A simple immersed boundary technique for simulating complex flows with rigid boundary

Shen-Wei Su\textsuperscript{a}, Ming-Chih Lai\textsuperscript{b}, Chao-An Lin\textsuperscript{a,*}

\textsuperscript{a}Department of Power Mechanical Engineering, National Tsing Hua University, Hsinchu 300, TAIWAN
\textsuperscript{b}Department of Applied Mathematics, National Chiao Tung University, Hsinchu 300, TAIWAN

Abstract

A new immersed boundary technique for the simulation of flow interacting with solid boundary is presented. The method employs a mixture of Eulerian and Lagrangian variables, where the solid boundary is represented by discrete Lagrangian markers embedding in the Eulerian fluid domain. A novel force formulation on the Lagrangian markers is proposed to ensure the exact satisfaction of the no-slip condition on the boundary. Interactions between the Lagrangian markers and the fluid variables on the fixed Eulerian grid are linked by a simple discretized delta function. One major advantage of the present formulation is that no complicated interpolation procedure is present to track the complex solid boundary. The stability limit and second order accuracy of the adopted numerical scheme are not altered by the proposed force formulation. Three different test problems are simulated using the present technique (decaying vortex, lid-driven cavity, and flow over a cylinder), and the results are compared with previous experimental and numerical results. The numerical evidences show the accuracy and the capability of the proposed method for solving complex geometry flow problems.

Key words: Immersed boundary method; fluid-solid interaction; Navier-Stokes equations; flow around a cylinder.

* Corresponding author.

Email address: calin@pme.nthu.edu.tw (Chao-An Lin).

Preprint submitted to Elsevier Science 26 March 2004
1 Introduction

The fluid-solid interaction problems are frequently encountered in many engineering applications. When the immersed object is complex and moving, it poses severe challenge on how to solve the problem accurately and efficiently. Traditionally, the body-fitted or unstructured grid methods are adopted to simulate the flow with complex rigid boundary. However, the most serious drawback of these methods is that they require expensive preprocessing such as unstructured grid generation when the boundary is moving. It turns out that this kind of approaches are quite time and memory consuming.

Another simple alternative to tackle the complex fluid-structure problem is to use the Cartesian grid instead of body-fitted grid. The immersed boundary (IB) method proposed by Peskin [1], has been applied successfully to blood-valve interaction and other biological problems. Readers who are interested in IB method and its applications can refer to Peskin’s recent review article [2]. The immersed boundary formulation employs a mixture of Eulerian and Lagrangian variables, where the complex structure is represented by a couple of discrete Lagrangian markers embedding in the Eulerian fluid domain. Those markers can be treated as force generators to the fluid, and meanwhile move along with the fluid. This interaction between the Lagrangian markers and the fluid variables defined on the fixed Eulerian grid is linked by the well-chosen discretized delta function. The resulting flow computation is performed on the Cartesian grid; thus, many fast fluid solvers can be implemented directly.

Although the IB method is developed to handle mostly the fluid problem with elastic structures, it has also been used to simulate the flow with rigid boundaries or structures. In [3], Lai and Peskin proposed a new formally second-order accurate immersed boundary method and applied to simulate the flow past a rigid cylinder. The authors simply used a stiff spring to connect the boundary points to their target positions. Since this feedback force is distributed into the fluid at the beginning of each time step, the problem becomes very stiff which has the consequence of the small time step. The force also generates oscillations along the surface of the boundary.

Here, several techniques which are related to the IB method to tackle the solid boundary are described. The virtual boundary method was proposed by Goldstein et al. [4][5] to simulate the flow with solid boundary within the spectral method framework. Similar to the IB method, the solid boundary is treated as a force generator where the force field is calculated by a feedback way of the difference between the predicted velocity and the actual velocity of the boundary. To avoid the interpolating procedure of the boundary and the grid points, those forces are applied to the grid points directly. Thus, a step-like surface is simulated rather than a smooth surface. Besides, it was shown that
in order to prevent the generation of the spurious oscillations in simulating the
start-up flow around a cylinder, the magnitude of the CFL number has been
kept below $O(10^{-2})$. Later on, Saiki and Biringen [6] further improved the work
of Goldstein et al. and used the area-weighted average function to interpolate
the fluid velocity to the boundary points and distribute the boundary force
back to grid points. A very good agreement has been found between their
numerical computations and experimental results for the simulation of flow
around a cylinder.

Mohd-Yusof [7] proposed a different approach to evaluate the required force
field to impose the no-slip condition at the immersed boundary. The force field
is directly computed from the momentum equations so that the time step can
be larger than the previous methods. This direct momentum forcing is applied
either inside the body or on the surface. Fadlun et al. [8] further extended
the Mohd-Yusof approach to a finite-difference formulation on a stagger gird
system, and showed that the direct forcing suggested by Mohd-Yusof is more
efficient than the feedback forcing of Goldstein et al.

Based on the concept of direct forcing, Kim et al. [9] introduced both the
momentum forcing and mass source/sink to properly represent the immersed
body. The discrete-time momentum forcing and the mass source/sink are ap-
plied only on the body surface or inside the body so that the no-slip boundary
condition on the immersed boundary and the continuity for the cell contain-
ing the boundary are both satisfied. Since the immersed boundary in general
does not coincide with the grid points, an interpolation scheme for computing
the momentum forcing must be employed. In [10], Silva et al. calculated
the momentum forcing at the beginning of each time step. The pressure and
velocity derivatives at those boundary points are interpolated by a second-
order Lagrange polynomial approximation. Once the momentum forcing at
the boundary points has been calculated, they are distributed over the Eule-
rian grid by the discrete delta function. The whole numerical scheme is very
similar to the immersed boundary method of Lai and Peskin except the way of
evaluating the forcing term. While Silva et al.’s approach is ideally simple, the
calculations of momentum forcing at the boundary points are quite tedious.

Recently, Tseng and Ferziger [11] extended the idea of Fadlun et al. via a
ghost cell approach. The immersed boundary is represented by piecewise lin-
ear segments and the ghost cells are defined to lie just inside the body but
adjacent to computational grids of the fluid domain. The values of fluid vari-
ables at those ghost cells are obtained by extrapolation using a local quadratic
scheme which involves the neighboring flow nodes and the associated velocity
boundary condition.

There is another different (from above) Cartesian grid method proposed by
Ye at el. [12] for simulating two-dimensional viscous incompressible flows with
complex geometries. In this method, a cut-cell strategy and interpolation near
the immersed boundary has been applied. More specifically, a control volume
containing the immersed boundary must be reshaped and complex interpolat-
ing procedures to approximate the fluxes must be introduced. The concept of
the momentum forcing is not used here.

It can be summarized that the existing IB techniques suffer from either the
adoption of small CFL number and hence small time step or the complex
interpolating procedure to track the complex boundary. To remedy these de-
ficiencies, a simple technique is presented. The present IB method employs
a mixture of Eulerian and Lagrangian variables, where the solid boundary is
represented by discrete Lagrangian markers embedding in the Eulerian fluid
domain. A novel force formulation on the Lagrangian markers is proposed to
ensure the exact satisfaction of the no-slip condition on the boundary. Inter-
actions between the Lagrangian markers and the fluid variables on the fixed
Eulerian grid are linked by a simple discretized delta function. The proposed
immersed boundary technique has been used to simulate three different two-
dimensional flows; namely, the decaying vortex, lid-driven cavity flow within
a square domain, and the flow past a cylinder to examine the capability of the
proposed method.

2 The methodology of immersed boundary technique

2.1 Mathematical formulation

Throughout this paper, we consider a problem of a viscous incompressible fluid
in a two-dimensional square domain $\Omega = [0, L] \times [0, L]$ containing an immersed
massless boundary in the form of a simple closed curve $\Gamma$, as shown in Figure
1. The immersed boundary is tracked by the parametric form, $X(s), 0 \leq s \leq
L_b$, where $s$ is the parameter of the reference configuration of the boundary.
As mentioned before, the influence of the immersed boundary to the fluid is
represented by some forces that exert to the fluid so that the fluid moves along
with the immersed boundary at the prescribed velocity $u_b(X(s), t)$ (no-slip
condition). Thus, the governing equations of this fluid-structure interaction
system are

$$\frac{\partial u}{\partial t} + \nabla (uu) = -\nabla p + \frac{1}{Re} \nabla^2 u + f,$$

$$\nabla \cdot u = 0.$$
\[ f(x, t) = \int_0^{L_b} F(X(s), t) \delta(x - X(s)) \, ds \]  
(3)

\[ u_b(X(s), t) = \int_\Omega u(x, t) \delta(x - X(s)) \, dx. \]  
(4)

Here, \( x = (x, y) \), \( u(x, t) \) is the fluid velocity, \( p(x, t) \) is the fluid pressure, and the number \( Re \) is the dimensionless Reynolds number.

Equations (1)-(2) are the familiar Navier-Stokes equations of a viscous incompressible fluid. Equations (3)-(4) represent the interaction between the immersed boundary and the fluid. In particular, Eq. (3) describes that the force acting on the fluid is due to the boundary force alone, and Eq. (4) represents the fluid moves with the same prescribed velocity of the immersed boundary. One can easily see that our formulation employs a mixture of Eulerian (\( x \)) and Lagrangian (\( X \)) variables which are linked by the two-dimensional Dirac delta function \( \delta(x) = \delta(x)\delta(y) \). To close the system, the initial and the physical boundary \( \partial\Omega \) conditions of velocity should be given too.

The main difficulty of the above mathematical formulation is that the forcing term \( f \) is not known a priori (since the boundary force \( F \) is unknown). The force field \( F \), however, can be determined by enforcing the no-slip boundary condition of the immersed boundary. Mathematically, it seems reasonable since we have introduced two unknown components of \( F \) defined on the boundary and, at the same time, we have imposed two additional constraints on the boundary (4).

2.2 Numerical scheme

The present numerical method is a mixed Eulerian-Lagrangian finite volume method for simulating the complex fluid flows interacting with an immersed boundary. So we need to introduce two distinct discretized grids: the regular lattice points cover the whole fluid domain, and the marker points discretize the immersed boundary. Throughout this work, the spatial discretization makes use of the staggered marker-and-cell (MAC) mesh introduced by Harlow and Welsh [13]. Thus, the fluid variables are defined at different locations of the lattice grids. For instance, the pressure is defined on the grid points labeled as \( x = (x_i, y_j) = ((i + 1/2)h, (j + 1/2)h) \) for \( i, j = 1, 2 \ldots, N \), where the spacing \( h = \Delta x = \Delta y = L/N \).

In the present study, we integrate the time step using the fractional-step method proposed by Choi and Moin [14], where the nonlinear convection term is treated by the Adams-Bashforth method and the diffusion term is treated by the Crank-Nicolson method. All the spatial derivatives are discretized by
the second order central difference scheme. At the beginning of the time step, the solutions \( u^{n-1}, u^n \) must be given in order to march to \( u^{n+1} \). This is can be done by the following steps:

\[
\frac{\tilde{u} - u^n}{\Delta t} = -\frac{3}{2} \nabla (uu)^n + \frac{1}{2} \nabla (uu)^{n-1} - \nabla p^n + \frac{1}{2Re} \nabla^2 (u^n + \tilde{u}), \tag{5}
\]

\[
\frac{u^* - \tilde{u}}{\Delta t} = f^{n+1/2}, \tag{6}
\]

\[
\frac{u^{**} - u^*}{\Delta t} = \nabla p^n, \tag{7}
\]

\[
\nabla \cdot u^{**} = \nabla^2 p^{n+1}, \tag{8}
\]

\[
\frac{u^{n+1} - u^{**}}{\Delta t} = -\nabla p^{n+1}, \tag{9}
\]

where \( \tilde{u}, u^*, \) and \( u^{**} \) are the intermediate velocity components between the time step \( n \) and \( n+1 \). The continuity is satisfied by solving the pressure Poisson equation (8). The systems of algebraic equations arising from equations (5) and (8) are solved using Bi-CGSTAB method [15]. The remaining question is to find the force field \( f^{n+1/2} \) properly in the step (6) so that the resulting velocity can be adjusted by the presence of the force.

The force density \( f \) (we suppress the superscript for simplicity) in Eq. (6) can be obtained from the discrete form of Eq. (3). To do this, let us first place \( M \) discrete Lagrangian markers \( X_k = (X_k, Y_k) \) along the immersed boundary. Suppose the forces at those marker points \( F(X_k) \) are given, then the force at any Cartesian grid point \( x = (x_i, y_j) \) can be extrapolated by the approximation

\[
f(x) = \sum_{k=1}^{M} F(X_k) \delta_h(x - X_k) \Delta s \tag{10}\]

where the marker mesh \( \Delta s = L_b / M \). Here, the discrete delta function is defined by

\[
\delta_h(x - X_k) = d_h(x_i - X_k) d_h(y_j - Y_k) \tag{11}\]

which \( d_h(r) \) is the hat function,

\[
d_h(r) = \begin{cases} 
(1 - |r|/h)/h, & \text{for } |r| \leq h \\
0, & \text{otherwise.}
\end{cases} \tag{12}\]

Next, we shall describe on how to evaluate the force field \( F(X_k) \).
2.3 The boundary force derivation

Before we proceed, let us introduce the velocity interpolation formula. Since the Lagrangian marker $X_k$ does not in general coincide with the Eulerian grid $x$, the marker velocity can be interpolated by the neighboring Eulerian grid velocity. This can be done via the link of the discrete delta function as

$$\tilde{u}(X_k) = \sum_x \hat{u}(x) \delta_h(x - X_k) h^2. \quad (13)$$

It is important to note that the summation $\sum_x$ is only over four neighboring grid points of $X_k$ since the support of the discrete delta function is as wide as the mesh width $h$. Therefore, the above approximation is nothing but the bi-linear interpolation of the velocity.

If the above interpolation procedure is applied to Eq. (6), the equation at Lagrangian marker $X_k$ takes the following form,

$$\sum_x f(x) \delta_h(x - X_k) h^2 = \frac{u_b(X_k) - \hat{u}(X_k)}{\Delta t}. \quad (14)$$

where $u_b(X_k)$ is the prescribed velocity at the marker $X_k$, and consequently $u_b(X_k) = \mathbf{u}^*(X_k)$.

From the above equation, the resulting $\mathbf{u}^*(X_k)$ will satisfy the known marker velocity distribution at every time step.

By combining Equations (10) and (14), the Lagrangian boundary force can be obtained as

$$\sum_x \sum_{j=1}^M \mathbf{F}(X_j) \delta_h(x - X_j) \Delta s \delta_h(x - X_k) h^2 = \frac{u_b(X_k) - \hat{u}(X_k)}{\Delta t}. \quad (15)$$

Therefore, we have

$$\sum_{j=1}^M \left( \sum_x \delta_h(x - X_j) \delta_h(x - X_k) \Delta s h^2 \right) \mathbf{F}(X_j) = \frac{u_b(X_k) - \hat{u}(X_k)}{\Delta t}. \quad (16)$$

There are $M$ equations for the $M$ Lagrangian marker forces $\mathbf{F}(X_k), k = 1, 2, \ldots, M$. In the present formulation, the markers can move in a prescribed velocity or remain stationary. The coupled algebraic equations can be easily solved using the LU decomposition technique if the Lagrangian markers are
fixed in space. One major advantage of the present formulation is that no com-
plicated interpolation procedure is present apart from the adopted simple hat 
function. Another is that at every time step the pre-defined boundary velocity 
is satisfied.

2.4 The full solution procedure

A whole numerical procedure for each time step of the present method is 
summarized as follows.

(1) Solve Eq. (5) in order to obtain the intermediate velocity component 
\( \tilde{u}(x) \).
(2) Calculate the Lagrangian boundary force \( F(X_k) \) for all markers using 
Eq. (16) and the prescribed marker velocity \( u_b(X_k) \).
(3) Distribute the Lagrangian force \( F(X_k) \) to the Eulerian grid \( f(x) \) using 
Eq. (10).
(4) Correct the intermediate velocity \( u^*(x) \) using the newly obtained force 
\( f(x) \) through Eq. (6).
(5) Obtain the intermediate velocity \( u^{**}(x) \) using Eq. (7).
(6) Compute the pressure by solving pressure Poisson equation (8).
(7) Update the fluid velocity \( u^{n+1}(x) \) by Eq. (9).

3 Numerical Results

3.1 Decaying vortex

In this subsection, we perform the numerical accuracy check for our Navier-
Stokes solvers in the cases of with and without immersed boundary. The test 
example is the decaying vortex problem which is used frequently since the 
analytic solution is available. The solution is listed in the following.

\[
\begin{align*}
  u(x, y, t) &= -\cos(\pi x)\sin(\pi y)e^{-2\pi^2 t/Re}, \\
  v(x, y, t) &= \sin(\pi x)\cos(\pi y)e^{-2\pi^2 t/Re}, \\
  P(x, y, t) &= -\frac{1}{4}[\cos(2\pi x) + \sin(2\pi y)]e^{-4\pi^2 t/Re}.
\end{align*}
\]

In this test, the computational domain is chosen as \([-2, 2] \times [-2, 2]\). In order to 
test the numerical accuracy of our immersed boundary technique, we assume
there exists an immersed boundary in a form of unit circle virtually such that the exact time varying boundary conditions from the analytic solutions are imposed along the boundary of the Lagrangian markers. Since the immersed boundary does not coincide with the Cartesian grids, so the interpolation procedure is required to obtain velocity along the immersed boundary.

Four different uniform grids ($N \times N$, $N = 20, 40, 80, 160$) are used in the simulations with the associated number of Lagrangian markers ($M = 40, 80, 160, 320$), respectively. The grid spacing ratio of the lattice points and marker points is 0.79. The time step size $\Delta t$ is chosen as the half of lattice grid spacing in which CFL number equals to 0.5. The computations are all up to time $t = 0.5$, and the Reynolds number $Re = 1$. It should be noted that the present boundary force formulation does not alter the stability limit ($CFL < 1$) of the present numerical scheme.

Fig. 2 shows the $L_2$ error of the velocity component $u$ for both cases (with and without the immersed boundary) using different grid resolutions. The error for $v$ is similar so we omit here. One can easily see that the second-order accuracy is confirmed for both cases since the slopes of both lines are roughly $-2$. Furthermore, the error of the case without immersed boundary is smaller than the case with immersed boundary. This is not surprising since an extra force-velocity interpolation procedure has been performed in the latter case.

### 3.2 Lid-driven cavity

The lid-driven cavity flow is widely used to verify the accuracy of numerical method too. Fig. 3 shows the geometric layout of the lid-driven cavity within a square domain $[-1, 1] \times [-1, 1]$, where the cavity ($|x| + |y| = 1/\sqrt{2}$) is oblique at $45^\circ$ with respect to the Eulerian grid. Thus, the width of the cavity is 1 and the lid locates along the edge $x + y = 1/\sqrt{2}$ with constant driven velocity one. This inclined lid-driven cavity is mimicked by the present immersed boundary method, where the the respective velocity distributions are assigned at Lagrangian markers. No slip conditions are prescribed along the computational boundary of the Eulerian grids.

Three different uniform grids ($N \times N$, $N = 40, 80, 160$) are used in the simulations with the associated number of Lagrangian markers ($M = 80, 160, 320$), respectively. The grid spacing ratio of the lattice points and marker points is 1. The Reynolds number is chosen as $Re = 100$ and the maximum CFL number is 0.4.

As discussed in Section 2, the key ingredient of our new technique is to compute the forces at Lagrangian markers accurately so that the no-slip boundary conditions can be satisfied exactly at those points. Fig. 4 shows the $L_1$ inter-
polating errors using the proposed new technique (Eq. (16)). One can see that the new technique brings the marker points to their desired velocity within the machine precision.

Fig. 5 shows the steady velocity component $U$ inside the cavity but along the line $y = x$. The coordinate $Y = 0$ represents the intersection of the lower left edge and $y = x$. Fig. 6 shows the steady velocity component $V$ inside the cavity but along the line $y = -x$. The coordinate $X = 0$ represents the intersection of the upper left edge and $y = -x$. We have shown all three numerical results calculated by different grids as well as the numerical result obtained by Ghia et al.[16]. One can see that our numerical results converge to the result obtained by Ghia et al. quite well.

### 3.3 The flow past a cylinder

The flow past a stationary circular cylinder is a typical problem and has been widely investigated experimentally and numerically. For Reynolds number below 47, the flow structure remains symmetric with stationary recirculating vortices behind the cylinder. As the Reynolds number is elevated, the symmetry breaks down and the vortex starts to shed up and down alternatively. This shedding frequency and the intensity of the vortex also increase in tandem with the elevated level of the Reynolds number.

In this test, we shall examine the flow with a broad range of Reynolds number, $Re = 20, 40, 80, 100$ and 150. The geometric set up in the computational domain and the associate physical boundary conditions are shown in Fig. 7. Based on the diameter of the cylinder $D (= 0.2m)$, we choose the range of the computational domain as $(-13.4D \leq x \leq 16.5D, -8.35D \leq y \leq 8.35D)$. A non-uniform grid ($250 \times 160$) is adopted to discretize the computational domain, within which a uniform grid $60 \times 60$ is employed in the region $-D \leq x, y \leq D$. Here, the maximum CFL number employed is 0.37. In the simulation, the boundary of the stationary cylinder is modeled by the present immersed boundary methods, where the Lagrangian markers are assigned the no slip boundary conditions.

There are three different quantities that are often used to make a comparison of the performance of numerical methods; namely, the drag and lift coefficients, and the Strouhal number. The definitions and how we compute these quantities are briefly described below.

The drag coefficient is defined as

$$C_D = \frac{F_D}{U_\infty^2 D/2} \quad (20)$$
where $F_D$ is the drag force. As in [3], the drag force can be obtained easily from

$$ F_D = - \int_{\Omega} f_1(x) \, dx = - \sum_{x} f_1(x) \, h^2 $$

(21)

where $f_1(x)$ is the $x$ component of the Eulerian force $f(x)$.

Similarly, the lift coefficient is defined as

$$ C_L = \frac{F_L}{\frac{1}{2} D U_{\infty}^2} $$

(22)

where $F_L$ is the lift force and it can be obtained by the Eulerian force as

$$ F_L = - \int_{\Omega} f_2(x) \, dx = - \sum_{x} f_2(x) \, h^2 $$

(23)

where $f_2(x)$ is the $y$ component of the Eulerian force $f(x)$.

The other interesting quantity called the Strouhal number ($St$) is the dimensionless frequency of vortex shedding. When the flow field becomes unstable, the originally stationary vortices behind the cylinder start moving downstream and shedding alternatively with frequency $f_q$. This dimensionless vortex shedding frequency is called the Strouhal number, and is defined as $St = f_q D / U_{\infty}$.

Table 1 shows the comparison of drag coefficient with the previous numerical and experimental results for different Reynolds numbers. Table 2 shows the comparison of lift coefficient for $Re = 100$. Table 3 shows the comparison of Strouhal number with the previous numerical and experimental results for different Reynolds numbers. One can see that we obtained the close results with those methods as mentioned in the introduction.

Fig. 8 and Fig. 9 show the evolution of the drag and lift coefficients and the vorticity contour lines for $Re = 40$. It has confirmed the experimental observation that at this Reynolds number, there are two vortices attached behind the cylinder and the lift force is constantly zero. However, if the Reynolds number becomes larger, the symmetry breaks down and the two vortices shed alternatively. Fig. 10 and Fig. 11 show the evolution of the drag and lift coefficients and the vorticity contour lines for $Re = 100$. 
Conclusion

In this paper, we propose a new immersed boundary technique for the simulation of two-dimensional viscous incompressible flow interacting with complex solid boundary. The present IB method employs a mixture of Eulerian and Lagrangian variables, where the solid boundary is represented by discrete Lagrangian markers embedding in the Eulerian fluid domain. A novel force formulation on the Lagrangian markers is proposed to ensure the exact satisfaction of the no-slip condition on the boundary. Interactions between the Lagrangian markers and the fluid variables on the fixed Eulerian grid are linked by a simple discretized delta function. The time integration of the present numerical scheme is based on the fractional-step method and the spatial discretization is on a staggered finite-volume arrangement. One major advantage of the present formulation is that no complicated interpolation procedure is present to track the complex solid boundary apart from the adopted simple hat function. The stability limit and second order accuracy of the scheme are not altered by the proposed force formulation. Three different test problems are simulated using the present technique (decaying vortex, lid-driven cavity, and flow over a cylinder), and the results are compared with previous experimental and numerical results. The numerical evidences show the accuracy and the capability of our method for solving complex geometry flow problems.

References


<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>20</td>
<td>2.20</td>
<td>-</td>
<td>-</td>
<td>2.04</td>
<td>2.03</td>
<td>2.22</td>
</tr>
<tr>
<td>40</td>
<td>1.63</td>
<td>-</td>
<td>1.51</td>
<td>1.54</td>
<td>1.52</td>
<td>1.48</td>
</tr>
<tr>
<td>80</td>
<td>1.43</td>
<td>-</td>
<td>1.40</td>
<td>1.37</td>
<td>1.37</td>
<td>1.29</td>
</tr>
<tr>
<td>100</td>
<td>1.40</td>
<td>1.44</td>
<td>1.33</td>
<td>1.39</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>150</td>
<td>1.39</td>
<td>1.44</td>
<td>1.37</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>

Table 1
The comparison of drag coefficients for different Reynolds numbers.
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>$C_L$</td>
<td>0.34</td>
<td>0.33</td>
<td>0.32</td>
</tr>
</tbody>
</table>

Table 2
The comparison of lift coefficient at $Re=100$.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>80</td>
<td>0.153</td>
<td>-</td>
<td>0.15</td>
<td>0.15</td>
<td>0.15</td>
</tr>
<tr>
<td>100</td>
<td>0.168</td>
<td>0.165</td>
<td>0.16</td>
<td>-</td>
<td>0.166</td>
</tr>
<tr>
<td>150</td>
<td>0.187</td>
<td>0.184</td>
<td>0.18</td>
<td>-</td>
<td>0.183</td>
</tr>
</tbody>
</table>

Table 3
The comparison of Strouhal number for different Reynolds numbers.

Fig. 1. Flow domain ($\Omega$) with an immersed boundary ($\Gamma$).
Fig. 2. The $L_2$ errors for the decaying vortex problem with and without immersed boundary.

Fig. 3. The configuration of the lid-driven cavity in the computational domain.
Fig. 4. The $L_1$ errors of the interpolating velocity along the immersed boundary (cavity edges) using the proposed force formulation (equation 16). The result is obtained by using $160 \times 160$ grid.

Fig. 5. The steady velocity component $U$ along the line ($y = x$) through the center of cavity with different Eulerian grid sizes.
Fig. 6. The velocity component $V$ along the central line ($y = -x$) through the center of cavity with different Eulerian grid sizes.

\[
\begin{align*}
\frac{\partial p}{\partial x} &= 0 \\
\frac{\partial u}{\partial x} &= 0 \\
\frac{\partial p}{\partial y} &= 0 \\
\frac{\partial u}{\partial y} &= 0, v = 0 \\
\frac{\partial v}{\partial x} &= 0 \\
\frac{\partial p}{\partial x} &= 0 \\
\frac{\partial p}{\partial y} &= 0 \\
\frac{\partial u}{\partial y} &= 0, v = 0
\end{align*}
\]

Fig. 7. The boundary condition and computational domain of flow over cylinder
Fig. 8. The time evolution of drag and lift coefficients of $Re = 40$.

Fig. 9. Instantaneous vorticity contours near the cylinder of $Re = 40$. 
Fig. 10. The time evolution of drag and lift coefficients of $Re = 100$.

Fig. 11. Instantaneous vorticity contours near the cylinder of $Re = 100$