A General Scheme for the Boundary Conditions in Convective and Diffusive Heat Transfer With Immersed Boundary Methods

We describe the implementation of an interpolation technique, which allows the accurate imposition of the Dirichlet, Neumann, and mixed (Robin) boundary conditions on complex geometries using the immersed-boundary technique on Cartesian grids, where the interface effects are transmitted through forcing functions. The scheme is general in that it does not involve any special treatment to handle either one of the three types of boundary conditions. The accuracy of the interpolation algorithm on the boundary is assessed using several two- and three-dimensional heat transfer problems: (1) forced convection over cylinders placed in an unbounded flow, (2) natural convection on a cylinder placed inside a cavity, (3) heat diffusion inside an annulus, and (4) forced convection around a stationary sphere. The results show that the scheme preserves the second-order accuracy of the equations solver and are in agreement with analytical and/or numerical data. [DOI: 10.1115/1.2764083]

Keywords: heat transfer, immersed boundary method, numerical simulations, forced convection, natural convection, general boundary conditions

1 Introduction

The use of the immersed-boundary (IB) technique, and of other Cartesian-grid methods, for simulating geometrically complex fluid flow problems, has increased substantially in the last three decades. The advantages provided by methodologies on Cartesian grids, such as simplicity in grid generation, savings in memory and CPU time, and straightforward parallelization, have been key factors for their expanded use in the analysis and design of engineering equipment. Numerical schemes on Cartesian grids can be broadly classified in two categories: (1) methods where the effects of the boundary are transmitted via forcing functions (IB methods) [1,2] and (2) methods where the boundary effects are embedded in the discrete spatial operators, e.g., ghost-cell and sharp-interface methods [3–6], and the immersed interface method [7–9], which have been applied to the simulation of flows around stationary and moving immersed boundaries. The present work is concerned with the implementation and application of the IB method for heat transfer analyses.

The application of most of the IB schemes reported in the literature has been directed toward the analysis of fluid flow [1,2,10,11] and has only recently been extended to simulate heat transfer phenomena [12–14]. Nevertheless, regardless of the application, the different versions of the IB technique are developed upon the same principle, i.e., to apply a “forcing term” in the discretized momentum and/or energy equations, such that the boundary conditions are satisfied on the body surface [2,10,13]. Though the technique has also drawbacks, e.g., mass conservation near the boundary where the forcing is applied, is not strictly satisfied [15], efforts to alleviate this problem have increased in recent years [10,16].

On the other hand, when using Cartesian grids, the body does not often coincide with the grid points and interpolation schemes are needed to enforce the boundary conditions on the body surface. In this context, several interpolation schemes have been developed and successfully applied to enforce Dirichlet boundary conditions [2,10,12,13]. To a lesser extent, and with less success, interpolation algorithms for enforcing the Neumann (isoflux) conditions have also been reported in the literature [12–14]. In the above mentioned investigations, not only the interpolation schemes developed for Dirichlet boundary conditions were different from those constructed for Neumann conditions owing to their differences in nature but there was no explicit assessment of the accuracy of the isoflux interpolation algorithms.

Mixed Dirichlet–Neumann (Robin) conditions arise in heat and mass diffusion processes when coupled with convection. Examples include the description of heat transfer in microvascular tissues [17], electrokinetic remediation [18], one-phase solidification and melting [19], and reaction-diffusion problems [20], among others. Thus, it would be important to have an algorithm that could handle all three possible combinations of linear boundary conditions that occur in heat transfer phenomena using fixed grid approaches.

The aim of the present article is to address this need, i.e., to develop a second-order single-interpolation scheme that can be applied to enforce either Dirichlet, Neumann, or Robin conditions on the body surface, to analyze heat transfer processes in the context of the IB method. To this end, a brief overview of the IB technique is provided first. The details of the general interpolation scheme are presented next. The accuracy of the approach is then demonstrated by applying the interpolation algorithm to several phenomenologically different two-dimensional heat transfer prob-
prestational number 

2 Mathematical Formulation

We consider the different types of heat transfer problems namely, forced and natural convection heat transfer over heated cylinders, diffusion of heat in an annulus, and forced convection heat transfer around a stationary sphere.

2.1 Governing Equations. For heat convection, a nondimensional version of the governing equations for an unsteady, incompressible, Newtonian fluid flow with constant properties, in the Boussinesq limit, with negligible viscous dissipation, can be written as

\[ \nabla \cdot \mathbf{u} = 0 \]  

\[ \partial_t \mathbf{u} + (\mathbf{u} \cdot \nabla) \mathbf{u} = -\nabla p + \eta \nabla^2 \mathbf{u} + \mathbf{f} + \mathbf{P}_0 \nabla^2 \Theta + h \nabla^2 \mathbf{u} \]  

\[ \partial_t \Theta + (\mathbf{u} \cdot \nabla) \Theta = P_1 \nabla^2 \Theta + h \]  

where \( \mathbf{u} \) is the Cartesian velocity vector of components \( u_i \) (\( i = 1, 2, 3 \)), \( p \) is the pressure, \( \mathbf{f} \) and \( \mathbf{f} \) are the unit vector in the vertical direction and the momentum forcing, respectively, \( \Theta \) is the temperature of the fluid, and \( h \) is the energy forcing. \( \Theta \) and \( p \) are defined according to the scaling of Eqs. (1)–(3) and depend on the problem under analysis. For example, for forced and mixed convection, we scale length with \( L_c \), velocity with \( U \), time with \( L_c / U \), and pressure with \( \rho U^2 \). We define a nondimensional temperature as \( \Theta = (T - T_0) / (T_0 - T_w) \), where \( T_w \) is the wall/body temperature and \( T_0 \) is a reference temperature of the fluid. A Reynolds number for the flow can be defined as \( Re = U L_c / \nu \), where \( \nu = \mu / \rho \) is the kinematic viscosity of the fluid. The Prandtl number is \( Pr = \nu / \alpha \), where \( \alpha \) is the thermal diffusivity of the fluid, and the Grashof number \( Gr = (g \beta L_c^3 \rho / \nu^2) (T_0 - T_w) / (L_c / \nu) \), where \( g \) is the gravitational acceleration, and \( \beta \) is the coefficient of thermal expansion. Therefore, \( P_1 = 1 / Re \), \( P_2 = Gr / Re^2 \), and \( P_3 = 1 / Re Pr \). On the other hand, for natural convection, \( P_1 = Pr \), \( P_2 = Ra Pr \), and \( P_3 = 1 \), where we scale length with \( L_c \), velocity with \( \alpha L_c / \nu \), time with \( L_c^2 / \alpha \), and pressure with \( \rho \alpha^2 L_c^2 / \nu^2 \). The nondimensional temperature \( \Theta = (T - T_0) / (T_0 - T_w) \); hence, the Grashof number becomes \( Gr = (g \beta L_c^3 / \nu^2) (T_0 - T_w) / (T_0 - T_0) \). Note that different quantities can be used in the normalization of the temperature and the definition of the Grashof number. For instance, in the case of nonhomogeneous Neumann conditions, for which there is no temperature difference characteristic of the problem, one could use a mean temperature difference \( T_{\text{avg}} - T_0 \) or a value of the temperature difference halfway along the body, as suggested by Sparrow and Gregg [21].

For the case of unsteady heat conduction within a solid surrounded by a fluid, the nondimensional governing equation with constant properties is given by

\[ \partial_t \Theta = \mathbf{P} \nabla^2 \Theta + H \]  

where \( \Theta \) is the temperature within the solid, and \( H \) is the corresponding energy forcing. We scale length with \( L_c \) and time with a characteristic diffusion time \( t_c \). The nondimensional temperature in the equation above is defined as \( \Theta = (T - T_0) / (T_0 - T_w) \), where \( T_0 \) is the reference temperature, \( T_w \) is the fluid temperature (used here as an upper-bound reference), and \( P_4 = a_c t_c / L_c^2 \), where \( a_c \) is the thermal diffusivity of the solid.

2.2 Projection Method and Time Integration. The nonstaggered-grid layout is employed in this analysis. The pressure and the Cartesian velocity components are defined at the cell center and the volume fluxes are defined at the midpoint of their corresponding faces of the control volume in the computational space. The spatial derivatives are discretized using a variation of QUICK [22], which calculates the face value from the nodal value with a quadratic interpolation scheme. The upwinding schemes are carried out by computing negative and positive volume fluxes. Using a semi-implicit time-advancement scheme with the Adams–Bashforth method for the explicit terms and the Crank–Nicholson method for the implicit terms as described in Refs. [23–26], the discretized equations corresponding to Eqs. (1)–(3) can be written as follows:

\[ \frac{\mathbf{u} - \mathbf{u}^n}{\Delta t} = \frac{1}{2} \left[ -3\left( \mathbf{u} \cdot \nabla \mathbf{u} \right) \mathbf{u} + \left[ \left( \mathbf{u} \cdot \nabla \mathbf{u} \right) \mathbf{u} \right]^{n+1} + P_1 \nabla^2 \left( \mathbf{u}^* + \mathbf{u}^n \right) + \mathbf{f}^{n+1/2} + P_2 \Theta^{n+1} \mathbf{e} \right] \]  

\[ \nabla \cdot \mathbf{u}^* = \frac{\mathbf{u}^* - \mathbf{u}^n}{\Delta t} \]  

\[ \Theta^{n+1} - \Theta^n = \frac{1}{2} \left[ -3\left( \mathbf{u} \cdot \nabla \Theta \right) \Theta + \left[ \left( \mathbf{u} \cdot \nabla \Theta \right) \Theta \right]^{n+1} - P_3 \nabla^2 \left( \Theta^{n+1} + \Theta^n \right) \right] \]  

where \( \mathbf{u}^* \) is the predicted intermediate velocity, and \( \phi \) is often called “pseudopressure.” The Poisson equation for the pressure is solved iteratively using a multigrid method [23].

In the context of the direct forcing method [27], to obtain \( \mathbf{u}^* \) we need to compute the forcing function \( \mathbf{f} \) with the appropriate bound-
To develop the interpolation scheme for the above boundary condition, we consider a two-dimensional body shown in Fig. 1. In reference to this figure, two different types of nodes are possible, e.g., nodes labeled (a), or nodes named either (b) or (c), where \( \mathbf{n}_a, \mathbf{n}_b, \) and \( \mathbf{n}_c \) are the unit vectors defining the tangent planes at the corresponding node.

Consider first the case of the cell on the left end of Fig. 1 for point labeled (a). This case is shown in Fig. 2(a) as point \((p, q)\). In this case, three nodal values (outside of the body) surrounding point \((p, q)\) are known. In the following, for clarity “bars” and “tildes” are the temperature values inside and outside the body, respectively.

We need to find \( \bar{\Theta}_{i,j} \) to compute \( h \) in Eq. (10), such that Eq. (11) is satisfied at node \((p, q)\) (the same applies to \( \bar{\Theta} \), in the diffusion equation Eq. (4) to find \( H \)). Thus, we use a bilinear interpolation scheme, as shown in Fig. 3, where

\[
\alpha = \left[ x_1 - x_{1(i+1,j)} \right] / \left[ x_{1(i+1,j)} - x_{1(i,j)} \right], \quad \beta = \left[ x_2 - x_{2(i,j+1)} \right] / \left[ x_{2(i+1,j)} - x_{2(i,j)} \right], \quad \text{and} \quad \bar{\Theta}_{p,q} = \Theta(x_1, x_2) \text{ on } (p, q).
\]

This is given as

\[
\bar{\Theta}_{p,q} = \alpha \beta \bar{\Theta}_{i+1,j} + \alpha(1 - \beta) \bar{\Theta}_{i+1,j+1} + (1 - \alpha)(1 - \beta) \bar{\Theta}_{i,j+1} + (1 - \alpha)(1 - \beta) \bar{\Theta}_{i,j} \quad (12)
\]

On the other hand, the values of auxiliary nodes \( \bar{\Theta}_{p,j}, \bar{\Theta}_{p,j+1}, \bar{\Theta}_{p,q}, \) and \( \bar{\Theta}_{i+1,q} \), which are required to compute the derivatives for Eq. (11), can be obtained by a linear interpolation scheme, with the following expressions:

\[
\bar{\Theta}_{p,j} = \alpha \bar{\Theta}_{i,j} + (1 - \alpha) \bar{\Theta}_{i+1,j} \quad (13a)
\]

\[
\bar{\Theta}_{p,j+1} = \alpha \bar{\Theta}_{i+1,j+1} + (1 - \alpha) \bar{\Theta}_{i,j+1} \quad (13b)
\]

\[
\bar{\Theta}_{p,q} = \beta \bar{\Theta}_{i+1,j} + (1 - \beta) \bar{\Theta}_{i,j+1} \quad (13c)
\]

\[
\bar{\Theta}_{i+1,q} = \beta \bar{\Theta}_{i+1,j+1} + (1 - \beta) \bar{\Theta}_{i,j} \quad (13d)
\]

On combining Eq. (11) with Eq. (12) and Eqs. (13a)–(13d), the value for \( \bar{\Theta}_{i,j} \) can now be written as

\[
\bar{\Theta}_{i,j} = \left\{ c - \alpha \beta a \bar{\Theta}_{i+1,j+1} - \left[ \frac{ab}{\Delta x_2} n_2 + a(1 - \beta)a \right] \bar{\Theta}_{i,j+1} - \left[ \frac{\beta b}{\Delta x_1} n_1 \right] \bar{\Theta}_{i+1,j} + \left[ \frac{b}{\Delta x_1} n_1 + \frac{(1 - \alpha)b}{\Delta x_2} n_2 \right] \right\} + (1 - \alpha)(1 - \beta)a \quad (14)
\]

where \( n_1 \) and \( n_2 \) are the projections of \( \mathbf{n} \) on the \( x_1 \) and \( x_2 \) axes, respectively. \( \Delta x_1 \) and \( \Delta x_2 \) are the corresponding spatial increments (see Fig. 2). By setting either \( b=0 \) or \( a=0 \) in Eq. (14), \( \bar{\Theta}_{i,j} \) can be computed under Dirichlet or Neumann boundary conditions.

In the second case, two nodal values surrounding point \((p, q)\) lie inside the body, as shown in Fig. 2(b). This case corresponds to the cell on the center of Fig. 1 for point labeled as (b). Apparently, there are only two known temperatures outside the boundary \((\bar{\Theta}_{i,j} \) and \( \bar{\Theta}_{i+1,j} \) and two unknown temperatures inside the boundary \((\bar{\Theta}_{i,j} \) and \( \bar{\Theta}_{i+1,j} \)). However, we regard \( \bar{\Theta}_{i+1,j+1} \) as a known quantity, since \( \bar{\Theta}_{i+1,j} = \Theta_{1a} \), where \( \Theta_{1a} \) was previously obtained from Eq. (14) as
explained above. Therefore, \( \tilde{\Theta}_{ij} (=\Theta_{ib}) \) of Fig. 1) is computed with Eq. (14) and the known surrounding values. Equation (14) can be used to determine the temperature values inside the body such that the desired boundary condition is satisfied. The solution procedure involves the following steps.

1. Find a nodal point where we want to satisfy the Robin boundary condition and three nodal points lie outside the body, e.g., node (a) of Fig. 1.
2. Determine temperature \( \Theta_{ib} (=\tilde{\Theta}_{ib}) \) from the known surrounding values using Eq. (14).
3. Determine the corresponding node temperature of the adjacent cell, e.g., node (b) of Fig. 1, from Eq. (14) with \( \Theta_{ib+1} \) being replaced by \( \tilde{\Theta}_{ib+1} \). In this case, \( \tilde{\Theta}_{ib+1} = \Theta_{ib} \) was previously determined from a bilinear interpolation along with the adjacent nodes external to the body \( \tilde{\Theta}_{ib2}, \tilde{\Theta}_{ib3}, \tilde{\Theta}_{ib4} \), and Eq. (11) being evaluated at node (a), which are all known.
4. Repeat step 3 on the adjacent cell (right end of Fig. 1).
5. Since \( \tilde{\Theta}_{ib} \) must equal \( \tilde{\Theta}_{ib} \), this procedure must be repeated until all the nodes near the body have been exhausted, and the difference in values between consecutive iterations is negligible, e.g., \( \tilde{\Theta}_{ib} - \tilde{\Theta}_{ib} \approx 0 \).

It is to be noted that the number of iterations required to achieve zero machine accuracy, either for two- or three-dimensional simulations, is typically 10 per node.

4 Heat Transfer Simulations

In order to assess the correct implementation of the interpolation algorithm, simulations of four different heat transfer problems are carried out next.

4.1 Forced Convection Over Heated Circular Cylinders

We consider first the forced heat convection over circular cylinders placed in an unbounded uniform flow. Two different cases are analyzed in this section: (i) a single heated cylinder of nondimensional diameter \( d = 1 \) and (ii) an arrangement of two heated cylinders, one being the main cylinder of nondimensional diameter \( d = 1 \), whereas the other is a secondary cylinder of diameter \( d_2 = d/3 \). Note that the characteristic length for these problems is \( L_c = d \). For the two cases, the following considerations take place: The fluid is air, and thus \( Pr = 0.71 \); the inflow boundary conditions are \( u_1 = u_2 = 0 \), and \( \Theta = 0 \); the outflow boundary conditions are \( \partial u/\partial x + c_u \partial \Theta / \partial x = 0 \) and \( \partial \Theta / \partial x + c_\Theta \partial \Theta / \partial x = 0 \) [28] (where \( c_u \) is the space-averaged velocity) for \( i = 1, 2 \); the far-field boundary conditions are \( \partial u/\partial x = 0 \) and \( \partial \Theta / \partial x = 0 \); and the value of the isothermal surface for all the cylinders is \( \Theta = 1 \). The flow is governed by Eqs. (1)–(3) with \( P_f = 1/Re, P_2 = Gr/Re^2 \) (Gr = 0), and \( P_3 = 1/RePr \). The computations presented next were carried out using nonuniform grids, which are stretched away from the vicinity of the body using a hyperbolic sine function for test case (i) and a logarithmic function for test case (ii). A number of 1200 grid points inside the cylinder and 112 grid points close to the boundary were used. In all the cases analyzed here, a 400 \( \times \) 400 mesh secured grid independence. During the computations, the time step value was changed dynamically to ensure a CFL=0.5.

In test problem (i), the cylinder was placed at the center of a computational domain of size 30d \( \times \) 30d. The center of the cylinder has coordinates \( (x_1, x_2) = (0, 0) \), where \( -15 \leq x_1, x_2 \leq 15 \). The results shown next were obtained for Re=40, 80, 120, 150. Table 1 shows a comparison in both the hydrodynamics and the heat transfer of this flow in terms of the drag coefficient \( C_D = F_D/\rho U^2d \), the wake bubble size \( x_b \), or the Strouhal number \( St = d/U \), and the averaged Nusselt number \( \bar{Nu} = \bar{h}d/k \). In the above expressions, \( F_D \) is the drag force on the cylinder [13], \( f \) is the shedding frequency, \( h \) is the heat transfer coefficient averaged over its half arc length, and \( k \) its thermal conductivity. As can be noted, the present results for both fluid flow and heat transfer compare quantitatively well with the published numerical and laboratory experiments [12,29–31]. As an example, for Re=40, 120, and 150, the differences in Nu against the experiments of Eckert and Soehngen [29] are confined to less than 4%.

Figures 4(a) and 4(b) illustrate the streamlines and temperature contours for Re=80. From both figures, it can be seen the development of a Kármán vortex street, resembled by the alternating patterns that take place due to vortex shedding, which is typical for this value of Re [29,32]. On the other hand, Fig. 5 illustrates a comparison in the local values of the Nu number along the cylinder surface obtained from the proposed scheme and the experiments of Eckert and Soehngen [29] for Re=120. In the figure, the angle \( \varphi \) is measured from the leading edge of the cylinder. As can be noticed, there is a very good agreement between the present results and those of Eckert and Soehngen [29]. Also expected is a quantitative increase of the Nu value from 3.62 for Re=40, to 4.70 for Re=80, and to 5.50 for Re=120. The percentage difference between the averaged Nusselt obtained here (for Re=120) and the experimental one reported by Eckert and Soehngen [29], which is 5.69, is of only 3.4%.

Test problem (ii) has been studied experimentally [32] and numerically [12] for different arrangements. Here, we consider two of them for a value of Reynolds number of Re=80. In the first arrangement, the center of the main cylinder was located at \( (x_1, x_2) = (1, 0) \) with respect to a coordinate system placed at the center of a 30d \( \times \) 30d computational domain, whereas the secondary cylinder was centered at \( (x_1, x_2) = (1, 1) \).

Qualitative results for this problem are depicted in Figs. 6(a) and 6(b) in terms of the streamlines and isotherms, respectively. In contrast with the forced convection over the single cylinder, shown previously for the same Re number, the figures illustrate how the interaction between the cylinders, for this arrangement, suppresses the vortex shedding and produces a steady flow [32]. Qualitatively and quantitatively, these results are in good agreement with the data of Kim and Choi [12]. Actual values for the main and secondary cylinders are, respectively, 1.24 and 0.48 for \( C_D \), and 4.67 and 3.0 for Nu.

Table 1 Comparison in \( C_D, x_b, St, \) and Nu around a cylinder placed in an unbounded flow

<table>
<thead>
<tr>
<th>Re → Study</th>
<th>NuCp</th>
<th>40</th>
<th>x_b</th>
<th>Nu</th>
<th>C_D</th>
<th>120</th>
<th>St</th>
<th>Nu</th>
<th>C_D</th>
<th>150</th>
<th>St</th>
<th>Nu</th>
</tr>
</thead>
<tbody>
<tr>
<td>Eckert and Soehngen [29]</td>
<td>—</td>
<td>—</td>
<td>3.48</td>
<td>—</td>
<td>—</td>
<td>5.69</td>
<td>—</td>
<td>—</td>
<td>—</td>
<td>6.29</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Pan [30]</td>
<td>1.51</td>
<td>2.18</td>
<td>3.23</td>
<td>—</td>
<td>—</td>
<td>—</td>
<td>—</td>
<td>—</td>
<td>—</td>
<td>—</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Lima E Silva et al. [31]</td>
<td>1.54</td>
<td>—</td>
<td>—</td>
<td>—</td>
<td>—</td>
<td>—</td>
<td>—</td>
<td>—</td>
<td>—</td>
<td>—</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Current</td>
<td>1.53</td>
<td>2.28</td>
<td>3.62</td>
<td>—</td>
<td>—</td>
<td>1.40</td>
<td>0.170</td>
<td>5.5</td>
<td>1.38</td>
<td>0.170</td>
<td>6.13</td>
<td></td>
</tr>
</tbody>
</table>
for the main and secondary cylinders. Our data for $C_D$ are 1.22 and 0.12, and for $\text{Nu}$ 4.62 and 1.65, respectively.

4.2 Natural Convection Over a Heated Cylinder Inside a Square Cavity. The proposed algorithm is applied next to simulate the laminar natural convection from a heated cylinder placed eccentrically in a square duct of sides $L=1$. The geometry of this validation problem is shown in Fig. 7. The flow and heat transfer are governed by Eqs. (1)-(3) with $P_1=\text{Pr}$, $P_2=\text{RaPr}$, and $P_3=1$, where we have chosen $L_c=L$ as the characteristic length. The cylinder has a nondimensional wall temperature $\Theta=1$, whereas two different sets of thermal boundary conditions are considered for the walls of the cavity: (i) isothermal boundary conditions where all the walls are maintained at $\Theta=0$, and (ii) isothermal vertical sidewalls at $\Theta=0$ and adiabatic horizontal walls, i.e., $\nabla \Theta \cdot \mathbf{n}=0$. For the flow, in both cases, no-slip and no-penetration conditions are applied to all the surfaces.

Test case (i) has been studied by Moukalled and Darwish [33] using a bounded skew central-difference scheme, Sadat and Coutrier [34] with a meshless diffuse approximation method, and Pan [30] with an unstructured-Cartesian-mesh IB method. The cylinder has a dimensionless diameter $d=0.2$, with its center being located at $(x_1,x_2)=(-0.15,-0.15)$ as measured from the center of the duct (see Fig. 7). Our calculations were carried out until grid independence was achieved, with grid sizes ranging from 80\times80 to 200\times200, for values of the Rayleigh number $\text{Ra}=10^4$, $10^5$, and $10^6$. Streamlines and temperature contours for $\text{Ra}=10^4$ and $\text{Ra}=10^6$ are depicted in Figs. 8 and 9, respectively. As observed, the results shown here agree qualitatively well with those presented in Refs. [30,33,34]. Figures 8(a) and 8(b) illustrate the change on the character of the flow from two well-defined rotating vortices to two distorted ones as $\text{Ra}$ is increased from $10^4$ to $10^6$. In the case of the temperature contours, this change is noticed by the shift of the thermal plume now rising toward the left corner of the cavity, as illustrated in Figs. 9(a) and 9(b). Table 2 shows a quantitative comparison in the values of the average Nusselt $\text{Nu}$ number along the cylinder surface for the three values of $\text{Ra}$ considered against published data [30,33,34]. The maximum difference between the current results and those of Pan [30], for instance, is of only 2%.

For test case (ii), in reference to Fig. 7, the cylinder has a dimensionless diameter of $d=0.4$, and its center is located at $(x_1,x_2)=(0,0.1)$ from the center of the duct. This problem has been benchmarked by Demirdžić et al. [35]. The results obtained from the current algorithm are shown in Figs. 10 and 11. Values of the Nusselt number along the cold walls, obtained with a grid size of 200\times200, are compared in Fig. 10 with those of Demirdžić et al. [35], who used half of the domain and a $256 \times 128$ grid. From
the figure, it can be seen that the present results overlap with the “exact values” of Demirdžić et al. [35]. A comparison of the values for the Nusselt number along the cylinder surface, between the present work and that of Demirdžić et al. [35], is shown in Fig. 11. In the figure, the azimuthal angle $\phi$ is measured from top of the cylinder. As before, the agreement of the results obtained from the current scheme as compared to those of Demirdžić et al. [35] is excellent.

4.3 Heat Diffusion in an Annulus. We now carry out simulations of the steady and unsteady heat diffusion in an annulus, illustrated schematically in Fig. 12. This problem was studied by Barozzi et al. [8] using an immersed interface (II) method. The diffusion process within the annulus is governed by Eq. (4), with $P_1=1$. The annulus is initially at $\Theta_{s,0}=0$, with a uniform temperature $\Theta_{s,e}=0$ being applied to the outer surface, and Robin boundary conditions at the internal surface. The present results are compared to the solutions obtained by Barozzi et al. [8] using a nondimensional radius $r \in [1,2]$, where we have used $L_c=r$ as characteristic length.

Three different boundary conditions were applied at the internal boundary in order to test the spatial accuracy of the present formulation. For all cases, the boundary condition at the outer radius is of the Dirichlet type ($\Theta_{s,e}=0$). The boundary conditions applied at the inner radius were (a) Dirichlet $a=1$, $b=0$, $c=1$; (b) Neumann $a=0$, $b=1$, $c=1$; and (c) mixed $a=1$, $b=1$, $c=1$. Different numbers of regular grid points, ranging from 20 to 120 in an uniform Cartesian mesh, were used. For the three boundary conditions above, accuracy of the solutions is presented in terms of the maximum $\max$- and $L_2$-norm distributions in Figs. 13(a)–13(c), and average values in Table 3. For comparison purposes, the results reported by Barozzi et al. [8] are also included. From the figures, and the table, it can be seen that both techniques achieve second-order accuracy, with actual numerical orders being quantitatively similar. It is to be noted that the uneven distributions of the errors in Figs. 13(b) and 13(c), as the mesh is refined, may be due to the presence of the irregular boundary whose effect is to produce a local increase in the error when the derivative is computed.

The time-dependent system, given in Eq. (4) under general conditions of the inner boundary, is analyzed next, and the results are compared with analytical solutions. Following the procedure described by Carslaw and Jaeger [36], and Özişik [37], the solution of Eq. (4) with $\Theta_{s,0}=0$, $\Theta_{s,e}=0$, and $a=b=c=1$ in Eq. (11) is given in terms of Bessel series expansions as
where the eigenvalues, $\lambda_j$ for $j=1,2,\ldots$, are the roots of the equation

$$\Theta_j(r,t) = \sum_{j=1}^{\infty} c_j \left[ \frac{J_0(\lambda_j r)}{J_0(2\lambda_j)} - \frac{Y_0(\lambda_j r)}{Y_0(2\lambda_j)} \right] \exp(-P_4 \lambda_j^2 t) + \frac{\ln(r+1)}{\ln(1/2) - 1}$$

and the constants $c_j$ for $j=1,2,\ldots$, are defined as

$$c_j = \frac{(2/\lambda_j)[(J_1(\lambda_j)/J_0(2\lambda_j)) - (Y_1(\lambda_j)/Y_0(2\lambda_j))] - [(1/\lambda_j^2 + 1)][(J_0(\lambda_j)/J_0(2\lambda_j)) - (Y_0(\lambda_j)/Y_0(2\lambda_j))]^2}{4[(J_1(\lambda_j)/J_0(2\lambda_j)) - (Y_1(\lambda_j)/Y_0(2\lambda_j))]^2}$$

In the above equations, $J_0$, $Y_0$, $J_1$, and $Y_1$ are the Bessel functions of first and second kinds, of order 0 or 1, respectively.

The time-dependent results are shown in Fig. 14 as temperature distributions within the solid annulus at values of the nondimensional time $t=0.15$, 0.25, 0.35, and 1.40. From the figure, it can be observed an excellent agreement between the analytical solutions (symbols) with those obtained numerically (solid lines). After $t = 1.40$ nondimensional units, the steady state has been reached, with a value of $\Theta_s = 0.41$ at the inner boundary. The error between the two solutions is confined to less than 1%.

The temporal accuracy of the scheme is assessed next by choosing three radii locations within the annulus (see Fig. 12), corre-
Table 2 Comparison in $\overline{\text{Nu}}$ around a cylinder placed inside a cavity for $\text{Ra}=10^4$, $10^5$, and $10^6$

<table>
<thead>
<tr>
<th>Study</th>
<th>$\text{Ra}=10^4$</th>
<th>$\text{Ra}=10^5$</th>
<th>$\text{Ra}=10^6$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Moukalled and Darwish [33]</td>
<td>4.741</td>
<td>7.435</td>
<td>12.453</td>
</tr>
<tr>
<td>Sadat and Couturier [34]</td>
<td>4.699</td>
<td>7.430</td>
<td>12.421</td>
</tr>
<tr>
<td>Current</td>
<td>4.750</td>
<td>7.519</td>
<td>12.531</td>
</tr>
</tbody>
</table>

sponding to $r=1.03(P_A)$ close to the inner boundary, $r=1.49(P_B)$ approximately at the middle plane, and $r=1.94(P_C)$ close to the outer boundary. Note that the inner boundary condition is of the Robin type. Different time increments, i.e., $\Delta t=0.2$, 0.1, and 0.01, were used, and the results presented in Fig. 13(d) in terms of error distributions. From the figure, it is clear that for all the radii locations considered, the scheme is second-order accurate in time.

### 4.4 Three-Dimensional Forced Convection Over a Heated Sphere

The capacity of the present scheme to handle three-dimensional flows is shown next by solving the flow and heat transfer around a heated sphere. Published numerical and experimental results are used to assess the solutions obtained. Thus, let us consider a sphere of nondimensional diameter $d=1$, where $L_c = d$ is the characteristic length, placed in an unbounded flow. This flow is governed by Eqs. (1)–(3) where $P_i = 1/\text{Re}$, $P_2 = Gr/Re^2$ with $Gr=0$, and $P_3 = 1/\text{RePr}$ with $Pr=0.71$. The boundary conditions for the problem are inflow $u_1=1$, $u_2=u_3=0$, and $\Theta=0$; outflow $\partial u_1/\partial r + c_p\partial u_1/\partial x_1=0$ and $\partial \Theta/\partial r + c_p\partial \Theta/\partial x_1=0$; and far-field boundaries $\partial u_1/\partial x_2=\partial u_2/\partial x_3=0$, $\partial \Theta/\partial x_2=\partial \Theta/\partial x_3=0$, respectively, where $i=1,2,3$. The sphere is at a nondimensional temperature $\Theta=1$. The computations were carried out using a 100 $\times$ 100 nonuniform grid, which stretches away from the body to the outer boundaries using a hyperbolic sine function. The sphere is located in the center $(x_1,x_2,x_3)=(0,0,0)$ of a computational cube of size $30d \times 30d \times 30d$. The time step value was dynamically changed to ensure a $\text{CFL}=0.5$. The results from the current approach were obtained for Reynolds numbers $Re=50,100,150$ are 1.62, 1.07, and 0.85 for $CD$, 0.42, 0.90, and 1.23 for $x_c$, respectively. For $Re=220$, Fig. 15(e) shows the expected axial asymmetry of the flow, which has been experimentally determined to occur in the $Re \in [210,270]$ range [39].

The corresponding isotherms, shown also in Figs. 15(a)–15(e), reflect the behavior of the flow. As the Reynolds number increases, the isotherms in the wake of the sphere become more distorted due to the amount of recirculating flow. Though small values of the heat transfer coefficient are characteristics of this region, it can be seen that in the rear of the sphere (at an angle of approximately 180 deg), the heat transfer coefficient is increased locally with $Re$ number. The computed values of the Nusselt number averaged over the surface of the sphere $Nu$ are 4.99, 7.00, 8.27, 9.19, and 9.30 for $Re=50$, 100, 150, 200, and 220, respectively. These results are in good agreement with the correlation developed by Feng and Michaelides [40], for the same conditions.

Fig. 10 Nu number along cold wall for $\text{Ra}=10^4$ and $Pr=10$. $\diamond$, present scheme; $\bigcirc$, Demirdžić et al. [35].

Fig. 11 Nu number along cylinder surface for $\text{Ra}=10^6$ and $Pr=10$. Angle $\phi$ is measured from top of cylinder. $\bigdiamond$, present scheme; $\bigcirc$, Demirdžić et al. [35].

Quantitatively, the present results for both drag coefficient $CD$ and wake bubble $x_c$ are in good agreement with those of Marella et al. [4] and Johnson and Patel [38]. Actual values for $Re=50,100$, and 150 are 1.62, 1.07, and 0.85 for $CD$, and 0.42, 0.90, and 1.23 for $x_c$, respectively. For $Re=220$, Fig. 15(e) shows the expected axial asymmetry of the flow, which has been experimentally determined to occur in the $Re \in [210,270]$ range [39].

The corresponding isotherms, shown also in Figs. 15(a)–15(e), reflect the behavior of the flow. As the Reynolds number increases, the isotherms in the wake of the sphere become more distorted due to the amount of recirculating flow. Though small values of the heat transfer coefficient are characteristics of this region, it can be seen that in the rear of the sphere (at an angle of approximately 180 deg), the heat transfer coefficient is increased locally with $Re$ number. The computed values of the Nusselt number averaged over the surface of the sphere $Nu$ are 4.99, 7.00, 8.27, 9.19, and 9.30 for $Re=50$, 100, 150, 200, and 220, respectively. These results are in good agreement with the correlation developed by Feng and Michaelides [40], for the same conditions. The maximum difference between the corresponding values is less
than 8%.

Increasing the Reynolds number beyond 270 eventually leads to unsteady flow. For Re=300, Fig. 16 illustrates the vortical structures of the flow obtained with the method proposed by Hunt et al. [41] (more in-depth details are in Refs. [42,43]). As seen in the figure, these vortical structures resemble very well the established vortex shedding. The average values of the drag coefficient $C_D=0.570$ and the Strouhal number $St=0.133$ compare well to those of Marella et al. [4] and Johnson and Patel [38]. For this value of Re number, the averaged Nusselt number $Nu=10.50$ deviates from that given by Feng and Michaelides [40] correlation in only 3.6%.

5 Concluding Remarks

In the current work, we have presented a novel interpolation scheme that is able to handle either Dirichlet, Neumann, or mixed boundary conditions within the context of the IB methodology. An advantage of this algorithm is that both Dirichlet and Neumann conditions can be naturally implemented by setting appropriate values to the constants in the most general linear equation. Its

Table 3 Numerical order of accuracy in terms of $\epsilon_{\text{max}}$ and $L_2$ for the annulus problem

<table>
<thead>
<tr>
<th>Method / BC</th>
<th>Dirichlet $L_2$</th>
<th>Neumann $\epsilon_{\text{max}}$</th>
<th>Neumann $L_2$</th>
<th>Robin $\epsilon_{\text{max}}$</th>
<th>Robin $L_2$</th>
<th>Robin $\epsilon_{\text{max}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Barozzi et al. [8]</td>
<td>1.91</td>
<td>1.93</td>
<td>2.05</td>
<td>1.99</td>
<td>2.02</td>
<td>1.95</td>
</tr>
<tr>
<td>Current</td>
<td>2.16</td>
<td>2.09</td>
<td>2.02</td>
<td>1.93</td>
<td>2.03</td>
<td>1.93</td>
</tr>
</tbody>
</table>

Fig. 13 (a)–(c) $L_2$ norm and maximum norm ($\epsilon_{\text{max}}$) as functions of the mesh size $\delta$ for different inner boundary conditions. (d) Error versus $\Delta t$ for inner Robin boundary conditions. (a) Dirichlet ($a=1$, $b=0$, $c=1$). (b) Neumann ($a=0$, $b=1$, $c=1$). (c) Robin ($a=1$, $b=1$, $c=1$). (d) Radial locations: $\bigcirc$, $P_A$; $\triangle$, $P_B$; $\star$, $P_C$.

Fig. 14 Numerical (solid lines) versus analytical (symbols) time-dependent solutions. $\bigtriangleup$, $t=0.15$; $\bigcirc$, $t=0.25$; $\bigstar$, $t=0.35$; $\blacktriangle$, $t=1.40$ (steady state).
validation has been assessed by favorable comparison with numerical results available in the literature and/or analytical solutions for different heat transfer problems: forced heat convection over circular cylinders, natural heat convection over a cylinder placed inside a cavity, steady and unsteady diffusion of heat inside an annulus, and three-dimensional forced convection over a sphere. The interpolation algorithm developed here provides a method that is second-order accurate in space and time and is suitable for analyzing heat transfer phenomena in single or multiple bodies with complex boundaries on two- and three-dimensional Cartesian meshes.

Acknowledgment

The authors thank Professor R.L. Street of Stanford University for providing the multigrid subroutine used in this work and to G. Constantinescu of University of Iowa for his assistance with the vortex visualization. A. P.-V. also acknowledges financial support from PROMEP, México through grants PROMEP/PTC-68 and PROMEP-CA9/FCQ-UASLP, and FAI/UASLP research fund. We are grateful for the helpful comments of the anonymous referees which improved the quality of the manuscript. Computational resources for this work were provided by the Ira A. Fulton High Performance Computing Initiative at ASU.

Nomenclature

\( a, b, c \) = constants in general boundary condition

\( c_j \) = \( j \)th constant in Eq. (15)

\( c_u \) = space-averaged velocity

\( e \) = unit vector in vertical direction

\( f \) = momentum forcing

\( h, H \) = energy forcing

\( L_c \) = characteristic length

\( L_2 \) = Euclidean norm

\( n_1, n_2 \) = projections on \( x_1 \) and \( x_2 \) axes of unit vector \( n \)

\( P_1, P_2, P_3, P_4 \) = nondimensional parameters

\( P_A, P_B, P_C \) = radii locations

\( r \) = radius

\( u \) = Cartesian velocity vector

\( x_c \) = wake bubble

\( x_i \) = Cartesian coordinates

\( x_{1(i,j)}, x_{2(i,j)} \) = coordinates based on node \((i,j)\) for bilinear interpolation

\( X_1, X_2 \) = coordinates of immersed-boundary node

Greek Symbols

\( \alpha \) = thermal diffusivity of fluid, interpolation factor

\( \alpha_s \) = thermal diffusivity of solid

\( \beta \) = coefficient of thermal expansion, interpolation factor

\( \Delta x_1, \Delta x_2 \) = spatial increments in \( x_1 \) and \( x_2 \) directions

\( \delta \) = mesh size

\( \epsilon \) = error

\( \epsilon_{\text{max}} \) = maximum norm

\( \lambda_j \) = \( j \)th eigenvalue

\( \nabla \) = Nabla operator

\( \Theta \) = normalized temperature

\( \Theta_s \) = normalized initial temperature within the solid

\( \Theta_{s,0} \) = normalized initial temperature within the solid

Fig. 15 Streamlines and temperature contours for flow around a sphere in the \( 50 \leq \text{Re} \leq 220 \) range. (a) \( \text{Re}=50 \), (b) \( \text{Re}=100 \), (c) \( \text{Re}=150 \), (d) \( \text{Re}=200 \), and (e) \( \text{Re}=220 \).

Fig. 16 Oblique view of vortical structures of flow for \( \text{Re} = 300 \).
\( \Theta_{\infty} \)  = normalized reference fluid temperature
\( \Theta \)  = temperature on or inside the body
\( \Theta_e \)  = temperature outside the body in energy-forcing interpolation
\( \varphi \)  = reference angle

Subscripts and Superscripts
\( e \)  = outer-boundary value
\( i, j \)  = indices for the Cartesian coordinates
\( n \)  = index for time step

References