A level-set based sharp interface method for three-dimensional incompressible flows with complex moving boundaries

S. Marella, L. Tran & H. S. Udaykumar

University of Iowa, IowaCity, IA, USA

Abstract

This paper presents a computational technique capable of simulating three-dimensional flows involving stationary or moving boundaries on fixed grid. This method relies on finite-difference schemes for the flow governing equations and a level-set method to track the embedded boundaries. The mesh points adjoining the interface are subjected to special treatment to capture the immersed interface without smearing. The results of the accuracy studies indicate that the accuracy is not compromised in adopting finite-difference schemes instead of finite-volume methods. Several two- and three-dimensional flows simulated for validation show that the current method compares well with the benchmark results for a variety of problems.

1 Introduction

We present a finite-difference sharp interface technique to handle arbitrarily-shaped moving boundaries embedded on a Cartesian mesh. In previous work (Udaykumar et al. [1,2], Ye et al. [3]), a finite volume method with cut-cell treatment in combination with markers and curves for interface tracking was presented. That strategy proved effective for two-dimensional problems where the interface representation is straightforward and the various topologies of the cut-cells encountered can be tackled in a relatively straightforward fashion. The critical issues that would arise in such an approach when extending to three-dimensional moving boundaries are a) surface representation (involving surface meshing and re-meshing, which can be challenging for complex moving boundaries) and b) the cut-cell topology (numerous configurations of the control
volumes exist in 3D and special treatment for each case is tedious). As an alternative approach, we adopt a finite-difference scheme (which allow us to bypass the issues related to cut-cells) and level-set technique for interface tracking. The advantages of the level-set approach have been covered extensively in the literature (Osher and Sethian [4], Sussman et al. [5]). In adopting the finite-difference approach, the main concern is the accuracy and conservation properties of the flow solver. For the Cartesian grid approach the deviation of the finite-difference approach from the finite-volume approach appears only at the grid points adjoining the interface. In this work we evaluate the accuracy of the solutions obtained from the finite-difference and finite-volume discretizations.

2 Numerical method

2.1 Governing equations

The problems of interest are unsteady incompressible flows in the low to moderately high Reynolds number range ($Re \sim 1-10^5$). The non-dimensional unsteady, viscous, incompressible Navier-Stokes equations are solved:

$$\nabla \tilde{u} = 0$$  \hspace{1cm} (1)

$$\frac{\partial \tilde{u}}{\partial t} + \tilde{u} \nabla \tilde{u} = -\nabla p + \frac{1}{Re} \nabla^2 \tilde{u}$$  \hspace{1cm} (2)

where $\tilde{u}$ and $p$ represent the primitive variables velocity and pressure respectively and $Re$ is the Reynolds number of the flow. The problems under consideration are static/moving solid-fluid interfaces wherein the stress state inside the solid is assumed to be irrelevant to the flow (passive fluid-structure interactions). The interfacial boundary conditions that apply are no-slip and no-penetration conditions for velocity and zero normal gradient for the pressure.

2.2 Discretization for bulk cells

A cell-centered collocated arrangement of the flow variables is used to discretize the governing equations. A second-order two-step fractional step method (Chorin [6]) is used to advance the solution in time. The first step evaluates an intermediate velocity by solving an unsteady advection-diffusion equation. The second step solves a pressure Poisson equation obtained by explicit imposition of divergence free velocity field. An explicit Adams-Bashforth scheme and semi-implicit Crank-Nicholson scheme are used for the convection and diffusion terms respectively. The above procedure yields 5-point and 7-point stencils for two- and three-dimensional Cartesian meshes respectively. The above procedure provides a second-order accurate solver from the governing equations. A FAS multigrid is used to speed up the convergence for pressure. The details of the above discretization procedure are documented in Udaykumar et al. [1,2] and Ye et al. [3].
2.3 Level-set method

The basic idea of the level-set method is to maintain a scalar field \( \phi \) in addition to the flow variables at each mesh location. The value of \( \phi \) at any point is its signed normal distance from the interface. The interface location is implicitly embedded in the \( \phi \)-field since \( \phi = 0 \) contours represent the exact boundary. In case of moving interfaces, the motion of the boundary is tracked by level-set advection equation. A third-order ENO scheme in space and third-order Runge-Kutta method in time are used for the evolution of the level-set field. Since the velocity function is prescribed at the interface, the velocity field is extended off the front. A reinitializing procedure is carried out after level-set advection to reassign the new \( \phi \)-field as the signed distance function. The calculation of normal and curvature of the interface from the level-set field is trivial. The normal is given by \( \vec{N} = \vec{\nabla} \phi / \| \vec{\nabla} \phi \| \) and curvature is \( \kappa = -\vec{\nabla} \cdot \vec{N} \). Other issues related to level-set function are extensively published in the literature (Osher and Sethian [4], Sussman et al. [5]).

2.4 Discretization for the interfacial cells

As pointed out before, the main challenge in fixed-grid methods is accurate imposition of interfacial boundary conditions. Moreover, a sharp-interface method demands one-sided discretization for all the partial derivatives to avoid smearing of the interface characteristics.

Figure 1(a) illustrates a one-dimensional situation in which a cell (P) cut by the interface requires modifications in discretization procedure. The strategies adopted to deal with this situation for various differential operators in the governing equation are discussed below. A more complicated case is shown in Figure 1(b), in which both the neighbors of P (E & W) lie in dissimilar phases.

The operators of relevance for solving the governing equations are \( \partial^2 \phi / \partial x^2 \), \( \partial (\mu \phi) / \partial x \). The formulation of the first operator is shown in this section; extension to multiple dimensions is straightforward.

The second-derivative is encountered both in diffusion terms as well as pressure Poisson equation. The objective is to devise a second-order scheme for \( \partial^2 \phi / \partial x^2 \) at point P in terms of the function values at I, P, W, WW. Note that sharp treatment of the interface implies that discretization of all the operators
involves only points belonging to the same phase as P. In Figure 1(a), the neighbor E of P lies across the interface and hence cannot be used in discretization. Therefore:

\[
\frac{\partial^2 \phi}{\partial x^2} = \frac{\alpha \phi_I + \beta \phi_P + \gamma \phi_W + \eta \phi_{WW}}{\Delta x^2}
\]  

(3)

Using Taylor series expansions for each of \( \phi_i \)'s (i = I, W, WW) about P and further demanding \( \frac{\partial^2 \phi}{\partial x^2} \) to be of the order \( \Delta x^2 \) yields the following expressions for the coefficients.

\[
\alpha = 6/(\chi + 1)(\chi + 2) \quad \gamma = (4 - 2\chi)/(\chi + 1) \\
\eta = (\chi - 1)/(\chi + 2) \quad \beta = -\alpha - \gamma - \eta
\]  

(4a)

(4b)

In situations such as in Figure 1(b) where information for \( \phi_{WW} \) is not available, a first-order scheme is adopted. The corresponding expression can be derived with the same approach as outlined above. Similar procedures apply for discretizing the convection term.

3 Results and discussion

3.1 Accuracy studies

The objective of these accuracy studies is to compare the performance of the current finite-difference and level-set combination (FD+LS) to that of a finite-volume method and marker tracking (FV+MT) that was used in a previous sharp-interface method (Udaykumar et al. [1,2] and Ye et al. [3]). The computations of scalar diffusion and scalar convection-diffusion mechanisms are chosen as a basis for comparisons. The general computational setup for all of these cases is a circular cylinder of radius 0.25 placed at the center of a 1 x 1 domain. The interfacial boundary conditions and the domain boundary conditions are varied as per the situation.

For a simple diffusion problem, the cylinder surface and the left and bottom boundaries of the domain are maintained at a constant temperature of 1.0 and the right and bottom boundaries are maintained at 0.0. The system is allowed to reach steady state and the norm of error in temperature over the whole domain is calculated. We use FD+LS as well as FV+MT to solve this problem. Variation of the L2 norm with grid spacing using various methods is compared in Figure 2(a). This graph shows that the FD+LS combination compares well with the previous FV+MT method. The order of accuracy (given by the slope of the graph) is close to 2.0. The next case retains the same setup but imposes a uniform flow (\( u = 1.0 \), \( v = 1.0 \)) everywhere in the domain. Also, all the boundaries are maintained at isothermal temperature of 0.0. As in the previous case, both finite-difference based and finite-volume based methods are used to solve the convection-diffusion equation. The variation of L2 error norms with grid spacing, at steady state, for each of these methods is plotted in Figure 2(b). The slopes of the lines indicate that the order of accuracy and the magnitude of errors for both the
methods are similar. These accuracy studies indicate that the present method compares well with finite-volume formulation.

3.2 Validation studies

3.2.1 Two-dimensional flows

3.2.1.1 Flow past a stationary circular cylinder. This study validates the computational technique for immersed stationary boundaries over a range of Reynolds numbers by simulating steady and unsteady flow regimes past a circular cylinder immersed in an unbounded flow. We simulate flows for Reynolds number (Re_d) = 40, 80, and 300 and compare the results to various numerical and experimental results from the literature and also to our own finite-volume approach. A domain size of 30d x 30d with computational mesh of 452x452 has been used to minimize the boundary effects and to capture the boundary layer accurately. The boundary conditions on the top, bottom and the left boundaries correspond to the potential flow past a cylinder and the right boundary is specified as an outlet.

For Re = 40 the flow develops an axisymmetric wake and attains a steady state. The steady recirculation zone behind the cylinder is shown in Figure 3(a). The length of the recirculation and the drag coefficient, \( C_D = \text{drag force}/(1/2)\rho U_o^2 d \), are computed for comparison with the established results. Table 1 shows that the results from the current method compare well with other experimental and numerical solutions.

At Re_d = 80, the wake behind the cylinder develops into the classic Karman vortex street. This is as expected given that the \( Re_{critical} = 46 \) marks the inception of unsteady flow. The asymmetry of the flow can be noted in Figure 3(b) which displays the streamlines around the cylinder. The steady vortex shedding is illustrated from the plot of drag and lift coefficients shown in Figure 3(c). The mean drag coefficient computed is 1.37 which is in agreement with our previous
(finite-volume) results. The Strouhal number for the vortex shedding defined as $St = f d / U_o$, where $f$ is the shedding frequency, characterizes the shedding process. A Strouhal number of 0.15 is estimated from the variation frequency of the lift coefficient. Figure 4(a) and 4(b) show the streamlines and spanwise velocity contours at a certain instant of time for $Re_d = 300$. The variation of drag and lift coefficients are shown in Figure 4(c). The mean drag coefficient of 1.28 and $St = 0.21$ agree well with the experimental value of Weiselsberger [11] as well as our previous results.

### Table 1: Comparison with benchmark data for flow around cylinder.

<table>
<thead>
<tr>
<th>Re</th>
<th>40</th>
<th>80</th>
<th>300</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$C_b$</td>
<td>$L/D$</td>
<td>$C_D$</td>
</tr>
<tr>
<td>Study ↓</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tritton [7]</td>
<td>1.48</td>
<td>-</td>
<td>1.29</td>
</tr>
<tr>
<td>Fornberg [8]</td>
<td>1.50</td>
<td>2.24</td>
<td>-</td>
</tr>
<tr>
<td>Mittal and Balachandar [9]</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Williamson [10]</td>
<td>-</td>
<td>-</td>
<td>0.15</td>
</tr>
<tr>
<td>Finite volume [3]</td>
<td>1.52</td>
<td>2.27</td>
<td>1.37</td>
</tr>
<tr>
<td>Current</td>
<td>1.52</td>
<td>2.30</td>
<td>1.36</td>
</tr>
</tbody>
</table>

#### 3.2.1.2 Flow past an oscillating cylinder. The present study is carried out to validate the capability of handling flows past moving boundaries. To illustrate this feature we simulate the flow past a transversely oscillating cylinder and examine the classical "lock-on" phenomenon for vortex shedding. Flow past a cylinder at $Re = 200$ undergoing sinusoidal transverse oscillation is simulated for a range of oscillation frequencies and the results are compared to the experimental results of Koopmann [12].

The cylinder center $(x_o, y_o)$ is located at $(8d, 10d)$ relative to the left bottom corner in a $20d \times 20d$ domain with 400x400 computational mesh points. A uniform flow $U_o$ is prescribed on the left, top and bottom boundaries and the right boundary has an outlet boundary condition. A sinusoidal motion of the form $x_c(t) = x_o, y_c(t) = y_o + A \sin(2\pi ft)$ is imposed on the cylinder, where $t$ is the non-dimensional time, $A$ is the amplitude of oscillation, $f$ is the forced frequency of oscillation. The mesh employed is fine enough to capture the flow features in the vicinity of the cylinder.

The flow past a stationary cylinder at $Re = 200$ has been simulated before proceeding to an oscillating cylinder. The Strouhal number of $f_o = 0.198$ and drag coefficient of $C_D = 1.37$ were calculated for this flow. As the next step, flows past a cylinder at various frequencies around the natural shedding frequency have been simulated. A fixed amplitude of $A = 0.1$ and $f_t = 0.17, 0.19, 0.2$ and 0.23 have been used to study the lock-on phenomenon.
Figure 3: (a) Axisymmetric wake behind the cylinder at $Re = 40$ (b) Streamlines past the cylinder at $Re = 80$ (c) Variation of Lift and Drag coefficients at $Re = 80$.

Figure 4: Flow past a cylinder at $Re = 300$. (a) Streamlines (b) Spanwise velocity contours (c) Variation of Lift and Drag coefficients.

Figure 5: Flow past an oscillating cylinder at $Re = 200$ (a) Spanwise velocity contours (b) Fluctuations in spanwise velocity component at a point in the wake region. (c) Comparison of the present results with the Koopmann [12] curve for lock-on region. The closed squares indicate lock-on frequencies and Open Square indicate no lock-on.
These simulations have been performed for about 200 non-dimensional time units and the shedding frequency has been calculated using the fluctuations in the velocity of points in the near wake region. The results are compared to the experimental results of Koopmann [12] for Re = 200. In the present study we have obtained two points in the lock-on (0.19, 0.20) region and two points in the non-lock on region (0.17, 0.23). Figure 5(c) shows the agreement between the current results and experimental results for amplitude of 0.1. The pattern of the vortices shed by the oscillating cylinder is illustrated in Figure 5(a). The variation of x-component of velocity at a mesh point in the wake region is plotted in Figure 5(b). This is used to calculate the vortex shedding frequency. Our results indicate that the lock-on occurs for oscillation frequencies close to the natural frequency. A wider range of frequencies and amplitudes could be studied to establish the exact lock-on curve. However, the purpose of the present study (validation) confines us to limited values in the parameter space.

3.2.2 Three-dimensional flows

3.2.2.1 Flow around stationary sphere. In the present study, simulation of flow around a stationary sphere is viewed as a suitable case for validating the numerical method for problems involving three-dimensional fixed immersed boundaries. The experimental and numerical results available in the literature indicate the transition from axisymmetric flow to non-axisymmetric flow and transition from steady to unsteady flow as two important flow features in this case. Tracking these transition points exactly as well as obtaining recirculation lengths and drag coefficients comparable to the benchmark data would validate the current method. We simulate laminar flows up to Re = 215 and compare the results to benchmark results.

A 15d x 15d x 15d domain with 130 x 110 x 110 mesh points is used for these simulations. The intensive computational effort limits us from using a larger domain and finer mesh. The boundary conditions on all the boundaries (except the right boundary) correspond to the potential flow past a sphere and the right boundary is specified as an outlet. Unlike its two-dimensional analogue (cylinder), uniform flow around sphere remains stable till much higher Reynolds number. It has been established in several studies, that the transition from axisymmetric to non-axisymmetric flow occurs at Re ~ 210 and the flow becomes unsteady at Re of around 270.

The flow simulations for Re = 50,100,150 and 215 are presented in this section. The graphs in Figure 6(a),(b)&(c) illustrate the streamlines, on a z=7.5 plane for Reynolds numbers of 50,100 and 150 respectively. As expected, the flow has a steady, axisymmetric wake in all these cases. The length of the recirculation zone and the drag coefficient, defined by $C_D = \frac{\text{drag force}}{(1/2)\rho U_0^2d}$, are quantified to compare with the benchmark data. Table 2 shows a detailed comparison of the current results with the standards for a range of Reynolds numbers.
Figure 6: The axisymmetric streamlines past the sphere. (a) Re = 50 (b) Re = 100 (c) Re = 150.

Figure 7: Streamlines of the velocity vectors at Re = 215. (a) u,v vectors on x-y plane (b) u,w vectors on x-z plane.

Table 2: Comparison of results with benchmark data for flow around a sphere.

<table>
<thead>
<tr>
<th>Re</th>
<th>50</th>
<th>100</th>
<th>150</th>
<th>215</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mittal [13]</td>
<td>1.57</td>
<td>0.44</td>
<td>1.09</td>
<td>0.87</td>
</tr>
<tr>
<td>Clift et al. [14]</td>
<td>1.57</td>
<td>-</td>
<td>1.09</td>
<td>0.89</td>
</tr>
<tr>
<td>Johnson &amp; Patel [15]</td>
<td>1.57</td>
<td>0.40</td>
<td>1.08</td>
<td>0.86</td>
</tr>
<tr>
<td>Current</td>
<td>1.56</td>
<td>0.39</td>
<td>1.06</td>
<td>0.88</td>
</tr>
</tbody>
</table>

It has been experimentally established that a non-axisymmetric flow regime prevails in the range 210<Re< 270. The streamlines and the pressure contours on z= 7.5 and y= 7.5 planes are shown in Figure 7(a) & (b) to illustrate the axial asymmetry for a Re = 215 case which is close to the point where asymmetry occurs. The simulations to track the transition to unsteady flow and simulation of flow past oscillating spheres are in progress.
4 Conclusions

A numerical technique to solve incompressible fluid flows with immersed bodies in two and three dimensions has been developed. The salient features of the technique are a second-order accurate finite-difference scheme to discretize the flow governing equations, a special treatment for the cells along the immersed interface to capture the interface features in a sharp fashion and a level-set based interface tracking technique.

Several computational tests have been carried out to ascertain the performance of the method. The results from the accuracy studies, in which the present method is compared to a finite volume method, show that the second order accuracy is not compromised by adopting the current method. Validation of two-dimensional flows around stationary and oscillating cylinders have been performed. The current results match well with the benchmark results. Flow around a stationary sphere has been simulated for a range of Reynolds numbers to validate the three-dimensional capability. The results compare well with the established experimental and numerical results.

In the ongoing work, simulations of flow around sphere for higher Reynolds numbers to track the transition from steady to unsteady flow as well as validation of flows involving three-dimensional moving boundaries are being performed.

References


