Boundary treatment for the unsteady surface velocity in an immersed boundary method†

Seung Jeon and Haecheon Choi*

School of Mechanical and Aerospace Engineering, Seoul National University, Seoul 151-744, Korea

(Manuscript Received May 15, 2009; Revised May 26, 2009; Accepted May 26, 2009)

Abstract

In the present paper, we demonstrate that an unsteady velocity boundary condition on a solid surface induces artificial force oscillations on the solid body, and these oscillations can be effectively removed by imposing the Neumann boundary condition on the solid surface for the pseudo-pressure Poisson equation.

Keywords: Immersed boundary method; Artificial force oscillation; Poisson equation; Neumann boundary condition

1. Introduction

Since Peskin [1] first developed the immersed boundary (IB) method for flow over a complex geometry, various versions of immersed boundary methods have been developed aiming for better accuracy, simpler implementation, or less CPU time. A review on the immersed boundary method can be found in Mittal and Iaccarino [2]. In most studies on the immersed boundary method, however, the boundary condition on the solid surface is steady, or focus is given to the flow field rather than to the force on the body even when the boundary condition on the solid surface is unsteady.

In the present paper, we show in the framework of the immersed boundary method (e.g., [3]) that artificial force oscillations on the body occur when an unsteady boundary condition is given at the solid boundary. These force oscillations are clearly non-physical and sometimes result in the blow-up of numerical solutions. We suggest a new boundary treatment for the pseudo-pressure Poisson equation, which is shown below, to remove effectively the artificial force oscillations on the body.

2. Violation of mass conservation

The governing equations for the unsteady incompressible flow are

\[
\begin{align*}
\frac{\partial u_i}{\partial t} + \frac{\partial u_i u_j}{\partial x_j} &= \frac{\partial p}{\partial x_i} + \frac{1}{Re} \frac{\partial^2 u_i}{\partial x_i}, \\
\frac{\partial u_i}{\partial x_i} &= q = 0,
\end{align*}
\]

where \( t \) is time; \( x_j \) is the coordinate; \( u_i \) is the corresponding velocity component; \( p \) is the pressure; \( f \) and \( q \) are the momentum forcing and mass source/sink, respectively, proposed by [3]; and \( Re = u_\infty d/\nu \) is the Reynolds number. Here, \( u_\infty \) is the free-stream velocity; \( d \) is the characteristic length; and \( \nu \) is the kinematic viscosity. To solve Eqs. (1) and (2), a fractional step method is used together with a semi-implicit time advance scheme (a third-order Runge-Kutta method [RK3] for the convective term and a second-order Crank-Nicolson method for the diffusion term) as follows:

† This paper was recommended for publication in revised form by Associate Editor Dongshin Shin
*Corresponding author. Tel.: +82 2 880 8361, Fax.: +82 2 878 3662
E-mail address: choi@snu.ac.kr
© KSME & Springer 2009
\[ \frac{\dot{u}_i^k - u_i^{k-1}}{\Delta t} = \alpha_i L(u_i^k) + \alpha_{ij} L(u_j^{k-1}) - 2\alpha_i \frac{\partial p_i^k}{\partial x_i} - \gamma_i N(u_i^{k-1}) - \beta_i N(u_i^{k-2}) + f_i^k, \]

(3)

\[ \frac{\partial \phi_i}{\partial x_i} = \frac{1}{2\alpha_i \Delta t} \left( \frac{\partial u_i^{k-1}}{\partial x_i} - q_i^k \right), \]

(4)

\[ u_i^k = u_i^{k-1} - 2\alpha_i \Delta t \frac{\partial \phi_i^k}{\partial x_i}, \]

(5)

\[ p^k = p^{k-1} + \phi^k - \frac{\alpha_k \Delta t}{\text{Re}} \frac{\partial \phi_i^k}{\partial x_i} \]

(6)

where \( L(u_i) = (1/\text{Re}) \frac{\partial^2 u_i}{\partial x_i^2} \), \( N(u_i) = \frac{\partial u_i}{\partial x_i} \), \( \dot{u}_i \) is the intermediate velocity; \( \phi \) is the pseudo-pressure; \( \Delta t \) is the computational time step; \( k \) is the substep index; and \( \alpha_k, \gamma_k, \) and \( \rho_k \) are the coefficients of RK3 (\( \alpha_1 = 4/15, \gamma_1 = 8/15, \rho_1 = 0; \alpha_2 = 1/15, \gamma_2 = 5/12, \rho_2 = -17/60; \alpha_3 = 1/6, \gamma_3 = 3/4, \rho_3 = -5/12 \)). The momentum forcing \( f_i \) is given on or near the immersed boundary to satisfy the no-slip velocity conditions, \( u_i \) and \( v_i \), at the immersed boundary (Fig. 1).

The continuity equation should be satisfied for the cell containing the immersed boundary and for the fluid region in this numerical cell as well. From these constraints, one can obtain the amount of mass source/sink \( q \) as follows [3]:

\[ q = \frac{1}{\Delta V} \sum_i \omega_i (u_i - \dot{u}_i) \cdot n \Delta S_i, \]

(7)

where \( \Delta V \) is the cell volume; \( \Delta S_i \) is the area of each cell face \( i \); \( u_i \) is the surface velocity vector; and \( n \) is the unit normal vector outward of each cell face. Here, \( \omega_i \) is defined as one for the cell faces with non-zero \( f_i \) and zero elsewhere. In two dimensions (Fig. 1), the mass source/sink is given as

\[ q = \frac{u_i^k - u_i^{k-1} + v_i^k - v_i^{k-1}}{\Delta x \Delta y}. \]

(8)

More detailed procedures on how to obtain the momentum forcing and mass source/sink are shown in Kim et al. [3]

The \( u_i^k \) and \( v_i^k \) in Eq. (8) are unknown until Eq. (5) is solved, but they should be used to solve Eq. (4). Thus, \( \dot{u}_i \) and \( \dot{v}_i \) are used to determine \( q \) in Eq. (8), that is,

\[ q = \frac{u_i^k - u_i^{k-1} + v_i^k - v_i^{k-1}}{\Delta x \Delta y}. \]

(9)

For this reason, an error is generated from the continuity for the cell containing the immersed boundary:

\[ e = \frac{\dot{u}_i^k - \dot{u}_i^{k-1}}{\Delta x} + \frac{\dot{v}_i^k - \dot{v}_i^{k-1}}{\Delta y}. \]

(10)

In Eqs. (5) and (6), \( u_i^k - \dot{u}_i^k = \partial(\Delta r^i) \) because \( \phi = \partial(\Delta r) \), and thus \( e_i = \partial(\Delta r^i) \). This error does not cause any problem for flows with steady boundary conditions but may produce pressure oscillations and spurious velocities in the vicinity of the immersed boundary when an unsteady boundary condition is given on the immersed boundary. Therefore, this error should be eliminated or significantly reduced.

3. Remedy

As shown in Eq. (10), the error is proportional to the difference between the velocity and intermediate velocity on or near the immersed boundary (i.e., at the grid points where \( f_i \neq 0 \)). This difference is \( -2\alpha_i \Delta t / \partial \phi_i^k / \partial x_i \) as in Eq. (5), and the error becomes zero when \( \partial \phi_i^k / \partial x_i = 0 \) there. Therefore, we apply this Neumann boundary condition on the grid points where the momentum forcing \( f_i \) is applied when we solve the Poisson Eq. (4). An example of applying this boundary condition is schematically drawn in Fig. 2 for flow over a circular cylinder.

4. Numerical examples

First, we consider a model problem shown in Fig. 3, where a jet is issued at the bottom center of the