Numerical simulations of heat transfer and fluid flow problems using an immersed-boundary finite-volume method on non-staggered grids*

J. R. Pacheco¹, A. Pacheco–Vega², T. Rodić¹ and R. E. Peck¹

¹Mechanical and Aerospace Department
Arizona State University
Tempe, AZ 85287 USA

²CIEP-Facultad de Ciencias Químicas
Universidad Autónoma de San Luis Potosí
San Luis Potosí, SLP 78210, México

Printed December 24, 2004

*Address all correspondence to J.R.Pacheco: ASU P.O. Box 6106, Tempe AZ 85287-6106
¹E-mail:rpacheco@asu.edu
²E-mail:apacheco@uaslp.mx. Assistant Professor
³E-mail:rtamara@asu.edu
⁴E-mail:rep@asu.edu. Professor
ABSTRACT

This paper describes the application of the immersed boundary technique for simulating fluid flow and heat transfer problems over or inside complex geometries. The methodology is based on a fractional step method to integrate in time. The governing equations are discretized and solved on a regular mesh with a finite volume non-staggered grid technique. Implementations of Dirichlet and Neumann type of boundary conditions are developed and completely validated. Several phenomenologically different fluid flow and heat transfer problems are simulated using the technique considered in this study. The accuracy of the method is second-order, and the efficiency is verified by favorable comparison with previous results from numerical simulations and laboratory experiments.

NOMENCLATURE

$Bi$  
Biot number

c  
Space-averaged velocity

$C$  
Convective operator for temperature

$C_D$  
Drag coefficient

$Ci$  
Convective operator for velocity

d  
Diameter of cylinder

$D_I$  
Diffusive operator for velocity

$D_T$  
Diffusive operator for temperature

$e_i$  
Unit vector component

$f$  
Shedding frequency

$f_i$  
Momentum forcing

$F_D$  
Drag force

$F_i$  
Total force

$g,g$  
Gravitational acceleration

$G_{mn}$  
Mesh skewness tensor
\begin{tabular}{ll}
Gr & Grashof number \\
h & Energy forcing \\
h_f & Coefficient of thermal convection \\
J^{-1} & Jacobian \\
k_s & Thermal conductivity of the solid \\
L & Cavity length \\
L_c & Characteristic length \\
n & Unit vector normal to the surface \\
N & Number of mesh points \\
Nu & Nusselt number \\
p & Pressure \\
ṗ & Dimensional pressure \\
p_\infty & Reference pressure \\
P_1, P_2, P_3 & Nondimensional parameters \\
Pr & Prandtl number \\
Ra & Rayleigh number = GrPr \\
Re & Reynolds number \\
R_i & Gradient operator \\
S & Surface variable \\
St & Strouhal number \\
t & Normalized time \\
ṫ & Dimensional time \\
T & Temperature \\
T_0 & Reference temperature \\
u_i & Normalized Cartesian velocity at the center of the cell \\
\hat{u}_i & Dimensional Cartesian velocity at the center of the cell \\
u_i^* & Intermediate Cartesian velocity \\
\end{tabular}
\( \ddot{u}_i \)  Intermediate forcing velocity

\( U \)  Reference velocity

\( U_m \)  Volume flux

\( \bar{U}_m \)  Velocity at the immersed body

\( V \)  Volume variable

\( x_c \)  Separation length of the cylinder

\( x_i \)  Normalized Cartesian coordinates

\( \hat{x}_i \)  Dimensional Cartesian coordinates

\( x_s \)  Separation length of the sphere

\( X_m \)  Coordinates of the immersed boundary point

\textit{Greek symbols}

\( \alpha \)  Thermal diffusivity, interpolation factor

\( \beta \)  Coefficient of thermal expansion, interpolation factor

\( \gamma \)  Rotation angle

\( \delta \)  Discretization

\( \delta_{ij} \)  Kronecker delta

\( \Delta \)  Increment

\( \epsilon \)  Eccentricity

\( \epsilon_{u_1} \)  Relative error

\( \mu \)  Dynamic viscosity

\( \nabla \)  Nabla operator

\( \nu \)  Kinematic viscosity

\( \rho \)  Density

\( \Theta \)  Normalized temperature

\( \bar{\Theta} \)  Temperature in the energy forcing

\( \ddot{\Theta} \)  Intermediate temperature in energy forcing
$\varphi$ Inclination angle

$\xi_m$ Curvilinear coordinates

**Subscripts and superscripts**

$c$ Characteristic

$f$ Fluid

$i,j$ Indices for the Cartesian coordinates

$m$ Index for the immersed location

$n$ Index of time step

$m,n$ Indices for the curvilinear coordinates

$w$ Wall or immersed body surface

$+, -$ Forward, backward increments
INTRODUCTION

One of the main streams in the analysis and design of engineering equipment has been the application of computational fluid dynamics (CFD) methodologies. Despite the fact that experiments are important to study particular cases, numerical simulations of mathematical models allow more general analyses. While simple geometries can be handled by most CFD algorithms, the majority of the engineering devices have complex geometries, making their numerical analysis a difficult task, since the discretization of the governing equations of geometrically complex flows is still one of the main challenges in CFD.

Nowadays there are three main approaches for the simulation of complex geometry flows: the boundary-fitted curvilinear method, the unstructured mesh technique, and the simulation of immersed boundaries on Cartesian grids. In the latter methodology, a body-force is introduced in the momentum equations discretized on orthogonal Cartesian grids, such that a desired velocity can be obtained on an imaginary boundary. One of the main advantages of this approach, unlike the unstructured-mesh and the boundary-fitted methodologies, is that grid generation in time is not needed, e.g. in the study of free surface flows and phase change problems.

Depending on the way the boundary conditions are enforced on the surface of the immersed body, the methodologies implemented can be sub-categorized as: (a) immersed boundary techniques [1, 2, 3, 4]; (b) cut-cell methods [5, 6, 7, 8]; (c) hybrid Cartesian/immersed boundary formulations [9, 10, 11]; and (d) the novel immersed continuum method [12, 13].

The immersed boundary method has been widely applied by Peskin and co-workers [1, 2, 3, 4] to analyze the dynamics of blood in heart valves, where the interaction between the fluid and immersed boundary was modeled by a discretized approximation to the Dirac delta function. Goldstein et al. [14] also used this technique, coupled with spectral methods, to study the transient flow around a circular cylinder, and called it virtual boundary method. The main drawback of their virtual boundary method is that it contains two free constants that need to be tuned according to the flow.

In the case of the cut-cell or Cartesian-grid technique, most of the applications have been done
by Udaykumar and collaborators [5, 6, 7, 8] using non-staggered grid layouts. In their method, the concept of momentum forcing is not used, but a control volume near the body is re-shaped into a body-fitted trapezoid adding neighboring cells to account for the immerse boundary. A main drawback of this method consists of the use of the body-fitted trapezoid that introduces a stencil different from that of a regular cell, and thus, an iteration technique is applied to solve the governing equations at each time step.

The hybrid Cartesian/immersed boundary formulation introduced by Mohd-Yusof [15] and Fadlun et al. [11] is very attractive because it is a non-boundary conforming formulation based on a direct forcing, where no free constants need to be determined. Its accuracy is second-order, and the method can be applied on sharp solid interfaces. Kim et al. [9] developed a new immerse boundary method that introduced both the momentum forcing and a mass source/sink to represent the immerse body. Their algorithm, based on a finite-volume approach on a staggered grid, uses a bilinear interpolation procedure that is reduced to one-dimensional linear scheme when there are no points available in the vicinity of the boundary. A linear interpolation along the normal to the body was developed by Balaras [16] but it is restricted to two-dimensional or axisymmetric shapes. Gilmanov [10] further developed the reconstruction algorithm of Balaras [16] using an unstructured triangular mesh to discretize an arbitrary three-dimensional solid surface, where a linear interpolation near the interface is applied on the line normal to the body.

In the immersed continuum technique [12, 13], which is based on finite element methods (FEM) for both solid and fluid subdomains, the continuity at the solid-fluid boundaries is enforced through both interpolation of the velocities and distribution of the nodal forces by means of high order Dirac delta functions. A main advantage of this approach is that the modeling of immersed flexible solids with possible large deformations is feasible.

Although immersed boundary methods have been widely applied to fluid flow problems, very few attempts have been made to solve the passive-scalar transport equation, i.e. the energy equation. Examples include those of Fadlun et al. [11] analyzed a vortex-ring formation solving the passive scalar equation, and Francois and Shyy [17, 18] to calculate liquid drop dynamics with emphasis in
the analysis of drop impact and heat transfer to a substrate during droplet deformation.

In a recent paper, Kim and Choi [19] suggested an two-dimensional immersed-boundary method applied to external-flow heat transfer problems. Their method is based on a finite-volume discretization on a staggered-grid mesh. They implemented the isothermal (Dirichlet) boundary conditions using second-order linear and bilinear interpolations as described in Kim et al. [9]. In the case of the so called iso-flux (Neumann) boundary condition, the interpolation procedure was different from that of the Dirichlet condition, owing to the requirement of the temperature derivative along the normal to the wall. The interpolation scheme used to determine the temperature along the wall-normal line is similar to that of [16] and it is also restricted to two-dimensional or axisymmetric shapes. Their results were compared to those available in the literature for isothermal conditions only. Therefore, the validity of their interpolation scheme for the iso-flux condition remains to be assessed.

Our objective is to describe an immersed-boundary-based algorithm on non-staggered grids capable of solving fluid flow and heat transfer processes under different flow conditions, where either Dirichlet or Neumann boundary conditions could be implemented on two- or three-dimensional bodies. To this end, a general formulation of the governing equations for the problems at hand are presented first, followed by the details of the interpolation schemes and implementation of the boundary conditions. Finally, several geometrically complex fluid flow and heat transfer problems (on cylinders and in cavities), are considered to demonstrate the robustness of the technique.

1 GOVERNING EQUATIONS

The present work concentrates on unbounded forced-convection fluid flow problems around square shapes and cylinders and on natural-convection heat transfer inside cavities. The common denominator of these problems is the complexity of the fluid flow and heat transfer due to the immersed body. A non-dimensional version of the governing equations for an unsteady incompressible Newtonian fluid flow with constant properties, in the Boussinesq limit for the buoyancy force, can be
written as
\[ \frac{\partial u_i}{\partial x_i} = 0, \]  
\[ \frac{\partial u_i}{\partial t} + \frac{\partial}{\partial x_j} (u_j u_i) = -\frac{\partial p}{\partial x_i} + P_1 \left( \frac{\partial^2 u_i}{\partial x_j \partial x_j} \right) + f_i + P_2 \Theta \mathbf{e}_i, \]  
\[ \frac{\partial \Theta}{\partial t} + \frac{\partial}{\partial x_j} (u_j \Theta) = P_3 \left( \frac{\partial^2 \Theta}{\partial x_j \partial x_j} \right) + h, \]  
where \( i, j = 1, 2; \) \( u_i \) represents the Cartesian velocity components, \( p \) is the pressure, \( \mathbf{e}_i \) and \( f_i \) represent the \( i \)-th unit vector component and the momentum forcing components respectively, \( \Theta \) is the temperature of the fluid and \( h \) is the energy forcing. Note that viscous dissipation has been neglected.

According to the problem at hand, different normalizations for the non-dimensional variables in Eqs. (1)–(3) are possible. For instance, for forced and mixed convection, the variables can be normalized as
\[ x_i = \hat{x}_i ; \quad u_i = \hat{u}_i ; \quad t = \hat{t} ; \quad p = \frac{\hat{p}}{\rho U^2} ; \quad \Theta = \frac{T - T_0}{T_w - T_0}, \]
\[ Re = \frac{UL_c}{\nu} ; \quad Pr = \frac{\nu}{\alpha} ; \quad Gr = \frac{g\beta L_c}{U^2} \left( T_w - T_0 \right) \left( \frac{UL_c}{\nu} \right)^2, \]
where the ‘hats’ denote dimensional quantities, \( Re \) is the Reynolds number, \( Pr \) is the Prandtl number, \( Gr \) is the Grashof number, and \( P_1 = 1/Re, P_2 = Gr/Re^2 \) and \( P_3 = 1/RePr \). For natural convection, the normalization is given by
\[ x_i = \frac{\hat{x}_i}{L_c} ; \quad u_i = \frac{\hat{u}_i}{\alpha/L_c} ; \quad t = \frac{\hat{t}}{L_c^2/\alpha} ; \quad p = \frac{\hat{p}}{\rho c^2/\alpha} ; \quad \Theta = \frac{T - T_0}{T_w - T_0}, \]
\[ Pr = \frac{v}{\alpha} ; \quad Gr = \frac{g\beta L_c^3}{v^2} \left( T_w - T_0 \right), \quad Ra = \frac{g\beta L_c^3}{\nu \alpha} \left( T_w - T_0 \right), \]
where \( Ra \) is the Rayleigh number, and \( P_1 = Pr, P_2 = RaPr \) and \( P_3 = 1 \). In the above formulations, \( T_0 \) is a reference temperature, \( T_w \) is the temperature of either a wall or the immersed body, \( L_c \) is a characteristic length, e.g. the height of a cavity or the diameter of a cylinder, \( U \) is a characteristic velocity, \( \nu \) is the kinematic viscosity, \( \beta \) is the coefficient of thermal expansion, and \( \alpha \) is the thermal diffusivity.

In order to have a better resolution in regions where the immersed body is present, as well as in others where potential singularities may arise, e.g. sharp corners, we use a non-uniform mesh by...
means of a body-fitted-like grid mapping. Thus, Eqs. (1)–(3) with the appropriate normalization

\[ \frac{\partial u_i}{\partial t} + \frac{\partial (U_m u_i)}{\partial \xi_m} = - \frac{\partial}{\partial \xi_m} \left( J^{-1} \frac{\partial \xi_m}{\partial x_i} p \right) + \frac{\partial}{\partial \xi_m} \left( P_l G_{mn} \frac{\partial u_i}{\partial \xi_n} \right) \]

\[ + J^{-1} \left( f_i + P_2 \Theta e_i \right), \]

\[ \frac{\partial \left( J^{-1} \Theta \right)}{\partial t} + \frac{\partial (U_m \Theta)}{\partial \xi_m} = \frac{\partial}{\partial \xi_m} \left( P_3 G_{mn} \frac{\partial \Theta}{\partial \xi_n} \right) + J^{-1} h, \]

where \( J^{-1} \) is the inverse of the Jacobian or the volume of the cell, \( U_m \) is the volume flux (contravariant velocity multiplied by \( J^{-1} \)) normal to the surface of constant \( \xi_m \), and \( G_{mn} \) is the ‘mesh skewness tensor’. These quantities are defined by:

\[ U_m = J^{-1} \frac{\partial \xi_m}{\partial x_j} u_j, \quad J^{-1} = \det \left( \frac{\partial x_i}{\partial \xi_j} \right), \quad G_{mn} = J^{-1} \frac{\partial \xi_m}{\partial x_j} \frac{\partial \xi_n}{\partial x_j}. \]

A non-staggered-grid layout [20] is employed in this analysis. We use a semi-implicit time-advancement scheme with the Adams-Bashforth method for the explicit terms and the Crank-Nicholson method for the implicit terms. The discretized equations corresponding to Eqs. (4)–(6) are

\[ \frac{\delta U_m}{\delta \xi_m} = 0, \]

\[ J^{-1} \frac{u_i^{n+1} - u_i^n}{\Delta t} = \frac{1}{2} \left( 3C_i^n - C_i^{n-1} + D_I(u_i^{n+1} + u_i^n) \right) + R_i(p^{n+1}) \]

\[ + J^{-1} \left( f_i^{n+1/2} + P_2 \Theta^{n+1} e_i \right), \]

\[ J^{-1} \frac{\Theta^{n+1} - \Theta^n}{\Delta t} = \frac{1}{2} \left( 3C^n - C^{n-1} + D_I(\Theta^{n+1} + \Theta^n) \right) + J^{-1} h^{n+1/2}, \]

where \( \delta / \delta \xi_m \) represents discrete finite-difference operators in the computational space, superscripts represent the time steps, \( C_i \) and \( C \) represent the convective terms for velocity and temperature, \( R_i \) is the discrete operator for the pressure gradient terms, and \( D_I \) and \( D_I \) are the discrete operators representing the implicitly treated diagonal viscous terms. The equation for each term follows:

\[ C_i = - \frac{\delta}{\delta \xi_m} (U_m u_i); \quad C = - \frac{\delta}{\delta \xi_m} (U_m \Theta); \quad R_i = - \frac{\delta}{\delta \xi_m} \left( J^{-1} \frac{\delta \xi_m}{\delta x_i} \right); \]

\[ D_I = \frac{\delta}{\delta \xi_m} \left( P_l G_{mn} \frac{\delta}{\delta \xi}_n \right), \quad m = n; \quad D_I = \frac{\delta}{\delta \xi_m} \left( P_3 G_{mn} \frac{\delta}{\delta \xi}_n \right), \quad m = n. \]
The diagonal viscous terms are treated implicitly in order to remove the viscous stability limit. The spatial derivatives are discretized using second-order central differences with the exception of the convective terms which are discretized using a variation of QUICK which calculates the face value from the nodal value with a quadratic upwind interpolation on non-uniform meshes [21]. We use a fractional step method to solve Eqs. (7)–(9) as described in [20, 22, 23, 24]. The fractional step method splits the momentum equation in two parts by separating the pressure gradient terms. A solution procedure is formulated such that: (1) a predicted velocity field, which is not constrained by continuity, is computed, (2) the pressure field is calculated, and (3) the correct velocity field is obtained by correcting the predicted velocity with pressure to satisfy continuity. The former step is a projection of the predicted velocity field onto a subspace in which the continuity equation is satisfied.

The momentum forcing $f_i$ and energy forcing $h$ are direct in the context of the approach first introduced by [15] and similar to [9]. The value of the forcing depends on the velocity of the fluid and on the location, temperature and velocity of the immersed boundary. Since the location of the boundary $X_i$ is not always coincident with the grid the forcing values must be interpolated to these nodes. The forcing is zero inside the fluid and non-zero on the body surface or inside the body.

1.1 Energy forcing and momentum forcing

The basic idea consists of determining the forcing on or inside the body such that the boundary conditions on the body are satisfied. The boundary conditions could be either Dirichlet, Neumann or a combination of the two. In the context of energy forcing, if the grid point coincides with the body, then we simply substitute in Eq. (9) the temperature value at the boundary point, $\Theta$, and solve for the forcing as

$$h^{n+1/2} = \left( I - \frac{\Delta t}{2J^{-1}} D_I \right) \frac{\Theta - \Theta^n}{\Delta t} - \frac{1}{J^{-1}} \left[ \frac{3}{2} C^n - \frac{1}{2} C^{n-1} + D_I(\Theta^n) \right].$$

(10)

After substituting Eq. (10) into Eq. (9) we can see that the boundary condition $\Theta^{n+1} = \Theta$ is satisfied. Frequently, however, the boundary point $X_i$ does not coincide with the grid and an interpolation from neighboring points must be used to obtain $\Theta$ inside the body. In this context $\Theta$
is determined by interpolation from temperature values $\Theta$ at grid points located nearby the forcing point. The temperature $\tilde{\Theta}$ is obtained first from the known values at time level $n$ and 

$$J^{-1}\tilde{\Theta} - \Theta^n = \frac{1}{2} \left( 3C^n - C^{n-1} + D_I(\tilde{\Theta} + \Theta^n) \right) + J^{-1}h,$$

(11) with $h = 0$. Once $\tilde{\Theta}$ is known, $\Theta$ is determined next by a linear or bilinear interpolation (see below), and then substituted back into Eq. (10) to find $h^{n+1/2}$. The energy forcing, $h^{n+1/2}$, is now known and can be used in Eq. (9) to advance to the next time level $n + 1$.

The method to determine the momentum forcing $f_i$ is described in detail in Kim et al. [9]. Suffice to say that the method enforces the proper boundary conditions on an intermediate velocity $u_\ast$ that is not restricted by the divergence free condition without compromising the temporal accuracy of the method. The forcing function $f_i$ that will yield the proper boundary condition on the surface of the immersed body can be expressed as:

$$f_{i}^{n+1/2} = \left( 1 - \frac{\Delta t}{2J^{-1}}D_I \right) \bar{U}_i - u_i^n - \frac{1}{J^{-1}} \left[ \frac{3}{2}C_i^n - \frac{1}{2}C_i^{n-1} + D_I(u_i^n) \right] + P_2 \Theta^{n+1} e_i,$$

(12) where $\bar{U}_i$ is the boundary condition on the body surface or inside the body with $f_i = 0$ inside the fluid.

2 TREATMENT OF THE IMMERSED BOUNDARY

In order to illustrate the algorithm for obtaining the nodal values inside the immersed body we treat first the bilinear interpolation scheme shown in Figure 1, for the temperature $\Theta$. Taking the temperatures at the four nodal values of the cell, the temperature $\Theta(X_1, X_2)$ is given by

$$\Theta(X_1, X_2) = \Theta_{i,j}[(1 - \alpha)(1 - \beta)] + \Theta_{i,j+1}[(1 - \alpha)\beta] + \Theta_{i+1,j}[\alpha(1 - \beta)] + \Theta_{i+1,j+1}[\alpha\beta],$$

where the interpolation factors $\alpha$ and $\beta$ have the form:

$$\alpha = \frac{X_1 - x_{1[i,j]}}{x_{1[i+1,j]} - x_{1[i,j]}}, \quad \beta = \frac{X_2 - x_{2[i,j]}}{x_{2[i,j+1]} - x_{2[i,j]}}.$$

In the above, $(X_1, X_2)$ is the coordinate of the point where we want to satisfy the boundary condition. Note that this scheme is also valid for other variables such as the velocity components $u_i$ and pressure $p$, since they are defined at the same location on the cell. This scheme may be easily extended to a trilinear interpolation whenever three-dimensional problems are studied.
2.1 Dirichlet boundary condition

Examples of different nodes classified according to the location inside the body, labeled as (a), (b) and (c), are shown in Figure 2. The unit vectors defining the tangent plane to the surface are \( \mathbf{n}_a \), \( \mathbf{n}_b \) and \( \mathbf{n}_c \). With reference to Figure 2, if (a) is the point where we want to satisfy the boundary condition \( \Theta_a \), then \( \Theta_{1a} \) is uniquely determined from a bilinear interpolation using the adjacent nodes external to the body \( \Theta_{2a}, \Theta_{3a}, \Theta_{4a}, \) and \( \Theta_a \).

If we consider now the adjacent node, (b), we can see that there are two temperature components outside the boundary (\( \Theta_{3b} \) and \( \Theta_{4b} \)) and two inside the boundary (\( \Theta_{1b} \) and \( \Theta_{2b} \)). This case was treated by Kim et al. [9] using a linear interpolation to obtain \( \Theta_{1b} \) from \( \Theta_{4b} \) and the condition at node (b), that is the cross sectional point between the immersed boundary and the line connecting \( \Theta_{1b} \) and \( \Theta_{4b} \). In our algorithm we use a bilinear interpolation which considers \( \Theta_{2b} \) a known quantity. The above is true since \( \Theta_{2b} = \Theta_{1a} \), with the condition that \( \Theta_{1a} \) is previously obtained using the proper bilinear interpolation as explained earlier. Clearly, \( \Theta_{1b} \) could have been obtained using the condition imposed from \( \Theta_b \) (or from \( \Theta_c \)) and all the values surrounding this point. Therefore we use an iterative scheme to obtain the nodal value in the vicinity of the body, except when the bilinear interpolation is proper, i.e. when all the surrounding values except one exist outside the body as in case (a). To be definitive, \( \Theta_{1b} \) must equal \( \Theta_{2c} \), so they must converge to the same value. The initial guess for the nodal values inside the body is obtained from a linear interpolation using the scheme of Kim et al. [9]. Afterwards we iterate using a bilinear interpolation along the normal to the surface boundary until the difference between nodal values is negligible, e.g. \( \Theta_{1b} - \Theta_{2c} \approx 0 \).

2.2 Neumann boundary condition

If we consider a Neumann boundary condition on the immersed body, different interpolation schemes may be used to obtain \( \Theta \), depending on the number of known variables surrounding the point where the condition is being applied. In this study, we use bilinear-linear or linear-linear interpolation schemes, as illustrated in Figure 3. In Figure 3(a), the cell is subdivided in two sub-
cells as two nodal values surrounding the point \((p,j)\) for each sub-cell are known. Thence, \(\Theta = \Theta_{i,j}\) may be obtained using the following procedure:

1. Write the discretized form of the Neumann boundary condition as

\[
\nabla \Theta \cdot \mathbf{n} = \left[ \frac{\Theta_{i+1,j} - \Theta_{i,j}}{\delta x_1^+} \right] n_1 + \left[ \frac{\Theta_{p,j+1} - \Theta_{p,j}}{2\delta x_2^+} + \frac{\Theta_{p,j} - \Theta_{p,j-1}}{2\delta x_2^-} \right] n_2 = 0. 
\]

(13)

2. Express \(\Theta_{p,j}, \Theta_{p,j-1},\) and \(\Theta_{p,j+1}\) using a linear interpolation scheme as

\[
\begin{align*}
\Theta_{p,j} & = \alpha \tilde{\Theta}_{i+1,j} + (1 - \alpha) \tilde{\Theta}_{i,j}, \\
\Theta_{p,j-1} & = \alpha \tilde{\Theta}_{i+1,j-1} + (1 - \alpha) \tilde{\Theta}_{i,j-1}, \\
\Theta_{p,j+1} & = \alpha \tilde{\Theta}_{i+1,j+1} + (1 - \alpha) \tilde{\Theta}_{i,j+1}.
\end{align*}
\]

(14)

3. Combine Eq. (13) and Eq. (14) to obtain \(\tilde{\Theta}_{i,j}\) inside the body, i.e.

\[
\tilde{\Theta}_{i,j} = \left( \frac{n_2}{2} \left[ \frac{\Theta_{p,j+1} - \Theta_{p,j-1}}{\delta x_2^+} \right] + \tilde{\Theta}_{i+1,j} \left[ \frac{\alpha n_2}{2} \left( \frac{\delta x_2^+}{\delta x_2} \right) + \frac{n_1}{\delta x_1^-} \right] \right) \left[ \frac{n_1}{\delta x_1^-} - \frac{(1 - \alpha)n_2}{2} \left( \frac{\delta x_2^-}{\delta x_2} \right) \right].
\]

(15)

On the other hand, if three nodal values surrounding the point \((p,q)\) are known, as shown in Figure 3(b), an expression for \(\overline{\Theta} = \overline{\Theta}_{i,j}\) can be derived following a similar procedure as before. The discretized form of the Neumann boundary can be written as

\[
\nabla \overline{\Theta} \cdot \mathbf{n} = \left[ \frac{\overline{\Theta}_{i+1,q} - \overline{\Theta}_{i,q}}{\delta x_1^+} \right] n_1 + \left[ \frac{\overline{\Theta}_{p,j+1} - \overline{\Theta}_{p,j}}{\delta x_2^-} \right] n_2 = 0.
\]

(16)

We apply a linear interpolation scheme on \(\overline{\Theta}_{p,j}, \overline{\Theta}_{p,j+1}, \overline{\Theta}_{i,q}\) and \(\overline{\Theta}_{i+1,q}\), and a bilinear interpolation scheme on \(\overline{\Theta}_{p,q}\); that is

\[
\begin{align*}
\overline{\Theta}_{p,j} & = \alpha \overline{\Theta}_{i+1,j} + (1 - \alpha) \overline{\Theta}_{i,j}, \\
\overline{\Theta}_{p,j+1} & = \alpha \overline{\Theta}_{i+1,j+1} + (1 - \alpha) \overline{\Theta}_{i,j+1}, \\
\overline{\Theta}_{i,q} & = \beta \overline{\Theta}_{i,j+1} + (1 - \beta) \overline{\Theta}_{i,j}, \\
\overline{\Theta}_{i+1,q} & = \beta \overline{\Theta}_{i+1,j+1} + (1 - \beta) \overline{\Theta}_{i+1,j}, \\
\overline{\Theta}_{p,q} & = \alpha \beta \overline{\Theta}_{i+1,j+1} + \alpha(1 - \beta) \overline{\Theta}_{i+1,j} + (1 - \alpha)\beta \overline{\Theta}_{i,j+1} + (1 - \alpha)(1 - \beta) \overline{\Theta}_{i,j}.
\end{align*}
\]

(17)
The value for $\tilde{\Theta}_{i,j}$ can now be derived by combining Eq. (16) and Eq. (17) so that

$$
\tilde{\Theta}_{i,j} = \frac{\tilde{\Theta}_{i+1,q} - \beta \tilde{\Theta}_{i,j+1}}{\delta x_1^+} n_1 + \frac{\tilde{\Theta}_{p,j+1} - \alpha \tilde{\Theta}_{i+1,j}}{\delta x_2^+} n_2 \\
\left[ (1 - \beta) \frac{n_1}{\delta x_1^+} + (1 - \alpha) \frac{n_2}{\delta x_2^+} \right].
$$

The implementation of the boundary conditions, as described above, has been tested on several other three-dimensional problems. Computations of forced and natural convection heat transfer on spheres are in good agreement with published experimental results, and will be reported elsewhere.

### 3 FLUID FLOW AND HEAT TRANSFER SIMULATIONS

In order to test the proposed immersed-body formulation, we carry out simulations of different fluid flow and heat transfer problems. The first corresponds to the decay of vortices, which was selected to determine the order of accuracy of the method. The second involves an external flow in two dimensions, i.e. a circular cylinder placed in an unbounded uniform flow. The two remaining cases are buoyancy-driven flows, selected to assess the correct implementation of the Dirichlet and Neumann boundary conditions in two-dimensional domains.

#### 3.1 Decaying vortices

This test case is the classical two-dimensional unsteady decaying vortices problem, which has the following exact solution:

$$
\begin{align*}
    u_1(x_1, x_2, t) &= -\cos(x_1) \sin(x_2)e^{-2t}, \\
    u_2(x_1, x_2, t) &= \sin(x_1) \cos(x_2)e^{-2t}, \\
    p(x_1, x_2, t) &= -\frac{1}{4}[\cos(2x_1) \sin(2x_2)]e^{-4t}.
\end{align*}
$$

The governing equations for this problem are Eqs. (1)–(2) with $P_1 = 1/Re$ and a negligible buoyancy force. The computations are carried out in the domain $-1.5 \leq x_1, x_2 \leq 1.5$, with the immersed-boundary lines being at $|x_1| + |x_2| = 1$. Here, the momentum forcing is applied at the points close to the outside immersed boundary as we are interested in the solution inside (see Figure 4). The exact solution is imposed at the boundaries. We use a uniform grid with the refinement of the time
step being proportional to the grid spacing. The maximum relative error for the $u_1$ velocity at the dimensionless time of 0.5, is plotted in Figure 5 as a function of the mesh refinement corresponding to the number of grid points inside the boundary. The figure indicates that the present method is indeed second-order accurate in space and time. The prediction of this unsteady flow illustrates the time-accurate capability of the formulation.

### 3.2 Cylinder placed in an unbounded uniform flow

We analyze the flow around a circular cylinder immersed in an uniform unbounded laminar flow for different Reynolds numbers $Re = UL_c/\nu$. Here the characteristic length is the diameter of the cylinder $L_c = d$, and the characteristic velocity is the freestream velocity $U$. The governing equations for this problem are the same as those used in the previous example. The simulations were performed on a domain size of $30L_c \times 30L_c$. 220 $\times$ 220 grid points in the streamwise and transverse directions were used. The drag coefficient was computed using $C_D = F_D/\frac{1}{2}\rho U^2 L_c$, and the Strouhal number using $St = f d/U$, with $f$ being the shedding frequency. For the present case, we used a Dirichlet condition ($u_1 = 1, u_2 = 0$) at the inflow boundary, a convective condition ($\partial u_i/\partial t + c\partial u_i/\partial x_1 = 0$, where $c$ is the space-averaged velocity at the exit) at the outflow boundary, and $\partial u_i/\partial x_2 = 0$ at the far-field boundaries. To verify the accuracy of the results, the drag coefficient $C_D$ was calculated using two different procedures: (1) a direct integration of the stresses around the cylinder, and (2) an integration along a square domain placed around the cylinder as described by [3], where the total force on the cylinder is given by

$$F_i = -\frac{d}{dt} \int_V \rho u_i dV - \int_S \left( \rho u_i u_j + p \delta_{ij} - \mu \left[ \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right] \right) n_j dS. \tag{22}$$

In the above, $\mu$ is the dynamic viscosity and $V$ and $S$ are the control volume and control surface per unit width around the cylinder. It is to be noted that the maximum differences in the values of $C_D$ from both approaches were confined to less than 0.1%. The results from the immersed-boundary technique suggested here for $Re = 20, 40, 80$ and 100 are compared in Table 1 with those of [3, 5, 9, 25, 26, 27, 28, 29]. The drag coefficient $C_D$ and Strouhal number $St$ are time-averaged values for $Re = 80$ and 100. It can be noted that the present results compare quantitatively well
with other numerical and laboratory experiments. Qualitative results are shown in Figure 6 as plots of the streamlines around the cylinder. Recirculation regions behind the circular cylinder are shown in Figure 6 for values of \( Re = 20, 40 \) and \( 80 \). This simulation serves to demonstrate the capability of the method to simulate separated flows.

### 3.3 Natural convection in an inclined cavity

A standard two-dimensional enclosure consisting of two adiabatic walls and two walls which are heated at different temperatures, shown in Figure 7, is considered next to test the numerical implementation of both Dirichlet and Neumann boundary conditions. The cavity is rotated clockwise an angle \( \gamma = 3\pi/8 \) between the axes \( x_1 \) and \( x_1^* \), with the gravity force acting along the \( x_2^* \)-axis. The sides of the cavity are of length \( L = 1 \), and the inclination angle is \( \varphi = \pi/4 \). In the present case we use Eqs. (1)–(3) with \( P_1 = Pr, \ P_2 = RaPr, \) and \( P_3 = 1 \). The computations are carried out in the domain \(-0.5 \leq x_1 \leq 0.5\) and \(-1 \leq x_2 \leq 1\), with the four immersed-boundary lines \( x_2 = \pm \tan(\gamma) x_1 \pm \cos(\varphi/2) \). The momentum forcing is applied at the points close to the outside immersed boundary as we are interested in the solution inside (see Figure 7). No-slip/no-penetration conditions for the velocities are imposed on all the boundaries, and non-dimensional temperatures \( \Theta = 0 \) and \( \Theta = 1 \) are applied to the boundaries on the first and third quadrants, respectively. The walls located on the second and fourth quadrants are insulated. In this problem, flows at \( Ra = 10^6 \) were analyzed for two values of the Prandtl number: \( Pr = 0.1 \) and \( 10 \) corresponding to \( Re = 10^4 \) and \( 10^3 \). Our results were compared to the benchmark results of Demirdžić et al. [30].

The predicted streamlines for \( Ra = 10^6 \) and \( Pr = 10 \) and \( 0.1 \) are depicted in Figure 8, showing the effect of the Prandtl number on the flow pattern. In the case of \( Pr = 10 \) there is one free stagnation point located at the center of the cavity with no counter-rotating vortices as shown in Figure 8(a). In contrast, when \( Pr = 0.1 \), Figure 8(b) shows the presence of two free stagnation points and one central loci with three rotating vortices. Due to the effect of convection on the flow, horizontal isotherms, shown in Figure 9, appear in the central region of the cavity for the two Prandtl numbers considered. Steep gradients near the isothermal walls are also present. These qualitative results agree very well with those of Demirdžić et al. [30].
For comparison purposes, predicted Nusselt numbers along the cold wall for different grid points from our method and the results of Demirdžić et al. [30] are shown in Figure 10 for $Ra = 10^6$ and $Pr = 0.1$. As can be observed from the figure, the Nusselt number $Nu$ with a $100 \times 100$ grid is scattered along the wall showing poor convergence to the ‘exact result’ of Demirdžić et al. [30]. On the other hand, there is no discernible difference between the Nusselt values obtained with a $200 \times 200$ grid and the benchmark results from Demirdžić et al. [30] illustrating the correctness in the implementation of the Neumann boundary conditions.

3.4 Cylinder placed eccentrically in a square enclosure

The algorithm proposed here is now applied to simulate the laminar natural convection of a heated cylinder placed eccentrically in a square duct. The results were compared with those of Demirdžić et al. [30], who analyzed half of the cavity with a $256 \times 128$ grid. The geometry of this test case is shown in Figure 11. The dimensionless diameter (scaled by the side length of the duct) is $d = 0.4$, with the center cylinder being shifted upwards from the duct center by $\epsilon = 0.1$. The cylinder has a non-dimensional wall temperature $\Theta = 1$, whereas the cavity has isothermal vertical side walls at $\Theta = 0$ and adiabatic horizontal walls. The flow is governed by Eqs. (1)-(3) with $P_1 = Pr$, $P_2 = RaPr$ and $P_3 = 1$.

The predicted isothermal lines passing around the cylinder considering only the effect of pure conduction, are depicted in Figure 12. Labels in the figure correspond to temperatures in the range $0.05 \leq \Theta \leq 0.95$ with intervals of 0.1, so that $a= 0.95$, $b= 0.85$, etc. The value of the total heat flux on the left wall of the cavity was equal to that of the semi-cylinder on the left, since the heat flux is zero along the axis of symmetry. It can be seen a regular pattern in the isotherms being parallel near the hot and cold walls as well as on the heated cylinder. The isotherms are orthogonal on the adiabatic walls, as expected.

Flow and heat transfer features at $Ra = 10^6$ and $Pr = 10$ are presented in Figure 13, where the effect of natural convection can be readily seen in both the streamline- and isothermal-patterns. Figure 13(a) shows the corresponding streamlines, where closed contours rising up in the middle of the cavity and coming down at the cold walls can be noticed. As expected, for the chosen values of
the parameters, there is no flow separation. In the case of Figure 13(b), which shows the isotherms, one can observe the presence of steep gradients in regions between the cylinder and the walls of the cavity which are also expected in order to satisfy the imposed boundary conditions and the flow conditions. All the aforementioned results agree very well with those reported by Demirdžić et al. [30].

Values of the Nusselt number along the cold wall, obtained with grid numbers of 100 × 100, 200 × 200 and 250 × 250, are compared in Figure 14 with those of Demirdžić et al. [30], who used half of the domain and a 256 × 128 grid. We observe that grid independence of our results and the convergence to the ‘exact values’ of Demirdžić are achieved with the relatively small 200 × 200 grid.

CONCLUDING REMARKS

The capability to provide general analyses of fluid flow and heat transfer phenomena has made CFD techniques popular in the simulation and design of engineering equipment. However, while geometrically simple flows are efficiently handled by most CDF methodologies, the discretization of complex flows is still a challenging task. From the methodologies that have been developed to handle geometrically complex flows, the immersed boundary method is a promising alternative since grid generation in time is not needed.

In this work, phenomenologically different problems of fluid flow and convective heat transfer have been analyzed using an immersed boundary technique. The implementation of both Dirichlet and Neumann boundary conditions have been presented in detail, and their validation has been assessed by favorable comparison with numerical and experimental results available in the literature. The method is second-order accurate in space and time and is capable of simulating problems of bodies with complex boundaries on Cartesian meshes.

ACKNOWLEDGMENTS

The authors would like to thank Ms. Renate Mittelmann from the Department of Mathematics at ASU for access to computer facilities. We also acknowledge the useful comments of Dr. Andrew
Orr from University College London and of the anonymous referees for bringing to our attention the Immersed Continuum Method.

REFERENCES


List of Figures

1 Two-dimensional interpolation scheme. .............................................. 20
2 Interpolation scheme for Dirichlet boundary condition at points (a), (b) and (c). . 21
3 Interpolation scheme for Neumann boundary condition. .......................... 22
4 Grid layout and $45^\circ$ oblique immersed boundary for the decaying vortex problem. ● represent the points closest to the outside boundary. ............................... 23
5 Decaying vortex accuracy: $\sim$ is the maximum relative error $\epsilon_{u_1}$ for the velocity $u_1$; $\sim$ represents the leading order of the truncation $O(\delta x_i)^2$ and $N$ is the number of grid points inside the boundary. .............................................. 24
6 Streamlines past a circular cylinder on a $200 \times 200$ grid: (a) $Re = 20$; (b) $Re = 40$; (c) $Re = 80$. .......................................................... 27
7 Grid layout, geometry, and boundary conditions for the inclined cavity benchmark problem. ................................................................. 28
8 Streamlines on a $200 \times 200$ grid for $Ra = 10^6$. (a) $Pr = 10$, and (b) $Pr = 0.1$. ...... 29
9 Isotherms on a $200 \times 200$ grid for $Ra = 10^6$. (a) $Pr = 10$, and (b) $Pr = 0.1$. ........ 30
10 Values of Nusselt number along the cold wall of the inclined cavity problem and $Ra = 10^6$ and $Pr = 0.1$. ○ present results with a $100 \times 100$ grid; ● present results with a $200 \times 200$ grid; △ results of Demirdžić et al. [30] with a $224 \times 194$ grid. .... 31
11 Geometry and boundary conditions for the eccentric cylinder placed in an enclosure with $L = 1$, $d = 0.4$, and $\epsilon = 0.1$. .................................................. 32
12 Predicted isotherms passing around a cylinder with pure conduction. The cylinder is situated inside a cavity of $L = 1$. The solution was obtained with a $200 \times 200$ grid 33
13 Streamlines and isotherms for $Ra = 10^6$ and $Pr = 10$. The solution was obtained on a $200 \times 200$ grid. ................................................................. 35
14 Values of Nusselt number along the cold wall for $Ra = 10^6$ and $Pr = 10$: ○ present results with a $100 \times 100$ grid; △ present results with a $200 \times 200$ grid; --- present results with a $250 \times 250$ grid; ○ results of Demirdžić et al. [30] with a $256 \times 128$ grid. 36
Table 1: Results from laboratory experiments and numerical simulations of drag coefficient $C_D$, wake bubble $x_c$, and Strouhal number $St$ around a cylinder.

<table>
<thead>
<tr>
<th>Study</th>
<th>$Re \rightarrow$</th>
<th>$C_D$</th>
<th>$x_c$</th>
<th>$C_D$</th>
<th>$x_c$</th>
<th>$C_D$</th>
<th>$St$</th>
<th>$C_D$</th>
<th>$St$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lai and Peskin [3]</td>
<td>20</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
</tr>
<tr>
<td>Ye et al. [5]</td>
<td>40</td>
<td>2.03</td>
<td>0.92</td>
<td>1.52</td>
<td>2.27</td>
<td>1.37</td>
<td>0.15</td>
<td>--</td>
<td>--</td>
</tr>
<tr>
<td>Kim et al. [9]</td>
<td>80</td>
<td>--</td>
<td>--</td>
<td>1.51</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>1.33</td>
<td>0.165</td>
</tr>
<tr>
<td>Tritton [25]</td>
<td>100</td>
<td>--</td>
<td>--</td>
<td>1.48</td>
<td>--</td>
<td>1.29</td>
<td>--</td>
<td>--</td>
<td>--</td>
</tr>
<tr>
<td>Weiselsberger [26]</td>
<td></td>
<td>2.05</td>
<td>--</td>
<td>1.70</td>
<td>--</td>
<td>1.45</td>
<td>--</td>
<td>--</td>
<td>--</td>
</tr>
<tr>
<td>Fornberg [27]</td>
<td></td>
<td>2.00</td>
<td>0.91</td>
<td>1.50</td>
<td>2.24</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
</tr>
<tr>
<td>Williamson [28]</td>
<td></td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>0.16</td>
<td>--</td>
</tr>
<tr>
<td>Tseng and Ferziger [29]</td>
<td></td>
<td>--</td>
<td>--</td>
<td>1.53</td>
<td>2.21</td>
<td>--</td>
<td>--</td>
<td>1.42</td>
<td>0.164</td>
</tr>
<tr>
<td>Current</td>
<td></td>
<td>2.08</td>
<td>0.91</td>
<td>1.53</td>
<td>2.28</td>
<td>1.45</td>
<td>0.16</td>
<td>1.41</td>
<td>0.167</td>
</tr>
</tbody>
</table>
Figure 1: Two-dimensional interpolation scheme.
Figure 2: Interpolation scheme for Dirichlet boundary condition at points (a), (b) and (c).
Figure 3: Interpolation scheme for Neumann boundary condition.
Figure 4: Grid layout and 45° oblique immersed boundary for the decaying vortex problem. ● represent the points closest to the outside boundary.
Figure 5: Decaying vortex accuracy: –○– is the maximum relative error $\epsilon_{u_1}$ for the velocity $u_1$; ––– represents the leading order of the truncation $O(\delta x_i)^2$ and $N$ is the number of grid points inside the boundary.
(b)
Figure 6: Streamlines past a circular cylinder on a 200 × 200 grid: (a) $Re = 20$; (b) $Re = 40$; (c) $Re = 80$. 
Figure 7: Grid layout, geometry, and boundary conditions for the inclined cavity benchmark problem.
Figure 8: Streamlines on a $200 \times 200$ grid for $Ra = 10^6$. (a) $Pr = 10$, and (b) $Pr = 0.1$. 
Figure 9: Isotherms on a 200 × 200 grid for $Ra = 10^6$. (a) $Pr = 10$, and (b) $Pr = 0.1$. 
Figure 10: Values of Nusselt number along the cold wall of the inclined cavity problem and $Ra = 10^6$ and $Pr = 0.1$. ○ present results with a $100 \times 100$ grid; ◊ present results with a $200 \times 200$ grid; ▲ results of Demirdzić et al. [30] with a $224 \times 194$ grid.
Figure 11: Geometry and boundary conditions for the eccentric cylinder placed in an enclosure with $L = 1$, $d = 0.4$, and $\epsilon = 0.1$. 
Figure 12: Predicted isotherms passing around a cylinder with pure conduction. The cylinder is situated inside a cavity of $L = 1$. The solution was obtained with a $200 \times 200$ grid.
(a) Streamlines
Figure 13: Streamlines and isotherms for $Ra = 10^6$ and $Pr = 10$. The solution was obtained on a $200 \times 200$ grid.
Figure 14: Values of Nusselt number along the cold wall for $Ra = 10^6$ and $Pr = 10$: ◇ present results with a $100 \times 100$ grid; △ present results with a $200 \times 200$ grid; --- present results with a $250 \times 250$ grid; ○ results of Demirdžić et al. [30] with a $256 \times 128$ grid.