Journal of Computational Physics 354 (2018) 672–691

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp

An immersed boundary formulation for simulating high-speed compressible viscous flows with moving solids

Yegao Qu [a,b], Ruchao Shi [a], Romesh C. Batra [a,*]

[a] Department of Biomedical Engineering and Mechanics, MC 0219, Virginia Polytechnic Institute and State University, Blacksburg, VA 24061, USA

[b] State Key Laboratory of Mechanical System and Vibration, Shanghai Jiao Tong University, Shanghai 200240, China

ARTICLE INFO

Article history:
Received 11 March 2017
Received in revised form 6 September 2017
Accepted 24 October 2017
Available online 26 October 2017

Keywords:
Navier–Stokes equations
Fluid–solid interface
Immersed boundary method
Constrained moving least-squares method
Flow discontinuities

ABSTRACT

We present a robust sharp-interface immersed boundary method for numerically studying high speed flows of compressible and viscous fluids interacting with arbitrarily shaped either stationary or moving rigid solids. The Navier–Stokes equations are discretized on a rectangular Cartesian grid based on a low-diffusion flux splitting method for inviscid fluxes and conservative high-order central-difference schemes for the viscous components. Discontinuities such as those introduced by shock waves and contact surfaces are captured by using a high-resolution weighted essentially non-oscillatory (WENO) scheme. Ghost cells in the vicinity of the fluid–solid interface are introduced to satisfy boundary conditions on the interface. Values of variables in the ghost cells are found by using a constrained moving least squares method (CMLS) that eliminates numerical instabilities encountered in the conventional MLS formulation. The solution of the fluid flow and the solid motion equations is advanced in time by using the third-order Runge–Kutta and the implicit Newmark integration schemes, respectively. The performance of the proposed method has been assessed by computing results for the following four problems: shock-boundary layer interaction, supersonic viscous flows past a rigid cylinder, moving piston in a shock tube and lifting off from a flat surface of circular, rectangular and elliptic cylinders triggered by shock waves, and comparing computed results with those available in the literature.

1. Introduction

The interactions of high-speed compressible viscous flows with irregularly shaped objects are commonly encountered in aerospace applications. These interactions may encompass various flow phenomena including shock wave reflection and diffraction, as well as shock–shock, shock-vortex and shock-boundary layer interactions. In addition to the challenge of representing various discontinuities in a high-speed flow, simulating an irregular-shaped solid moving in a compressible viscous flow is very challenging. Numerical methods used to solve such problems employ either a finite-difference or a finite volume grid to represent the computational domain and a variety of algorithmic approaches to satisfy continuity conditions at the fluid–solid interface.

An ideal numerical method for simulating high-speed flows should be accurate and free from numerical dissipation in smooth parts of the flow, and must robustly capture flow discontinuities without significant Gibbs ringing that can lead

* Corresponding author.
E-mail addresses: quyegao@sjtu.edu.cn (Y. Qu), rbatra@vt.edu (R.C. Batra).

https://doi.org/10.1016/j.jcp.2017.10.045

to nonlinear instability [1]. The shock-capturing upwind-biased schemes commonly employed to suppress the Gibbs oscillations include, but are not limited to, the total variation diminishing (TVD) methods with flux or slope limiters [2], the monotonicity-preserving (MP) methods [3–5], the essentially non-oscillatory (ENO) [6,7] methods and the weighted essentially non-oscillatory (WENO) [8,9] methods. The TVD methods can be easily implemented but they reduce to first-order accuracy at the local extrema of solutions and can be numerically very diffusive for computing solutions involving oscillatory waves. The MP method proposed by Suresh and Huynh [3] generalizes the TVD schemes and shows very good performance in preserving both the accuracy in smooth flow regions as well as the monotonicity near discontinuous flow regions [4,5]. The ENO schemes determine the numerical flux from a high-order reconstruction over an adaptive stencil that is selected to minimize interpolation across discontinuities and hence diminish Gibbs oscillations. However, the ENO schemes are found to be less stable for computing steady flows than the TVD techniques since the TVD condition is not rigorously satisfied in the ENO schemes [10,11]. In a WENO scheme, a high-order numerical flux is constructed by using a convex linear combination of lower-order polynomial reconstructions over a set of staggered stencils, with weights selected to achieve the maximum formal order of accuracy in smooth regions, and nearly zero weight assigned to reconstructions on stencils crossed by discontinuities. The WENO schemes improve robustness, convergence and efficiency of the ENO schemes, and tend to have uniform higher-order accuracy in smooth regions and maintain the essentially non-oscillatory properties near shock waves. Several WENO methods including the third-, the fifth- and the higher-order have been developed [12,13]. For high-speed flows, the WENO schemes seem to have superceded other shock-capturing methods in the last decade and have proved to be extremely accurate and robust in the presence of strong shock waves and complex shock interactions [1].

For numerically simulating flows interacting with complex-shaped solids, the correct enforcement of boundary conditions on the fluid–solid interface is important for the accuracy and stability of the numerical method. The body-fitted grid methods [14–16] have been commonly employed for such problems that transform governing equations and boundary conditions of the fluid into body-fitted coordinate systems with either a structured or an unstructured grid thereby easily enforcing boundary conditions on the fluid–solid interface for stationary solids with smooth boundaries. However, the mesh generation for a complex-shaped solid is cumbersome. Moreover, for flows involving moving solids, transient re-meshing strategies are required which further increases the computational and algorithmic complexity of the body-fitted grid methods. A different approach that retains most of the favorable properties of structured grids but also provides a high level of flexibility in handling irregular-shaped geometry is the immersed boundary method [17,18]. In this method, the requirement of the grid conforming to a solid boundary is relaxed by using a non-conforming grid, and the effect of a complex object on the flow is considered through proper treatment of the solution variables at the grid cells in the vicinity of the body. This method can tackle flows with complex stationary or moving boundaries with relative ease. However, as the solid boundary can arbitrarily cut through the underlying mesh, one needs to treat the boundary in a way that does not adversely impact the accuracy and conservation property of the underlying solver.

Based on the representation of the fluid–solid interface, the immersed boundary methods may be classified as either diffused or sharp interface [18]. In diffused interface methods, an immersed boundary is smeared by distributing singular forces to the surrounding background grid nodes using discrete delta functions [19] or mask functions for penalty methods [20]. The diffused interface methods can be formulated independent of the spatial discretization, and therefore can be easily implemented in an existing fluid solver. However, they produce a "diffused" boundary, and the boundary conditions on an immersed surface are not precisely satisfied at its actual location but within a localized region around the boundary. The so called "sharp interface" methods include, to name a few, the ghost-cell [21–23], the cut-cell [24,25] and the immersed interface methods [26], which strongly depend upon the spatial discretization of the immersed boundary and in which a solid boundary is precisely tracked. The sharp interface methods are preferred because of accuracy, particularly for flows with thin boundary layers. The ghost-cell methods are considered to be less accurate than the cut-cell methods at the same resolution of an underlying Cartesian grid due to its inherent implicit representation of the solid boundary. However, they can be easily implemented and are computationally efficient as it is not necessary to modify flux calculations of an existing Cartesian-grid solver. Moreover, the complicated cell reshaping procedure required in a cut-cell method is not needed in a ghost-cell method.

A critical issue in a ghost-cell immersed boundary method is the accuracy of the reconstruction solution at nodes near the immersed interface via appropriate interpolation schemes using known values on the solid surface and the information from the interior of the flow. The accuracy of the interpolation/extrapolation is an important aspect of an immersed boundary method since it directly influences the number of computational cells required to resolve a flow field as economically as possible. A classical scheme computing the ghost-cell values is the bilinear interpolation for two-dimensional (2-D) problems [21] (trilinear interpolation for 3-D problems [22]). However, when the interpolation point is very close to the boundary, all neighboring points required for the interpolation may not be in the fluid domain. In such cases, the information at the desired point can be found either by using a reduced-order interpolation scheme or by employing body-intercepting points for the interpolation. The inverse distance weighting interpolation method has also been used to construct the fluid values in sharp interface immersed boundary methods [21,27,28]. This scheme is stable for reconstructing variables that smoothly vary without exhibiting large maximum values. Toja-Silva [29] developed an immersed boundary method based on radial basis functions for the interpolation of the near-boundary cells that had convergence rate of one. Note that the accuracy of the above-mentioned methods is at most second order. Interpolations based on higher-order polynomials are expected to be more accurate, but they often lead to numerical instabilities and the determination of appropriate stencils for such interpolations is very difficult. Seo and Mittal [30] applied a moving least-squares (MLS)