AN IMMERSED-BOUNDARY METHOD FOR
COMPRESSIBLE VISCOUS FLOWS

P. De Palma*, M. de Tullio*, G. Pascazio*, and M. Napolitano*

*DIMeG & CEMeC
Politecnico di Bari, via Re David 200, 70125, Bari, Italy
e-mails: depalma@poliba.it, detullio@imedado.poliba.it, pascazio@poliba.it, napolita@poliba.it

Key words: immersed-boundary technique, preconditioned equations, dual-time-stepping, alternating direction implicit

Abstract. This paper combines a state-of-the-art method for solving the preconditioned compressible Navier–Stokes equations accurately and efficiently for a wide range of the Mach number with an immersed-boundary approach which allows to use Cartesian grids for arbitrarily complex geometries. The method is validated versus well documented test problems for a wide range of the Reynolds and Mach numbers. The numerical results demonstrate the efficiency and versatility of the proposed approach as well as its accuracy, from incompressible to supersonic flow conditions, for moderate values of the Reynolds number. Further improvements, obtained via local grid refinement or nonlinear wall functions, can render the proposed approach a formidable tool for studying complex three-dimensional flows of industrial interest.
1 INTRODUCTION

Many fluid dynamic problems of engineering interest exhibit flow regions with very different Mach numbers, which render their numerical simulation very difficult. In fact, due to the different nature of the physical phenomena associated with flows at different Mach numbers, usually a single numerical method performs accurately and efficiently within a limited range of the Mach number. Moreover, the presence of complex and/or moving boundaries usually requires time consuming body-fitted grid generations. The aim of the present work is to remedy both of the aforementioned difficulties, by combining a state-of-the-art method for solving the preconditioned compressible Navier–Stokes equations accurately and efficiently for a wide range of the Mach number with an Immersed-Boundary (IB) approach which allows to use Cartesian grids for arbitrarily complex geometries. Concerning the preconditioning of the governing equations, the residual of the compressible Navier–Stokes or Reynolds-Averaged Navier–Stokes (RANS) equations is premultiplied by a suitable matrix which uniformizes the wave propagation speeds, thus greatly enhancing the accuracy and efficiency of the compressible flow solver when applied to low-Mach-number flows. Such a preconditioning technique, originally designed for steady flows [1, 2], has been extended to the unsteady ones using a Dual-Time-Stepping (DTS) technique with a three-level backward discretization of the time derivative, in conjunction with a third-order-accurate finite volume method based on flux-vector splitting for the convective terms [3, 4]. The IB technique, originally designed for incompressible flows [5, 6], allows the body surface to cut the computational cells, so that a simple Cartesian grid can be employed, independently of the complexity of the considered geometry. In this work, the IB technique has been extended to the preconditioned compressible Navier–Stokes and RANS equations to provide an accurate, efficient and versatile tool for studying complex three-dimensional flows of industrial interest. Here the proposed method is validated versus two-dimensional flows for a very wide range of the Reynolds and Mach numbers. In the following sections, a brief review of the governing equations and their solution technique is given at first, then, the IB technique is described and, finally, results are provided and compared versus numerical and experimental ones available in the literature.

2 GOVERNING EQUATIONS AND NUMERICAL METHOD

The Reynolds Averaged Navier–Stokes (RANS) equations, written in terms of Favre mass-averaged quantities and using the standard $k - \omega$ turbulence model, can be written as follows:

\[
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_j) = 0,
\]

\[
\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_i u_j) = - \frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ji}}{\partial x_j}.
\]
\[ \frac{\partial (\rho U)}{\partial t} + \frac{\partial (\rho u_j H)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ u_i \tilde{\tau}_{ij} + (\mu + \sigma^* \mu_t) \frac{\partial k}{\partial x_j} - q_j \right], \quad (3) \]

\[ \frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho u_j k)}{\partial x_j} = \tau_{ij} \frac{\partial u_i}{\partial x_j} - \beta^* \rho \omega k + \frac{\partial}{\partial x_j} \left[ (\mu + \sigma^* \mu_t) \frac{\partial k}{\partial x_j} \right], \quad (4) \]

\[ \frac{\partial (\rho \omega)}{\partial t} + \frac{\partial (\rho u_j \omega)}{\partial x_j} = \frac{\gamma \omega}{k} \tilde{\tau}_{ij} \frac{\partial u_i}{\partial x_j} - \beta^* \rho \omega^2 + \frac{\partial}{\partial x_j} \left[ (\mu + \sigma^* \mu_t) \frac{\partial \omega}{\partial x_j} \right]. \quad (5) \]

In the equations above, \( U \) and \( H \) are the total energy and enthalpy comprehensive of the turbulent kinetic energy, \( k \); the eddy viscosity, \( \mu_t \), is defined in terms of \( k \) and of the specific dissipation rate, \( \omega \), according to the \( k-\omega \) turbulence model of Wilcox [7], namely:

\[ \mu_t = \gamma^* \frac{\rho k}{\omega}. \quad (6) \]

Moreover, \( \tilde{\tau}_{ij} \) indicate the sum of the molecular and Reynolds (\( \tau_{ij} \)) stress tensor components. According to the Boussinesq approximation, one has:

\[ \tilde{\tau}_{ij} = (\mu + \mu_t) \left[ \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \right] - \frac{2}{3} \rho \frac{k}{\omega} \delta_{ij}. \quad (7) \]

Finally, the heat flux vector components, \( q_j \), are given as:

\[ q_j = - \left( \frac{\mu}{Pr} + \frac{\mu_t}{Pr_t} \right) \frac{\partial h}{\partial x_j}, \quad (8) \]

where \( Pr = 0.71 \) and \( Pr_t = 1 \) are the laminar and turbulent Prandtl numbers, respectively. The Sutherland law is used to compute the molecular viscosity coefficient. Finally, the standard coefficients of the turbulence \( k-\omega \) model are used [7], namely:

\[ \beta = \frac{3}{40}, \quad \beta^* = \frac{9}{100}, \quad \gamma = \frac{5}{9}, \quad \gamma^* = 1, \quad \sigma = \sigma^* = \frac{1}{2}. \]

The numerical method employed to solve the RANS equation is described in the following with reference to the two-dimensional case, for simplicity. The system of equations is written either in Cartesian or generalized curvilinear coordinates, \( (\xi, \eta) \); a pseudo-time derivative is added to the left-hand-side in order to use a time marching approach for both steady state and unsteady problems; the preconditioning matrix, \( \Gamma \), proposed in [1, 2] is finally used to premultiply the pseudo-time derivative in order to improve efficiency. The final system reads:

\[ \Gamma \frac{\partial Q_v}{\partial \tau} + \frac{\partial Q_v}{\partial t} + \frac{\partial E_v}{\partial \xi} + \frac{\partial F_v}{\partial \eta} - \frac{\partial E_v}{\partial \xi} - \frac{\partial F_v}{\partial \eta} = D, \quad (9) \]

where \( Q \) is the conservative variable vector, \( E, F, \) and \( E_v, F_v \) indicate the inviscid and viscous fluxes, respectively, \( D \) is the vector of the source terms for the turbulence equations, and \( Q_v = (p, u, v, T, k, \omega)^T \) is the primitive variable vector, which is related to \( Q \).
by the Jacobian \( P = \partial Q / \partial Q_v \). Discretizing equation (9) by an Euler implicit scheme in the pseudo-time and approximating the physical-time derivative by second-order-accurate three-point backward differences, the following equation in delta form is obtained:

\[
\left[ \Gamma + \frac{3 \Delta \tau}{2 \Delta t} P + \Delta \tau \frac{\partial}{\partial \xi} \left( A_v - R_{\xi \xi} \frac{\partial}{\partial \xi} - R_{\eta \eta} \frac{\partial}{\partial \eta} \right) \right.
+ \Delta \tau \frac{\partial}{\partial \eta} \left( B_v - R_{\eta \eta} \xi \frac{\partial}{\partial \eta} - R_{\eta \eta} \eta \xi \frac{\partial}{\partial \xi} \right) \Delta Q_v
\]

\[
= -\Delta \tau \left[ \frac{3 Q^r - 4 Q^n + Q^{n-1}}{2 \Delta t} + \mathcal{R}^r \right],
\]

where \( r \) and \( \Delta \tau \) indicate the pseudo-time level and step, \( n \) and \( \Delta t \) indicate the physical-time level and step, \( A_v = \partial E / \partial Q_v \), \( B_v = \partial F / \partial Q_v \), \( R_{ij} \) are the viscous coefficient matrices [8], and the matrix \( \Gamma \) is evaluated as proposed in [3, 4]. The residual is given as:

\[
\mathcal{R}^r = \frac{\partial (E^r - E^r_v)}{\partial \xi} + \frac{\partial (F^r - F^r_v)}{\partial \eta} - D^r,
\]

and the delta unknowns to be annihilated at every physical-time level are

\[
\Delta Q_v = Q_v^{r+1} - Q_v^r.
\]

The left-hand-side (LHS) of equation (10) is modified to improve the efficiency of the method, without affecting the residual, that is, the physical solution. Firstly, the non-orthogonal viscous coefficient matrices, \( R_{\xi \xi} \) and \( R_{\eta \eta} \), are neglected, and the remaining ones are approximated by the corresponding spectral radii multiplied times the identity matrix, \( R_{\xi \xi} = R_{\xi} \) and \( R_{\eta \eta} = R_{\eta} \); then, as proposed in [8], the pseudo- and physical-time terms are grouped together into a new term \( S \),

\[
S = \Gamma + \frac{3 \Delta \tau}{2 \Delta t} P,
\]

which is factored out of the LHS in equation (10), yielding:

\[
S \left[ I + \Delta \tau S^{-1} \frac{\partial}{\partial \xi} \left( A_v - R_{\xi \xi} \frac{\partial}{\partial \xi} \right) + \Delta \tau S^{-1} \frac{\partial}{\partial \eta} \left( B_v - R_{\eta \eta} \frac{\partial}{\partial \eta} \right) \right] \Delta Q_v
= -\Delta \tau \left[ \frac{3 Q^r - 4 Q^n + Q^{n-1}}{2 \Delta t} + \mathcal{R}^r \right].
\]

In order to solve the resulting linear system, the diagonalization procedure of Pulliam and Chausee [9] is firstly applied, so that the matrices \( S^{-1} A_v \) and \( S^{-1} B_v \) can be written as:

\[
S^{-1} A_v = M_{\xi} A_{\xi} M^{-1}_{\xi}, \quad S^{-1} B_v = M_{\eta} A_{\eta} M^{-1}_{\eta},
\]
where $M_\xi$, $M_\eta$ are the right-eigenvector matrices, $M_\xi^{-1}$, $M_\eta^{-1}$ are the left-eigenvector matrices; and $\Lambda_\xi$ and $\Lambda_\eta$ are diagonal matrices containing the eigenvalues of $S^{-1}A_v$ and $S^{-1}B_v$, respectively; then, the LHS of equation (14) is factorized,

$$SM_\xi \left[ I + \Delta \tau \frac{\partial}{\partial \xi} \left( \Lambda_\xi - R_\xi I \frac{\partial}{\partial \xi} \right) \right] M_\xi^{-1}M_\eta \left[ I + \Delta \tau \frac{\partial}{\partial \eta} \left( \Lambda_\eta - R_\eta I \frac{\partial}{\partial \eta} \right) \right] M_\eta^{-1} \Delta Q_v =$$

$$-\Delta \tau \left[ \frac{3Q^r - 4Q^n + Q^{n-1}}{2\Delta t} + R^r \right],$$

and solved by a standard scalar alternating direction implicit procedure [4]. A cell-centred finite volume space discretization is used on a multi-block structured mesh. A third-order-accurate flux vector splitting scheme is employed to discretize the convective terms, the minmod limiter being applied in the presence of shocks, whereas the viscous terms are discretized by second-order-accurate central differences. Further details of the method can be found in reference [8], which is available on the web [10], together with the code developed at the Pennsylvania State University.

### 3 IMMERSED-BOUNDARY TECHNIQUE

The Immersed Boundary (IB) technique used in this work is based on that proposed in [5] and [6]. In a preliminary step, the geometry under consideration, which is described by a closed polygon in two dimensions (a closed surface in three dimensions), is overlapped onto a Cartesian (non uniform) grid. Using the ray tracing technique based on the geometrical algorithms reported in [11], the computational cells occupied entirely by the flow are tagged as internal cells; those whose centres lie within the immersed body are tagged as external cells; the remaining ones are finally tagged as interface cells. The main feature of the IB technique is the evaluation of the unknowns at the centres of the interface cells. Here, the direct forcing method proposed by Mohd-Yusof [12] is employed.

For each interface cell, the shortest Cartesian distance between the cell centre and the solid wall is determined. Along the corresponding direction, see figure 1, the variables at the centre of the interface cell (point $P$) are linearly interpolated between the values to be imposed at the boundary point (point $B$) and the computed values at the neighbouring internal-cell centre (point $A$), except for the pressure, whose value at point $P$ is set equal to that at point $A$, which amounts to imposing a first-order-accurate homogeneous Neumann condition for the pressure. In the present work, Dirichlet boundary conditions are imposed to: $u_1$ and $u_2$ (the two Cartesian velocity components), $T$ (the temperature) and, for turbulent flows, $k$ and $\omega$. As shown in [5], such an approach provides an essentially second-order-accurate solution. It is noteworthy that, for high Reynolds number flows, a non linear interpolation procedure which utilizes adaptive wall functions is needed to obtain accurate results. Needless to say, the governing equations are solved at all internal cell centres, whereas all unknowns are set to zero at external ones.
4 RESULTS

The proposed methodology has been applied to compute two-dimensional steady and unsteady flows, for a wide range of the Reynolds and Mach numbers.

4.1 Incompressible flow past a circular cylinder

The two-dimensional incompressible flow past a circular cylinder has been considered at first to test both the preconditioning strategy and the immersed-boundary method versus steady as well as unsteady flows at very low Mach numbers. A single value of the free-stream Mach number, \( M_\infty = 0.03 \), and four values of the Reynolds number, based on the cylinder diameter, \( D \), the free-stream velocity, \( U_\infty \), and kinematic viscosity, \( \nu_\infty \), namely, 20, 40, 100, and 200, have been considered; the first two cases correspond to steady flow regimes and the last ones to unsteady ones [14]. A rectangular computational domain has been used with the inlet and outlet vertical boundaries located at \( x_i = -10D \) and \( x_o = 40D \) and the two horizontal boundaries located at \( y_f = \pm 20D \), respectively, the origin coinciding with the centre of the cylinder. Standard characteristic boundary conditions have been imposed at inlet and outlet points, whereas free-shear wall boundary conditions are imposed at the points on the far-field horizontal boundaries. Computations have been performed using a structured, non uniform, single-block grid with \( 244 \times 168 \) cells. A local view of the mesh is given in figure 2. Figure 3 shows the steady streamlines corresponding to \( Re = 20 \) and \( Re = 40 \), respectively. Finally, the computed geometrical properties of the symmetrical vortices, as defined in figure 4, and the drag coefficient, \( C_D \), are provided in tables 1 and 2, together with the corresponding experimental [13, 14] and numerical [15, 16, 17] results available in the literature. The agreement is quite sat-
isfactory. For both computations, the steady solver has been used, which annihilates the physical time derivative and iterates till the steady residual is reduced to $10^{-6}$. Convergence is achieved within about 18220 iterations, corresponding to 5160 CPU seconds on a single processor Pentium IV (2.66 GHz). Concerning the $Re = 100$ and $Re = 200$

![Figure 2: Local view of the grid.](image)

| Fornberg [15] | 0.91 | - | - | 45.7° | 2.00 |
| Dennis [16] | 0.94 | - | - | 43.7° | 2.05 |
| Coutanceau [13] | 0.93 | 0.33 | 0.46 | 45.0° | - |
| Tritton [14] | - | - | - | - | 2.09 |
| Linnick [17] | 0.93 | 0.36 | 0.43 | 43.5° | 2.06 |
| present | 0.93 | 0.36 | 0.43 | 44.6° | 2.05 |

Table 1: Steady flow past a circular cylinder at $Re = 20$.

unsteady flow computations, the non-dimensional physical time step has been set equal to 0.03, which corresponds to about 200 steps per period. About 200 inner iterations are needed to reduce the unsteady residual to $10^{-6}$ at every physical time step, corresponding to about 63 CPU seconds on the aforementioned processor. Two snapshots of the vorticity contours are given in figure 5; in both cases, the lift and drag coefficients have regular sinusoidal behavior in time, as shown in figure 6 for $Re = 200$. Finally, the computed Strouhal number based on the shedding frequency, $f$ ($St = fD/U_\infty$), as well as the drag and lift coefficients are given in tables 3 and 4 together with the experimental [18] and numerical [17, 19, 20] results available in the literature. Also for these unsteady flow cases, a very good agreement is obtained.
Figure 3: Steady streamlines for $Re = 20$ (left) and $Re = 40$ (right).

Figure 4: Definitions of the relevant geometrical parameters of the symmetric separation region behind the cylinder.

Figure 5: Snapshot of the vorticity contours for $Re = 100$ (left) and $Re = 200$ (right).
Table 2: Steady flow past a circular cylinder at $Re = 40$.

<table>
<thead>
<tr>
<th></th>
<th>$L$</th>
<th>$a$</th>
<th>$b$</th>
<th>$\theta$</th>
<th>$C_D$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fornberg</td>
<td>2.24</td>
<td>-</td>
<td>-</td>
<td>55.6°</td>
<td>1.50</td>
</tr>
<tr>
<td>Dennis</td>
<td>2.35</td>
<td>-</td>
<td>-</td>
<td>53.8°</td>
<td>1.52</td>
</tr>
<tr>
<td>Coutanceau</td>
<td>2.13</td>
<td>0.76</td>
<td>0.59</td>
<td>53.8°</td>
<td>-</td>
</tr>
<tr>
<td>Tritton</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>1.59</td>
</tr>
<tr>
<td>Linnick</td>
<td>2.28</td>
<td>0.72</td>
<td>0.60</td>
<td>53.6°</td>
<td>1.54</td>
</tr>
<tr>
<td>present</td>
<td>2.28</td>
<td>0.72</td>
<td>0.60</td>
<td>53.8°</td>
<td>1.55</td>
</tr>
</tbody>
</table>

Table 3: Unsteady flow past a circular cylinder at $Re = 100$.

<table>
<thead>
<tr>
<th></th>
<th>$St$</th>
<th>$C_D$</th>
<th>$C_L$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Berger</td>
<td>0.16</td>
<td>0.17</td>
<td>-</td>
</tr>
<tr>
<td>Liu</td>
<td>0.165</td>
<td>1.35</td>
<td>±0.012</td>
</tr>
<tr>
<td>Linnick</td>
<td>0.166</td>
<td>1.34</td>
<td>±0.009</td>
</tr>
<tr>
<td>present</td>
<td>0.163</td>
<td>1.32</td>
<td>±0.01</td>
</tr>
</tbody>
</table>

4.2 Unsteady flow past a heated circular cylinder

The unsteady very low-Mach number flow past a heated circular cylinder has been chosen in order to validate the proposed method for a flow in which the energy equation plays a significant role, insofar as experimental [21] and numerical [23] investigations indicate that the temperature field has a significant influence on the flow pattern, especially when the ratio between the cylinder wall temperature, $T_w$, and the free-stream one, $T_\infty (T^* = T_w/T_\infty)$ exceeds 1.1. In particular, it has been found that, for a given $Re_\infty$, the vortex shedding frequency, $f$, i.e., the Strouhal number, $St$, decreases for increasing values of $T^*$. Furthermore, by defining an effective Reynolds number, $Re_{eff}$, in terms
of the kinematic viscosity corresponding to the maximum temperature in the wake, $T_{\text{eff}}$
(which happens to be at the centre of the first vortex shed by the cylinder, see figure 10), the curves $St$ versus $Re_{\text{eff}}$, obtained for different values of $T^*$ in the range $1 \leq T^* \leq 2$, collapse into a single curve; see reference [21], where the following empirical correlation was proposed and employed to compute $T_{\text{eff}}$:

$$T_{\text{eff}} = T_{\infty} + 0.28(T_w - T_{\infty}).$$

(17)

In the present work, the same rectangular domain employed in the previous test-case has been used and discretized with a much finer (non uniform) multi-block Cartesian grid with 142200 cells and 12 blocks, see figure 7, where a local view of all blocks is provided (only one every five grid-lines is plotted). Computations have been performed on a cluster with 12 processors for $M_{\infty} = 0.01$ and several values of $Re$ and $T^*$. The non-dimensional time step has been set equal to 0.012, which corresponds to about 500 steps per period. About 200 inner iterations are needed to reduce the unsteady residual to $10^{-8}$ at every physical time step, corresponding to about 55 CPU seconds. The computed values of $St$ for $Re_{\infty} = 80, 100, 120, 140$ and $T^* = 1, 1.1, 1.5, 1.8$ are provided in figure 8, together with the experimental results of [21]: a very good agreement is obtained. Then, numerical results have been obtained for a wider range of $T^*$, for $Re_{\infty} = 140$ and 260. The computed values of $St$ versus $Re_{\text{eff}}$ are given in figure 9, together with those already shown in figure 8. Finally, table 5 provides the values of $St$ and $T_{\text{eff}}$ computed either at the centre of the first shed vortex, $T_{\text{eff}}^{(1)}$, or using equation (17), $T_{\text{eff}}^{(2)}$. The present results confirm the experimental data of [21], together with the validity of equation (17) in the range $1 \leq T^* \leq 2$; moreover, they show that for $Re_{\infty} = 140$ and $T^* = 3.5$ the flow becomes steady and for $Re_{\infty} = 260$ and $T^* \geq 2.5$ the values of $St$ do not fit the “universal” curve $St(Re_{\text{eff}})$, probably because a change in the physical nature of the unsteady phenomenon occurs at $140 < Re < 260$. Notice that for $Re_{\infty} = 260$ the values corresponding to $Re_{\text{eff}} > 160$ are not plotted insofar as they lie outside the range of laminar periodic wakes (they fall in the A-mode transition region for the case of unheated cylinders). Finally, for the two $Re_{\infty} = 260$ cases with $T^* = 1.6$ and $T^* = 3$, the computed

<table>
<thead>
<tr>
<th></th>
<th>$St$</th>
<th>$C_D$</th>
<th>$C_L$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Berger</td>
<td>0.18-0.19</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Belov</td>
<td>0.193</td>
<td>1.19 ± 0.042</td>
<td>±0.64</td>
</tr>
<tr>
<td>Rogers</td>
<td>0.185</td>
<td>1.23 ± 0.050</td>
<td>±0.65</td>
</tr>
<tr>
<td>Miyake</td>
<td>0.196</td>
<td>1.34 ± 0.043</td>
<td>±0.67</td>
</tr>
<tr>
<td>Liu</td>
<td>0.192</td>
<td>1.31 ± 0.049</td>
<td>±0.69</td>
</tr>
<tr>
<td>Linnick</td>
<td>0.197</td>
<td>1.34 ± 0.044</td>
<td>±0.69</td>
</tr>
<tr>
<td>present</td>
<td>0.190</td>
<td>1.34 ± 0.045</td>
<td>±0.68</td>
</tr>
</tbody>
</table>

Table 4: Unsteady flow past a circular cylinder at $Re = 200$. 

<table>
<thead>
<tr>
<th>$St$</th>
<th>$C_D$</th>
<th>$C_L$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.18-0.19</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>0.193</td>
<td>1.19 ± 0.042</td>
<td>±0.64</td>
</tr>
<tr>
<td>0.185</td>
<td>1.23 ± 0.050</td>
<td>±0.65</td>
</tr>
<tr>
<td>0.196</td>
<td>1.34 ± 0.043</td>
<td>±0.67</td>
</tr>
<tr>
<td>0.192</td>
<td>1.31 ± 0.049</td>
<td>±0.69</td>
</tr>
<tr>
<td>0.197</td>
<td>1.34 ± 0.044</td>
<td>±0.69</td>
</tr>
<tr>
<td>0.190</td>
<td>1.34 ± 0.045</td>
<td>±0.68</td>
</tr>
</tbody>
</table>
Figure 7: Local view of the multi-block grid.

Figure 8: Strouhal number vs Reynolds number for flow past heated cylinder: comparison between experimental (open symbols) and numerical (solid symbols) data.
values of $T_{\text{eff}}$ are in good agreement with the experimental data of [22] for the maximum temperature in the wake, namely, $T_{\text{max}} = 343.5$ K for $T^* = 1.61$ and $T_{\text{max}} = 484.1$ K for $T^* = 2.98$.

### 4.3 Supersonic flow past an NACA0012 airfoil

In order to test the proposed methodology versus a well documented viscous flow at high Mach number, the laminar supersonic flow past an NACA0012 airfoil with $M_\infty = 2$, $\alpha = 10^\circ$ and $Re_{\infty} = 1000$ has been considered [24]. Three grids with $125^2$, $250^2$, and $500^2$ cells have been used to discretize the computational domain $[-8c; 9c] \times [-8c; 8c]$, $c$ being the chord-length of the airfoil, whose leading edge is located at the origin; figure 11 shows a local view of the finest mesh (only one every five grid lines is plotted), which is partitioned into eighteen blocks, for parallel computing. A benchmark solution has also been obtained for comparison purposes, using the present approach on a very fine body-fitted grid ($87500$ cells with twelve blocks). The pressure coefficient distributions along the profile are given in figure 12 where the finest grid solution, which coincides with the aforementioned benchmark one, within plotting accuracy, is clearly seen to be grid-converged. The lift and drag coefficients obtained on the three grids are equal to 0.3296, 0.3335, 0.3353, and 0.2448, 0.2485, 0.2514, respectively, which tend towards the values of 0.3400 and 0.2515, obtained from the aforementioned benchmark solution, as the mesh is refined. Finally, the Mach number contours computed on the finest grid
Table 5: Temperatures in K. (1) present computations, (2) equation (17).

<table>
<thead>
<tr>
<th>$T^*$</th>
<th>$T_{eff}^{(1)}$</th>
<th>$T_{eff}^{(2)}$</th>
<th>$St^{(1)}$</th>
<th>$T^*$</th>
<th>$T_{eff}^{(1)}$</th>
<th>$T_{eff}^{(2)}$</th>
<th>$St^{(1)}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.1</td>
<td>303.5</td>
<td>303.3</td>
<td>0.178</td>
<td>1.1</td>
<td>303.7</td>
<td>303.3</td>
<td>0.191</td>
</tr>
<tr>
<td>1.4</td>
<td>329.6</td>
<td>328.0</td>
<td>0.170</td>
<td>1.4</td>
<td>329.4</td>
<td>328.0</td>
<td>0.189</td>
</tr>
<tr>
<td>1.6</td>
<td>345.1</td>
<td>344.6</td>
<td>0.167</td>
<td>1.6</td>
<td>346.2</td>
<td>344.6</td>
<td>0.188</td>
</tr>
<tr>
<td>1.8</td>
<td>364.2</td>
<td>361.1</td>
<td>0.163</td>
<td>1.8</td>
<td>364.4</td>
<td>361.1</td>
<td>0.187</td>
</tr>
<tr>
<td>2.0</td>
<td>382.2</td>
<td>377.6</td>
<td>0.161</td>
<td>2.0</td>
<td>381.5</td>
<td>377.6</td>
<td>0.185</td>
</tr>
<tr>
<td>2.5</td>
<td>422.5</td>
<td>418.9</td>
<td>0.150</td>
<td>2.5</td>
<td>423.9</td>
<td>418.9</td>
<td>0.183</td>
</tr>
<tr>
<td>3.0</td>
<td>476.2</td>
<td>460.2</td>
<td>0.140</td>
<td>3.0</td>
<td>475.0</td>
<td>460.2</td>
<td>0.177</td>
</tr>
<tr>
<td>3.5</td>
<td>-</td>
<td>501.5</td>
<td>-</td>
<td>3.5</td>
<td>520.0</td>
<td>501.5</td>
<td>0.171</td>
</tr>
</tbody>
</table>

are provided in figure 13 showing that the shock is computed monotonically. Using the steady solver, reducing the residual to $10^{-6}$ on the finest grid requires about $10^5$ iterations, corresponding to $2 \times 10^5$ CPU seconds on a single processor Pentium IV (2.66 GHz).

Two comments are in order. All of the lift and drag coefficients obtained from computation using the IB approach and Cartesian grids are computed performing a momentum balance of the fluid comprised within a rectangle surrounding the body. It has been verified that by varying the rectangle dimensions, the computed results vary less than one tenth of one percent.

In the heated cylinder case, the block containing the cylinder contains about one fifth of the total number of cells; in the present flow case, the eighteen blocks all contain about the same number of cells. Accordingly, the CPU time per iteration and gridpoint required by parallel computation is five and eighteen times lower than that required by a single processor calculation, respectively.

### 4.4 Supersonic flow past a circular cylinder

Finally, supersonic turbulent steady flows past a circular cylinder at $Re_\infty = 2 \times 10^5$ and $M_\infty = 1.3$ and 1.7, have been considered as formidable test-cases for the proposed IB method The inlet values of the turbulence kinetic energy and specific dissipation rate are $k/U_\infty^2 = 0.0009$ and $\omega D/U_\infty = 40$, respectively. For such values of $M_\infty$ a bow shock is formed upstream of the cylinder; the subsonic flow at the front part close to the wall accelerates along the surface forming a supersonic-flow region enveloping the subsonic recirculation region behind the cylinder; and two tail shocks are formed at the end of the separation region. Preliminary results have been obtained using a rectangular domain discretized by a single-block, rather coarse non uniform grid with $430 \times 200$ cells. Figures 14 and 15 provide a local view of the Mach number contours around the cylinder.

It is noteworthy that the length of the recirculation region decreases when increasing the Mach number. The computed positions of the separation points, $\theta_s$, where $\theta$ is the
Figure 10: Snapshot of the temperature contours for $T^* = 2.0$ and $Re_\infty = 260$: the point at which $T_{eff}$ has been evaluated is indicated.

Figure 11: Local view of the mesh for the flow past an NACA0012 airfoil.
Figure 12: Pressure coefficient distributions along the NACA0012 profile.

Figure 13: Mach number contours ($\Delta M = 0.1$).
Figure 14: Local view of the Mach number contours for \( M_\infty = 1.3 \) (\( \Delta M = 0.08 \)).

Figure 15: Local view of the Mach number contours for \( M_\infty = 1.7 \) (\( \Delta M = 0.08 \)).
Figure 16: Pressure coefficient distribution along the surface of the cylinder: comparison between experimental and numerical results for $M_\infty = 1.3$.

clockwise angle measured with respect to the horizontal direction starting from the leading edge, are $\theta_s = 107^\circ$ and $\theta_s = 118^\circ$ for $M_\infty = 1.3$ and $M_\infty = 1.7$, respectively. In spite of the low resolution of the boundary layer, such values are in satisfactory agreement with the corresponding experimental data provided in [25], namely, $103^\circ$ and $112^\circ$, respectively. Furthermore, the two values of the computed drag coefficient, 1.46 and 1.38, again agree reasonably well with the experimental data of [25], 1.48 and 1.43. Finally, the computed pressure coefficient distributions along the surface of the cylinder are provided in figures 16 and 17 together with the experimental data of [25]. The two sets of data are in good agreement in the front part of the surface, whereas the solution computed by the present IB method provides major discrepancies in the aft-body separation region due to the lack of resolution of the boundary layer.

5 CONCLUSIONS

This paper has combined a state-of-the-art method for solving the preconditioned compressible Navier–Stokes equations accurately and efficiently for a wide range of the Mach number with an immersed-boundary approach which allows to use Cartesian grids for arbitrarily complex geometries. The methodology has been applied to compute steady and unsteady flows past circular cylinders and an NACA0012 airfoil for a wide range of the Reynolds and Mach numbers demonstrating its versatility as well as its accuracy for moderate values of the Reynolds number. For high-Reynolds number flows the accuracy of the proposed approach remains to be assessed. A local refinement strategy or non-linear wall-laws, which can mimic an accurate resolution of the viscous sub-layer, may
be required to obtain a state-of-the-art tool for investigating three-dimensional flows of industrial interest.

6 ACKNOWLEDGMENTS

The present research has been supported by CofinLab2000 and Cofin2003. The authors are grateful to their colleagues of Pennsylvania State University, who have developed and made available the basic computer code, as well as to G. Iaccarino and R. Verzicco for valuable suggestions and discussions on the immersed boundary method.
REFERENCES


