Arturo Pacheco-Vega¹

CIEP-Facultad de Ciencias Químicas,
Universidad Autónoma de San Luis Potosí,
San Luis Potosí 78210, Mexico
e-mail: apacheco@uaslp.mx

J. Rafael Pacheco

Tamara Rodić

Mechanical and Aerospace Engineering
Department,
Arizona State University,
Tempe, Arizona 85287-6106

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 the 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 that 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-

¹Corresponding author.

Contributed by the Heat Transfer Division of ASME for publication in the JOURNAL OF HEAT TRANSFER. Manuscript received June 19, 2006; final manuscript received February 12, 2007. Review conducted by Louis C. Burmeister. Paper presented at the 2006 ASME International Mechanical Engineering Congress (IMECE2006), Chicago, IL, November 5–10, 2006.

62 lems. Finally, the capability of the scheme to handle three-
63 dimensional problems is demonstrated by solving the forced con-
64 vection around a stationary sphere.

65 2 Mathematical Formulation

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

70 **2.1 Governing Equations.** For heat convection, a nondimen-
71 sional version of the governing equations for an unsteady, incom-
72 pressible, Newtonian fluid flow with constant properties, in the
73 Boussinesq limit, with negligible viscous dissipation, can be writ-
74 ten as

$$75 \quad \nabla \cdot \mathbf{u} = 0 \quad (1)$$

$$76 \quad \partial_t \mathbf{u} + (\mathbf{u} \cdot \nabla) \mathbf{u} = -\nabla p + P_1 \nabla^2 \mathbf{u} + \mathbf{f} + P_2 \Theta \mathbf{e} \quad (2)$$

$$77 \quad \partial_t \Theta + (\mathbf{u} \cdot \nabla) \Theta = P_3 \nabla^2 \Theta + h \quad (3)$$

78 where $\mathbf{u}$ is the Cartesian velocity vector of components u_i (i
79 $= 1, 2, 3$), p is the pressure, $\mathbf{e}$ and $\mathbf{f}$ are the unit vectors in the
80 vertical direction and the momentum forcing, respectively, Θ is
81 the temperature of the fluid, and h is the energy forcing. P_1 , P_2 ,
82 and P_3 are defined according to the scaling of Eqs. (1)–(3) and
83 depend on the problem under analysis. For example, for forced
84 and mixed convection, we scale length with L_c , velocity with U ,
85 time with L_c/U , and pressure with ρU^2 . We define a nondimen-
86 sional temperature as $\Theta = (T - T_0)/(T_w - T_0)$, where T_w is the wall/
87 body temperature and T_0 is a reference temperature. A Reynolds
88 number for the flow can be defined as $Re = UL_c/\nu$, where ν
89 $= \mu/\rho$ is the kinematic viscosity of the fluid. The Prandtl number
90 is $Pr = \nu/\alpha$, where α is the thermal diffusivity of the fluid, and the
91 Grashof number $Gr = (g\beta L_c^3/U^2)(T_w - T_0)/(UL_c/\nu)^2$, where g is the
92 gravitational acceleration, and β is the coefficient of thermal ex-
93 pansion. Therefore, $P_1 = 1/Re$, $P_2 = Gr/Re^2$, and $P_3 = 1/RePr$. On
94 the other hand, for natural convection, $P_1 = Pr$, $P_2 = RaPr$, and P_3
95 $= 1$, where we scale length with L_c , velocity with α/L_c , time with
96 L_c^2/α , and pressure with $\rho\alpha^2/L_c^2$. The nondimensional temperature
97 is defined as $\Theta = (T - T_0)/(T_w - T_0)$; thence, the Grashof number
98 becomes $Gr = (g\beta L_c^3/\nu^2)(T_w - T_0)$. Note that different quantities
99 can be used in the normalization of the temperature and the defini-
100 tion of the Grashof number. For instance, in the case of nonho-
101 mogeneous Neumann conditions, for which there is no tempera-
102 ture difference characteristic of the problem, one could use a mean
103 temperature difference $\bar{T}_w - T_0$ or a value of the temperature dif-
104 ference halfway along the body, as suggested by Sparrow and
105 Gregg [21].

106 For the case of unsteady heat conduction within a solid sur-
107 rounded by a fluid, the nondimensional governing equation with
108 constant properties is given by

$$109 \quad \partial_t \Theta_s = P_4 \nabla^2 \Theta_s + H \quad (4)$$

110 where Θ_s is the temperature within the solid, and H is the corre-
111 sponding energy forcing. We scale length with L_c and time with a
112 characteristic diffusion time t_c . The nondimensional temperature
113 in the equation above is defined as $\Theta_s = (T_s - T_0)/(T_\infty - T_0)$, where
114 T_0 is the reference temperature, T_∞ is the fluid temperature (used
115 here as an upper-bound reference), and $P_4 = \alpha_s t_c/L_c^2$, where α_s is
116 the thermal diffusivity of the solid.

117 **2.2 Projection Method and Time Integration.** The
118 nonstaggered-grid layout is employed in this analysis. The pres-
119 sure and the Cartesian velocity components are defined at the cell
120 center and the volume fluxes are defined at the midpoint of their
121 corresponding faces of the control volume in the computational
122 space. The spatial derivatives are discretized using a variation of
123 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:

$$\frac{\mathbf{u}^* - \mathbf{u}^n}{\Delta t} = \frac{1}{2} \{-3[(\mathbf{u} \cdot \nabla) \mathbf{u}]^n + [(\mathbf{u} \cdot \nabla) \mathbf{u}]^{n-1} + P_1 \nabla^2 (\mathbf{u}^* + \mathbf{u}^n)\} \\ + \mathbf{f}^{n+1/2} + P_2 \Theta^{n+1} \mathbf{e} \quad (5)$$

$$\nabla^2 \phi^{n+1} = \frac{\nabla \cdot \mathbf{u}^*}{\Delta t} \quad (6)$$

$$\mathbf{u}^{n+1} = \mathbf{u}^* - \Delta t \nabla \phi^{n+1} \quad (7)$$

$$\frac{\Theta^{n+1} - \Theta^n}{\Delta t} = \frac{1}{2} \{-3[(\mathbf{u} \cdot \nabla) \Theta]^n + [(\mathbf{u} \cdot \nabla) \Theta]^{n-1} \\ + P_3 \nabla^2 (\Theta^{n+1} + \Theta^n)\} + h^{n+1/2} \quad (8)$$

where $\mathbf{u}^*$ is the predicted intermediate velocity, and ϕ 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 $\mathbf{f}$ in advance, such that $\mathbf{u}^{n+1}$
satisfies the boundary condition on the IB (similar argument is
applied to the energy forcing h or H). One can enforce the proper
boundary condition on $\mathbf{u}^*$ instead of $\mathbf{u}^{n+1}$ without compromising
the temporal accuracy of the scheme [10]; thence, we replace $\mathbf{u}^*$
with $\bar{\mathbf{U}}$ in Eq. (5) and $\bar{\Theta}$ for Θ^{n+1} in Eq. (8) and solve for the
forcings:

$$\mathbf{f}^{n+1/2} = \frac{\bar{\mathbf{U}} - \mathbf{u}^n}{\Delta t} + \frac{1}{2} \{3[(\mathbf{u} \cdot \nabla) \mathbf{u}]^n - [(\mathbf{u} \cdot \nabla) \mathbf{u}]^{n-1} - P_1 \nabla^2 (\mathbf{u}^* + \mathbf{u}^n)\} \\ - P_2 \Theta^{n+1} \mathbf{e} \quad (9)$$

$$h^{n+1/2} = \frac{\bar{\Theta} - \Theta^n}{\Delta t} + \frac{1}{2} \{3[(\mathbf{u} \cdot \nabla) \Theta]^n - [(\mathbf{u} \cdot \nabla) \Theta]^{n-1} \\ - P_3 \nabla^2 (\Theta^{n+1} + \Theta^n)\} \quad (10)$$

where $\bar{\mathbf{U}}$ is the boundary condition for the velocity on the body
surface or inside the body with $\mathbf{f} = 0$ within the fluid. In the same
context, $\bar{\Theta}$ refers to the temperature at the energy-forcing location
that will ensure that the desired boundary condition is satisfied.
Taking the energy-forcing in Eq. (10) as an example, when the
location of h coincides with the boundary then $\bar{\Theta} = \Theta$; otherwise $\bar{\Theta}$
must be obtained by interpolation from the surrounding tempera-
ture values. The procedure just described also applies for the so-
lution of Eq. (4) to obtain H and Θ_s , with the appropriate bound-
ary conditions. Details of the methodology to determine $\mathbf{f}$ and h
(or H), are fully described in Ref. [13] and references therein.
Thence, we concentrate in the implementation of the general in-
terpolation scheme for the Dirichlet, Neumann, and mixed bound-
ary conditions to solve the energy equation.

3 General Interpolation Scheme

The most general linear boundary condition needed to solve the
energy equations, Eqs. (3) and (4), is given as

$$a\Theta + b \nabla \Theta \cdot \mathbf{n} = c \quad (11)$$

where a , b , and c are parameters defined in accordance with the
normalization that is used for the problem under analysis, and $\mathbf{n}$ is
the normal unit vector.

Fig. 1 Interpolation scheme at nodes (a), and (b) or (c)

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

Fig. 2 General interpolation scheme for Dirichlet–Neumann boundary conditions: (a) three nodes outside the boundary; (b) two nodes outside the boundary

Fig. 3 Bilinear interpolation

“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 $\bar{\theta}_s$ in the diffusion equation Eq. (4) to find H). Thus, we use a bilinear interpolation scheme, as shown in Fig. 3, where $\alpha = [X_1 - x_{1(i,j)}] / [x_{1(i+1,j)} - x_{1(i,j)}]$, $\beta = (X_2 - x_{2(i,j)}) / (x_{2(i+1,j)} - x_{2(i,j)})$, and $\bar{\theta}_{p,q} = \theta(X_1, X_2)$ on (p,q). This is given as

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

On the other hand, the values of auxiliary nodes $\bar{\theta}_{p,j}$, $\bar{\theta}_{p,j+1}$, $\bar{\theta}_{i,q}$ 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:

$$\bar{\theta}_{p,j} = \alpha\bar{\theta}_{i+1,j} + (1-\alpha)\bar{\theta}_{i,j} \quad (13a)$$

$$\bar{\theta}_{p,j+1} = \alpha\bar{\theta}_{i+1,j+1} + (1-\alpha)\bar{\theta}_{i,j+1} \quad (13b)$$

$$\bar{\theta}_{i,q} = \beta\bar{\theta}_{i,j+1} + (1-\beta)\bar{\theta}_{i,j} \quad (13c)$$

$$\bar{\theta}_{i+1,q} = \beta\bar{\theta}_{i+1,j+1} + (1-\beta)\bar{\theta}_{i+1,j} \quad (13d)$$

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

$$\begin{aligned} \bar{\theta}_{i,j} = & \left\{ c - \alpha\beta a\bar{\theta}_{i+1,j+1} - \left[\frac{\alpha b}{\Delta x_2} n_2 + \alpha(1-\beta)a \right] \bar{\theta}_{i+1,j} - \left[\frac{\beta b}{\Delta x_1} n_1 \right. \right. \\ & + (1-\alpha)\beta a \left. \right] \bar{\theta}_{i,j+1} - \frac{b}{\Delta x_1} n_1 \bar{\theta}_{i+1,q} \\ & - \frac{b}{\Delta x_2} n_2 \bar{\theta}_{p,j+1} \left. \right\} / \left[\frac{(1-\beta)b}{\Delta x_1} n_1 + \frac{(1-\alpha)b}{\Delta x_2} n_2 \right. \\ & \left. + (1-\alpha)(1-\beta)a \right] \end{aligned} \quad (14)$$

where n_1 and n_2 are the projections of $\mathbf{n}$ on the x_1 and x_2 axes, respectively. Δx_1 and Δx_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 ($\bar{\theta}_{3b}$ and $\bar{\theta}_{4b}$) and two unknown temperatures inside the boundary ($\bar{\theta}_{1b}$ and $\bar{\theta}_{2b}$). However, we regard $\bar{\theta}_{2b}$ as a known quantity, since $\bar{\theta}_{2b} = \bar{\theta}_{1a}$, where $\bar{\theta}_{1a}$ was previously obtained from Eq. (14) as

Table 1 Comparison in C_D , x_c , St , and Nu around a cylinder placed in an unbounded flow

Re→ Study	C_D	40 x_c	Nu	C_D	120 St	Nu	C_D	150 St	Nu
Kim and Choi [12]	—	—	3.23	—	—	5.62	—	—	—
Eckert and Soehngen [29]	—	—	3.48	—	—	5.69	—	—	6.29
Pan [30]	1.51	2.18	3.23	—	—	—	—	—	—
Lima E Silva et al. [31]	1.54	—	—	—	—	—	1.37	0.170	—
Current	1.53	2.28	3.62	1.40	0.170	5.5	1.38	0.179	6.13

explained above. Therefore, $\bar{\Theta}_{i,j} (= \bar{\Theta}_{1b})$ 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 $\bar{\Theta}_{i,j} (= \bar{\Theta}_{1a})$ 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 $\bar{\Theta}_{i,j+1}$ being replaced by $\bar{\Theta}_{i,j+1}$. In this case, $\bar{\Theta}_{i,j+1} = \bar{\Theta}_{1a}$ was previously determined from a bilinear interpolation along with the adjacent nodes external to the body $\bar{\Theta}_{2a}$, $\bar{\Theta}_{3a}$, $\bar{\Theta}_{4a}$, 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 $\bar{\Theta}_{1b}$ must equal $\bar{\Theta}_{2c}$, 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., $\bar{\Theta}_{1b} - \bar{\Theta}_{2c} \approx 0$.

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 $d_s=d/3$. 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 $u_1=1$, $u_2=0$, and $\Theta=0$; the outflow boundary conditions are $\partial u_i / \partial t + c_u \partial u_i / \partial x_1 = 0$ and $\partial \Theta / \partial t + c_u \partial \Theta / \partial x_1 = 0$ [28] (where c_u is the space-averaged velocity) for $i=1,2$; the far-field boundary conditions are $\partial u_i / \partial x_2 = 0$ and $\partial \Theta / \partial x_2 = 0$; and the value of the isothermal surface for all the cylinders is $\Theta=1$. The flow is governed by Eqs. (1)–(3) with $P_1=1/Re$, $P_2=Gr/Re^2$ ($Gr=0$), 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 400×400 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 $C_D = F_D / \frac{1}{2} \rho U^2 d$, the wake bubble x_c or the Strouhal number $St = fd/U$, and the averaged Nusselt number $Nu = \bar{h}d/k$. In the above expressions, F_D is the drag force on the cylinder [13], f is the shedding frequency, $\bar{h}$ 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 that the development of a Kármán vortex street resembled also by the alternating patterns in the isotherms 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 φ 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 $Re=40$ –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 4.7 and 3.0 for Nu .

Computations were also carried out for an arrangement where the main cylinder was centered at $(x_1, x_2) = (-1, 0)$ and the secondary cylinder at $(x_1, x_2) = (-1, 0.5)$. As expected for this arrangement [12,32], our results showed that the flow is unsteady. Again, good agreement with Ref. [12] is found in the values of both C_D

Fig. 4 Streamlines and temperature contours of flow around cylinder for $Re=80$: (a) streamlines; (b) isotherms

and $\overline{Nu}$ for the main and secondary cylinders. Our data for C_D are 1.22 and 0.12, and for 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 $P_1=Pr$, $P_2=RaPr$, 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 80×80 to 200×200 , for values of the Rayleigh number $Ra=10^4$, 10^5 , and 10^6 . Streamlines and temperature contours for $Ra=10^4$ and $Ra=10^6$ 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 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 number along the cylinder surface for the three values of 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 200×200 , are compared in Fig. 10 with those of Demirdžić et al. [35], who used half of the domain and a 256×128 grid. From

Fig. 5 Nu number along cylinder surface for $Re=120$. $\circ$, present scheme; $\diamond$, experiments by Eckert and Soehngen [29].

Fig. 6 Streamlines and temperature contours of flow around two cylinders for $Re=80$: (a) Streamlines; (b) Isotherms

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 φ 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 $P_4=1$. 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 $L_c=r_i$ 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 20×20 to 120×120 in a uniform Cartesian mesh, were used. For the three boundary conditions above, accuracy of the solutions is presented in terms of the maximum $\epsilon_{\max}$ 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

Fig. 7 Cylinder placed eccentrically in a square cavity

(a)

(b)

Fig. 8 Streamlines on a 200×200 grid for (a) $Ra=10^4$ and (b) $Ra=10^6$

(a)

(b)

Fig. 9 Isotherms on a 200×200 grid for (a) $Ra=10^4$ and (b) $Ra=10^6$

$$\Theta_s(r, t) = \sum_{j=1}^{\infty} c_j \left[\frac{J_0(\lambda_j r)}{J_0(2\lambda_j)} - \frac{Y_0(\lambda_j r)}{Y_0(2\lambda_j)} \right] \exp(-P_4 \lambda_j^2 t) + \frac{\ln(r) + 1}{\ln(1/2) - 1} \quad (15)$$

where the eigenvalues, λ_j for $j=1, 2, \dots$, are the roots of the equation

$$\frac{\lambda_j J_1(\lambda_j) + J_0(\lambda_j)}{J_0(2\lambda_j)} - \frac{\lambda_j Y_1(\lambda_j) + Y_0(\lambda_j)}{Y_0(2\lambda_j)} = 0 \quad (16)$$

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

$$c_j = \frac{2/\lambda_j \{ [J_1(\lambda_j)/J_0(2\lambda_j)] - [Y_1(\lambda_j)/Y_0(2\lambda_j)] \}}{4 \{ [J_1(2\lambda_j)/J_0(2\lambda_j)] - [Y_1(2\lambda_j)/Y_0(2\lambda_j)] \}^2 - [(1/\lambda_j^2) + 1] \{ [J_0(\lambda_j)/J_0(2\lambda_j)] - [Y_0(\lambda_j)/Y_0(2\lambda_j)] \}^2} \quad (17)$$

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 $\overline{Nu}$ around a cylinder placed inside a cavity for $Ra=10^4$, 10^5 , and 10^6

Study	$Ra=10^4$	$Ra=10^5$	$Ra=10^6$
Moukalled and Darwish [33]	4.741	7.435	12.453
Sadat and Couturier [34]	4.699	7.430	12.421
Pan [30]	4.686	7.454	12.540
Current	4.750	7.519	12.531

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 $P_1=1/Re$, $P_2=Gr/Re^2$ with $Gr=0$, and $P_3=1/RePr$ with $Pr=0.71$. The boundary conditions for the problem are inflow $u_1=1$, $u_2=u_3=0$, and $\Theta=0$; outflow $\partial u_i/\partial t + c_u \partial u_i/\partial x_1=0$ and $\partial \Theta/\partial t + c_u \partial \Theta/\partial x_1=0$; and far-field boundaries $\partial u_i/\partial x_2=\partial u_i/\partial x_3=0$, $\partial \Theta/\partial x_2=\partial \Theta/\partial x_3=0$, respectively, where $i=1,2,3$. The sphere is at a nondimensional temperature $\Theta=1$. The computations were carried out using a $100 \times 100 \times 100$ 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 CFL=0.5. The results from the current approach were obtained for Reynolds numbers $Re=\{50, 100, 150, 200, 220, 300\}$.

Figures 15(a)–15(e) illustrate the streamlines and isotherms on the symmetry plane $x_3=0$ for $Re=50, 100, 150, 200$, and 220 , respectively. The streamlines presented in Figs. 15(a)–15(d) show that the flow is steady and separated, with an axisymmetric recirculation bubble that grows as the Reynolds number increases.

Fig. 10 Nu number along cold wall for $Ra=10^6$ and $Pr=10$. $\diamond$, present scheme; $\circ$, Demirdžić et al. [35].

Fig. 11 Nu number along cylinder surface for $Ra=10^6$ and $Pr=10$. Angle ϕ is measured from top of cylinder. $\diamond$, present scheme; $\circ$, Demirdžić et al. [35].

Quantitatively, the present results for both drag coefficient C_D 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 C_D , 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 $\overline{Nu}$ 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 [10], for the same conditions. The maximum difference between the corresponding values is less

Fig. 12 Concentric cylindrical annulus.

Fig. 13 ((a)–(c)) L_2 norm and maximum norm ($\epsilon_{\max}$) as functions of the mesh size δ for different inner boundary conditions. (d) Error versus Δt 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) Radii locations: $\diamond$, P_A ; $\circ$, P_B ; $\square$, P_C .

493 than 8%.

494 Increasing the Reynolds number beyond 270 eventually leads to
 495 unsteady flow [38]. For $Re=300$, Fig. 16 illustrates the vortical
 496 structures of the flow obtained with the method proposed by Hunt
 497 et al. [41] (more in-depth details are in Refs. [42,43]). As seen in
 498 the figure, these vortical structures resemble very well the estab-
 499 lished vortex shedding. The average values of the drag coefficient
 500 $C_D=0.570$ and the Strouhal number $St=0.133$ compare well to
 501 those of Marella et al. [4] and Johnson and Patel [38]. For this
 502 value of Re number, the averaged Nusselt number $Nu=10.50$ de-
 503 viates from that given by Feng and Michaelides [40] correlation in
 504 only 3.6%.

505 5 Concluding Remarks

506 In the current work, we have presented a novel interpolation
 507 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 $\epsilon_{\max}$ and L_2 for the annulus problem

Method / BC $\rightarrow$	Dirichlet		Neumann		Robin	
	L_2	$\epsilon_{\max}$	L_2	$\epsilon_{\max}$	L_2	$\epsilon_{\max}$
Barozzi et al. [8]	1.91	1.93	2.05	1.99	2.02	1.95
Current	2.16	2.09	2.02	1.93	2.03	1.93

Fig. 14 Numerical (solid lines) versus analytical (symbols) time-dependent solutions. $\triangle$, $t=0.15$; $\diamond$, $t=0.25$; $\circ$, $t=0.35$; ∇ , $t=1.40$ (steady state).

512
513
514
515
516
517
518
519
520
521
522

523
524
525
526
527
528
529
530
531
532
533

534
535
536
537
538
539
540
541
542
543
544
545
546
547
548
549
550
551
552
553
554
555

556
557
558
559
560
561
562
563
564
565
566
567
568
569
570
571

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

Fig. 16 Oblique view of vortical structures of flow for $Re=300$

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. They 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
- $\mathbf{e}$ = unit vector in vertical direction
- $\mathbf{f}$ = 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 $\mathbf{n}$
- P_1, P_2, P_3, P_4 = nondimensional parameters
- P_A, P_B, P_C = radii locations
- r = radius
- $\mathbf{u}$ = 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

- α = thermal diffusivity of fluid, interpolation factor
- α_s = thermal diffusivity of solid
- β = coefficient of thermal expansion, interpolation factor
- $\Delta x_1, \Delta x_2$ = spatial increments in x_1 and x_2 directions
- δ = mesh size
- ϵ = error
- $\epsilon_{\max}$ = maximum norm
- λ_j = j th eigenvalue
- ∇ = Nabla operator
- Θ = normalized temperature
- Θ_s = normalized temperature within the solid
- $\Theta_{s,0}$ = normalized initial temperature within the solid

- 572 Θ_{∞} = normalized reference fluid temperature
 573 Θ = temperature on or inside the body
 574 $\bar{\Theta}$ = temperature outside the body in energy-forcing
 575 interpolation
 576 φ = reference angle
 577 **Subscripts and Superscripts**
 578 e = outer-boundary value
 579 i, j = indices for the Cartesian coordinates
 580 n = index for time step

581 References

- 582 [1] Peskin, C. S., 1972, "Flow Patterns Around Heart Valves: A Numerical
 583 Method," *J. Comput. Phys.*, **10**, pp. 252–271.
 584 [2] Peskin, C. S., 2002, "The Immersed Boundary Method," *Acta Numerica*, **11**,
 585 pp. 479–517.
 586 [3] Udaykumar, H. S., Mittal, R., and Rampunggoon, P., 2002, "Interface Tracking
 587 Finite Volume Method for Complex Solid-Fluid Interactions on Fixed
 588 Meshes," *Commun. Numer. Methods Eng.*, **18**, pp. 89–97.
 589 [4] Marella, S., Krishnan, S., Liu, H., and Udaykumar, H. S., 2005, "Sharp Inter-
 590 face Cartesian Grid Method I: An Easily Implemented Technique for 3D Mov-
 591 ing Boundary Computations," *J. Comput. Phys.*, **210**, pp. 1–31.
 592 [5] Liu, H., Krishnan, S., Marella, S., and Udaykumar, H. S., 2005, "Sharp Inter-
 593 face Cartesian Grid Method II: A Technique for Simulating Droplet Impact on
 594 Surfaces of Arbitrary Shape," *J. Comput. Phys.*, **210**, pp. 32–54.
 595 [6] Yang, Y., and Udaykumar, H. S., 2005, "Sharp Interface Cartesian Grid
 596 Method III: Solidification of Pure Materials and Binary Solutions," *J. Comput.*
 597 *Phys.*, **210**, pp. 55–74.
 598 [7] Lee, L., Li, Z., and LeVeque, R. J., 2003, "An Immersed Interface Method for
 599 Incompressible Navier-Stokes Equations," *SIAM J. Sci. Comput. (USA)*,
 600 **25**(3), pp. 832–856.
 601 [8] Barozzi, G. S., Bussi, C., and Corticelli, M. A., 2004, "A Fast Cartesian
 602 Scheme for Unsteady Heat Diffusion on Irregular Domains," *Numer. Heat*
 603 *Transfer, Part B*, **46**(1), pp. 59–77.
 604 [9] Fogelson, A. L., and Keener, J. P., 2001, "Immersed Interface Methods For
 605 Neumann and Related Problems in Two and Three Dimensions," *SIAM J. Sci.*
 606 *Comput. (USA)*, **22**(5), pp. 1630–1654.
 607 [10] Kim, J., Kim, D., and Choi, H., 2001, "An Immersed-Boundary Finite-Volume
 608 Method For Simulations Of Flow In Complex Geometries," *J. Comput. Phys.*,
 609 **171**, pp. 132–150.
 610 [11] Zhang, L., Gerstenberger, A., Wang, X., and Liu, W. K., 2004, "Immersed
 611 Finite Element Method," *Comput. Methods Appl. Mech. Eng.*, **193**, pp. 2051–
 612 2067.
 613 [12] Kim, J., and Choi, H., 2004, "An Immersed-Boundary Finite-Volume Method
 614 for Simulations of Heat Transfer in Complex Geometries," *KSME Int. J.*,
 615 **18**(6), pp. 1026–1035.
 616 [13] Pacheco, J. R., Pacheco-Vega, A., Rodic, T., and Peck, R. E., 2005, "Numerical
 617 Simulations of Heat Transfer and Fluid Flow Problems Using an
 618 Immersed-Boundary Finite-Volume Method on Nonstaggered Grids," *Numer.*
 619 *Heat Transfer, Part B*, **48**, pp. 1–24.
 620 [14] Fadlun, E. A., Verzicco, R., Orlandi, P., and Mohd-Yusof, J., 2000, "Combined
 621 Immersed-Boundary Finite-Difference Methods for Three-Dimensional Com-
 622 plex Flow Simulations," *J. Comput. Phys.*, **161**, pp. 35–60.
 623 [15] Mittal, R., and Iaccarino, G., 2005, "Immersed Boundary Methods," *Annu.*
 624 *Rev. Fluid Mech.*, **37**, pp. 239–261.
 625 [16] Tyagi, M., and Acharya, S., 2005, "Large Eddy Simulation of Turbulent Flows
 626 in Complex and Moving Rigid Geometries Using the Immersed Boundary
 627 Method," *Int. J. Numer. Methods Fluids*, **48**, pp. 691–722.
 628 [17] Deuschard, P., and Hochmuth, R., 2004, "Multiscale Analysis of Thermoregu-
 629 lation in the Human Microvascular System," *Math. Methods Appl. Sci.*, **27**,
 630 pp. 971–989.
 631 [18] Oyanader, M. A., Arce, P., and Dzurik, A., 2003, "Avoiding Pitfalls in Elec-
 632 trokinetic Remediation: Robust Design and Operation Criteria Based on First
 633 Principles For Maximizing Performance in a Rectangular Geometry," *Electro-*
 634 *phoresis*, **24**(19–20), pp. 3457–3466.
- [19] DeLima-Silva, W., Jr., and Wrobel, L. C., 1995, "A Front-Tracking BEM For-
 mulation for One-Phase Solidification/Melting Problems. Engineering Analy-
 sis With Boundary Elements," *Eng. Anal. Boundary Elem.*, **16**, pp. 171–182.
 [20] Amaziane, B., and Pankratov, L., 2006, "Homogenization Of A Reaction-
 Diffusion Equation With Robin Interface Conditions," *Appl. Math. Lett.*,
19(11), pp. 1175–1179.
 [21] Sparrow, E. M., and Gregg, J. L., 1958, "Similar Solutions for Free Convec-
 tion From a Non-Isothermal Vertical Plate," *ASME J. Heat Transfer*, **80**, pp.
 379–386.
 [22] Leonard, B. P., 1979, "A Stable And Accurate Convective Modelling Proce-
 dure Based on Quadratic Upstream Interpolation," *Comput. Methods Appl.*
Mech. Eng., **19**, pp. 59–98.
 [23] Zang, Y., Street, R. L., and Koseff, J. F., 1994, "A Non-Staggered Grid, Frac-
 tional Step Method for Time Dependent Incompressible Navier-Stokes Equa-
 tions in Curvilinear Coordinates," *J. Comput. Phys.*, **114**, pp. 18–33.
 [24] Pacheco, J. R., 1999, "On the Numerical Solution of Film and Jet Flows,"
 Ph.D thesis, Department Mechanical and Aerospace Engineering, Arizona
 State University, Tempe.
 [25] Pacheco, J. R., and Peck, R. E., 2000, "Non-Staggered Grid Boundary-Fitted
 Coordinate Method for Free Surface Flows," *Numer. Heat Transfer, Part B*,
37, pp. 267–291.
 [26] Pacheco, J. R., 2001, "The Solution of Viscous Incompressible Jet Flows Us-
 ing Non-Staggered Boundary Fitted Coordinate Methods," *Int. J. Numer.*
Methods Fluids, **35**, pp. 71–91.
 [27] Mohd-Yusof, J., 1997, "Combined Immersed-Boundary/B-Spline Methods For
 Simulations Of Flow In Complex Geometries," NASA Ames/Stanford Univer-
 sity Technical Report **■**.
 [28] Orlanski, I., 1976, "A Simple Boundary Condition For Unbounded Hyperbolic
 Flows," *J. Comput. Phys.*, **21**, pp. 251–269.
 [29] Eckert, E. R. G., and Soehngen, E., 1952, "Distribution of Heat-Transfer Co-
 efficients Around Circular Cylinders in Crossflow At Reynolds Numbers From
 20 to 500," *Trans. ASME*, **75**, pp. 343–347.
 [30] Pan, D., 2006, "An Immersed Boundary Method on Unstructured Cartesian
 Meshes for Incompressible Flows With Heat Transfer," *Numer. Heat Transfer,*
Part B, **49**, pp. 277–297.
 [31] Lima E Silva, A. L. F., Silveira-Neto, A., and Damasceno, J. J. R., 2003,
 "Numerical Simulation of Two Dimensional Flows over a Circular Cylinder
 Using the Immersed Boundary Method," *J. Comput. Phys.*, **189**, pp. 351–370.
 [32] Strykowski, P. J., and Sreenivasan, K. R., 1990, "On the Formation and Sup-
 pression of Vortex Shedding at Low Reynolds Numbers," *J. Fluid Mech.*, **218**,
 pp. 71–107.
 [33] Moukalled, F., and Darwish, M., 1997, "New Bounded Skew Central Differ-
 ence Scheme, Part II: Application to Natural Convection in an Eccentric Annu-
 lus," *Numer. Heat Transfer, Part B*, **31**, pp. 111–133.
 [34] Sadat, H., and Couturier, S., "Performance and Accuracy of a Meshless
 Method for Laminar Natural Convection," *Numer. Heat Transfer, Part B*, **37**,
 pp. 455–467.
 [35] Demirdžić, I., Lilek, Ž., and Perić, M., 1992, "Fluid Flow and Heat Transfer
 Test Problems for Nonorthogonal Grids: Bench-Mark Solutions," *Int. J. Numer.*
Methods Fluids, **15**, pp. 329–354.
 [36] Carslaw, H. S., and Jaeger, J. C., 1986, *Conduction of Heat in Solids*, 2nd ed.,
 Oxford University Press, Oxford.
 [37] Özisik, M. N., 1989, *Boundary Value Problems of Heat Conduction*, Dover,
 Mineola, NY.
 [38] Johnson, T. A., and Patel, V. C., 1999, "Flow Past a Sphere up to Reynolds
 Number of 300," *J. Fluid Mech.*, **378**, pp. 19–70.
 [39] Magarvey, R. H., and Bishop, R. L., 1961, "Transition Ranges for Three-
 Dimensional Wakes," *Can. J. Phys.*, **39**, pp. 1418–1422.
 [40] Feng, Z.-G., and Michaelides, E. E., 2000, "A Numerical Study on the Tran-
 sient Heat Transfer From a Sphere at High Reynolds and Peclet Numbers," *Int.*
J. Heat Mass Transfer, **43**, pp. 219–229.
 [41] Hunt, J. C. R., Wray, A. A., and Moin, P., 1988, "Eddies, Stream, and Con-
 vergence Zones in Turbulent Flows," Center for Turbulence Research Techni-
 cal Report No. CTR-S88.
 [42] Constantinescu, S. G., and Squires, K. D., 2004, "Numerical Investigation of
 the Flow Over a Sphere in the Subcritical and Supercritical Regimes," *Phys.*
Fluids, **16**(5), pp. 1449–1467.
 [43] Jeong, J., and Hussain, F., 1995, "On the Identification of a Vortex," *J. Fluid*
Mech., **285**, pp. 69–94.

AUTHOR QUERIES — 009711JHR

#1 AU: PLS. UPDATE REFS. 27 & 41.

PROOF COPY [HT-06-1248] 009711JHR