully conservative finite difference method(8). The diffusion and pressure terms are discretized by the conventional second order centered finite difference method. For the time integration, the fractional step approach based on the two-step Runge-Kutta scheme (10) is applied. For the incompressible Navier-Stokes equations in the seamless IBM, the two-step Runge-Kutta scheme is written by

\[
\begin{align*}
  u_i^{n+1} &= (1 - \gamma)u_i^n + \gamma u_i^{n-1} + \Delta t \left[ \alpha \left( F_i^n - \nabla p_i^n + G_i^n \right) + \beta \left( F_i^{n-1} - \nabla p_i^{n-1} + G_i^{n-1} \right) \right], \\
  u_i^* &= (1 - \gamma)u_i^n + \gamma u_i^{n-1} + \Delta t \left[ \alpha F_i^n + \beta \left( F_i^{n-1} - \nabla p_i^{n-1} + G_i^{n-1} \right) \right], \\
  u_i^{n+1} &= u_i^* + \Delta t \alpha \left( -\nabla p_i^n + G_i^n \right),
\end{align*}
\]

(6)
(7)

where \( \alpha, \beta \) and \( \gamma \) are the parameters of two-step Runge-Kutta scheme and \( F_i \) denotes the convective and diffusion terms. In this paper, the parameters are set as \( \alpha = 11/8, \beta = -5/8 \) and \( \gamma = -1/4 \) in order to satisfy the second order of time accuracy. The time integration of energy equation is also performed by the same procedure. The fractional step approach based on the two-step Runge-Kutta scheme can be written by

\[
\begin{align*}
  u_i^{*} &= (1 - \gamma)u_i^n + \gamma u_i^{n-1} + \Delta t \left[ \alpha F_i^n + \beta \left( F_i^{n-1} - \nabla p_i^{n-1} + G_i^{n-1} \right) \right], \\
  u_i^{n+1} &= u_i^* + \Delta t \alpha \left( -\nabla p_i^n + G_i^n \right),
\end{align*}
\]

where \( u_i^* \) denotes the fractional step velocity. The resulting pressure equation is solved by the SOR method. The convergence criterion of the pressure equation is set as \( p_{L_2} < 1.0 \times 10^{-6} \), where \( p_{L_2} \) is the \( L_2 \) residual of the pressure. Then, the conservation of mass is satisfied in the range of convergence criterion of the pressure equation.
3. Validation of Seamless Immersed Boundary Method

3.1. Flow around a 2D heated circular cylinder

In order to validate the seamless IBM, the flow around a 2D heated circular cylinder is considered. The computational domain is shown in Fig. 3. $D$ denotes a diameter of the circular cylinder. The grid resolution is $343 \times 303$ with non-uniform grid system. The impulsive start determined by the uniform flow is adopted. On the inflow boundary, the velocity and temperature are fixed by the uniform flow ($u = 1, v = 0, T = 0$) and the pressure is imposed by the Neumann condition obtained by the normal momentum equation. The velocity and temperature are extrapolated from the inner points and the pressure is obtained by the Sommerfeld radiation condition\(^\text{11}\) on the outflow and side boundaries. On the virtual boundary and inside the boundary, the non-slip ($u = 1, v = 0$) and the isothermal ($T = 1$) conditions are imposed. The Reynolds number is set as $Re = 200, 218$ and the Prandtl number is set as $Pr = 0.717$. The time increment is set as $\Delta t = 0.0005$.

Figure 4 shows the temperature contours with $Re = 200$. The Karman vortex sheet behind cylinder is observed clearly. The heat (temperature) is transported downstream by the vortex sheet. In the close-up views, it is observed that isothermal condition is satisfied on the virtual boundary and inside the boundary. Figure 5 shows the time averaged local Nusselt number on the circular cylinder surface with $Re = 218$. The local Nusselt number on the circular cylinder surface is defined by

$$Nu(\theta) = -\frac{D}{T_w - T_{\infty}} \frac{\partial T(\theta)}{\partial r}, \quad (9)$$

where, $T_w$ denotes the temperature on the circular cylinder surface and $\frac{\partial T(\theta)}{\partial r}$ denotes the temperature gradient of normal direction. In this paper, the present Nusselt number was averaged between five periods of oscillation. The present Nusselt number is in very good agreement with the reference ones\(^\text{12}–\text{14}\).
3.2. Flow around a 2D oscillating heated circular cylinder

In order to validate the present method for the moving boundary, the flow around a 2D oscillating heated circular cylinder is considered. The computational domain is shown in Fig. 6. The circular cylinder in the uniform flow moves vertically as,

\[ y(t) = y_0 + \frac{y_{\text{amp}}}{2} \sin(2\pi ft), \]  

where the amplitude is \( y_{\text{amp}}/2 = 0.2D \) and the non-dimensional frequency is \( f = 0.2 \). The initial location of circular cylinder is \((x_0, y_0) = (5D, 5D)\). Computational grid is the hierarchical Cartesian grid with 4 levels (shown in Fig. 7). The grid resolution is 0.0125D at near the circular cylinder. The initial and boundary conditions are the same as the previous simulation.

On the virtual boundary and inside the boundary, the moving circular cylinder velocity \((u = 0, v = \frac{dy}{dt})\) and isothermal \((T = 1)\) conditions are imposed. The Reynolds number and the Prandtl number are set as \( Re = 200 \) and \( Pr = 0.717 \). The time increment is set as \( \Delta t = 0.001 \).

Figure 8 shows the temperature contours at top and bottom dead centers. The heat (temperature) is transported downstream by the vortex sheet. In the close-up views, it is observed that isothermal condition is satisfied on the virtual boundary and inside the boundary. The present drag and lift coefficients and the Strouhal number shown in Table 1 are in very good agreement with the reference result\(^{(15)}\). The present drag and lift coefficients are defined by

\[ C_D = \frac{-2 \int_s (G_x - u \frac{\partial u}{\partial x} - v \frac{\partial u}{\partial y} - \frac{\partial u}{\partial t}) \, ds}{\rho U_0^2 S}, \]  
\[ C_L = \frac{-2 \int_s (G_y - u \frac{\partial v}{\partial x} - v \frac{\partial v}{\partial y} - \frac{\partial v}{\partial t}) \, ds}{\rho U_0^2 S}, \]  

where, \( S \) denotes the frontal projected area. Figure 9 shows the time averaged local Nusselt number on the circular cylinder surface. The present Nusselt number was averaged between
five periods of oscillation. At $t = m/f (m = 0, 1, 2, \cdots)$ which the upward velocity is maximum, the time averaged local Nusselt number in upper side becomes larger than that in lower side. At $t = (2m + 1)/2f$ which the downward velocity is maximum, conversely, the time averaged local Nusselt number in lower side becomes larger than that in upper side. It corresponds that the heat which goes around upper half of a heated circular cylinder becomes larger at $t = m/f$ and the reverse phenomenon appears at $t = (2m + 1)/2f$.

![Overall view (at top dead center).](image1)

![Overall view (at bottom dead center).](image2)

![Close-up view (at top dead center).](image3)

![Close-up view (at bottom dead center).](image4)

Fig. 8 Snap shot of temperature contours ($Re = 200$).

![Time averaged local Nusselt number (Re = 200).](image5)

Fig. 9 Time averaged local Nusselt number ($Re = 200$).

Table 1  Comparison of characteristic quantities for flow around a oscillating circular cylinder ($Re = 200$).

<table>
<thead>
<tr>
<th></th>
<th>$C_D$</th>
<th>$C_L$</th>
<th>$St$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Present</td>
<td>$1.59 \pm 0.21$</td>
<td>$\pm 0.57$</td>
<td>$0.20$</td>
</tr>
<tr>
<td>Wu et al.$^{[3]}$</td>
<td>$1.58 \pm 0.20$</td>
<td>$\pm 0.58$</td>
<td>$0.20$</td>
</tr>
</tbody>
</table>
4. Application to Flow around a Swimming Fish Model

In this paper, we consider the acheilognathus moriokae (Javaese bitterling) in the uniform flow. The surface shape generated by the triangular polygon is shown in Fig. 10. The reference length $D$ is body length of a fish and the total number of triangular polygons is 5866. The characteristic shape of a fish, i.e., the breast fins, abdomen fins, hip fin, back fin and tail fin, can be reappeared. Figures 11 and 12 show the computational domain and the used hierarchical grid with 5 levels.

The fish model is controlled by 2D rotation on four rotating axes ($S_1, S_2, S_3, S_4$) shown in Fig.13. $S_1$ is fixed at $(x, y) = (5.2D, 5.5D)$. The rotating angle on each rotating axis is defined by

$$\begin{align*}
\theta_1 &= -\psi_1 \sin(2\pi n_f / N_f) \\
\theta_2 &= -\psi_2 \sin(2\pi n_f / N_f) \\
\theta_3 &= -\psi_3 \sin(2\pi n_f / N_f) \\
\theta_4 &= -\psi_4 \sin(2\pi n_f / N_f - \pi / 2)
\end{align*}$$

where $n_f$ is the frame number and $N_f$ denotes the total number of frames ($N_f = 200$). $\psi_i$ is the maximum rotating angle, i.e., $\psi_1 = \pi / 90, \psi_2 = \pi / 45, \psi_3 = \pi / 18, \text{and } \psi_4 = \pi / 12$ in this paper. The swimming behavior is shown in Fig.14.

Fig. 10 Surface shape of a fish model.

Fig. 11 Computational domain.

Fig. 12 Hierarchical grid with 5 levels.
The initial and boundary conditions are the same as the previous simulation. On the virtual boundary and inside the boundary, the moving velocity \( u_i = \overline{u_i}_{move} \) and isothermal \( T = 1 \) conditions are imposed. \( \overline{u_i}_{move} \) is the moving velocity determined by Eq.(13).

The simulation with \( Re = 100 \) and \( Pr = 7.17 \) and \( \Delta t = 0.0005 \) is executed. Figure 15 shows the pressure field and Fig.16 shows the temperature field on \( x-y \) plane (\( z = 5.45D \)). At the tail fin, the high pressure area appears in the motion direction side of the tail fin. And, in the reverse side, the low pressure area appears. The heat (temperature) is transported to downstream by swimming motion of the tail fin. Figure 17 shows the second invariant of velocity gradient tensor (\( Q = 0.1 \)) and temperature (\( T = 0.2 \)). The white surface denotes the second invariant of velocity gradient tensor and the red surface denotes the temperature. The vortices are formed near each fin by swimming motion. These vortices near the tail fin are transported to downstream. And, the heat (temperature) is also transported to downstream with vortices transport. At the rear of tail fin, the large vortices are formed, because of the effect of large viscosity.
Fig. 15  Pressure distributions on x-y plane ($z = 5.45D$).

Fig. 16  Temperature distributions on x-y plane ($z = 5.45D$).
5. Concluding Remarks

In this paper, the seamless IBM is applied to the numerical simulation of incompressible flow around a moving object with heat transfer. In the flows around a 2D heated circular cylinder and a 2D oscillating heated circular cylinder with isothermal boundary condition, the present method gives the appropriate flows fields with the smooth pressure distributions and the specified velocity and temperature conditions on the virtual boundary and inside the boundary. And, it is observed that heat (temperature) is transported downstream by the vortex sheet. Finally, the present method applied to the swimming fish model with isothermal boundary condition in uniform flow. The vortices are formed near each fin by swimming motion. The heat transportation in the downstream by these vortices was observed. Then, it is concluded that the present seamless IBM is very fruitful in the numerical simulation of incompressible flow around a moving object with heat transfer.

References


