COMMUNICATIONS IN COMPUTATIONAL PHYSICS
Vol. 2, No. 6, pp. 1095-1124

Commun. Comput. Phys.
December 2007

A Mesh Adjustment Scheme for Embedded Boundaries

J. S. Sachdev and C. P. T. Groth*

University of Toronto Institute for Aerospace Studies, 4925 Dufferin Street, Toronto, Ontario, M3H 5T6, Canada.

Received 20 December 2006; Accepted (in revised version) 20 March 2007

Communicated by Kun Xu

Available online 15 June 2007

Abstract. An adaptive meshing technique and solution method is proposed in which a two-dimensional body-fitted multi-block mesh is locally adjusted to arbitrarily embedded boundaries that are not necessarily aligned with the mesh. Not only does this scheme allow for rapid and robust mesh generation involving complex embedded boundaries, it also enables the solution of unsteady flow problems involving bodies and interfaces moving relative to the flow domain. This scheme has been implemented within a block-based adaptive mesh refinement (AMR) numerical framework which can ease computational expense while maintaining a detailed representation of the embedded boundary and providing an accurate resolution of the spatial characteristics of the fluid flow. Rigid body motion and evolving motion due to physical processes are considered. A block-based AMR level set method is used to deal with evolving embedded boundaries. Numerical results for various test problems are presented to verify the validity of the scheme as well as demonstrate the capabilities of the approach for predicting complex two-dimensional inviscid and laminar fluid flows.

AMS subject classifications: 65M50, 68W10, 74S10, 76M12

PACS (2006): 47.11.-j, 47.11.Df, 47.55.N-, 47.40.-x

Key words: Dynamic and stationary embedded boundaries, mesh adjustment algorithm, block-based adaptive mesh refinement.

1 Introduction and motivation

Fluid flows involving moving boundaries are relevant to many aerospace engineering applications, such as: aerodynamic control surfaces, helicopter rotor blades, compressor and turbine blades, stage separation in launch vehicles, and the combustion interface in solid propellant rocket motors. Numerical solution of these unsteady flows must account for the motion the boundaries through the domain of interest. This can be accomplished

*Corresponding author. *Email addresses:* `j.sachdev@utoronto.ca` (J. S. Sachdev), `groth@utias.utoronto.ca` (C. P. T. Groth)

by regenerating or adjusting the computational mesh according to the object's location [6,25], by using overlapping meshes [7], or by representing the boundary as a force-term which influences the flow solution in a manner consistent with fluid dynamics [40, 41]. These approaches are now briefly summarized.

The Arbitrary Lagrangian-Eulerian (ALE) method pioneered by Hirt et al. [25] is a popular method for dealing with moving boundaries [3, 19, 20, 54]. This scheme involves the use of a grid, typically unstructured, that moves according to the motion of the boundaries. Conservation is strictly satisfied in ALE methods, however, the quality of the mesh can be severely degraded when large scale motions are involved requiring re-meshing and/or untangling procedures [50].

An alternative approach is the use of overlapping meshes, also known as overset mesh or the Chimera method, where a separate body-fitted mesh is constructed around each object in the computational domain and is overlapped with each other and a simpler, in many cases Cartesian, mesh encompassing the computational domain of interest. This method was originated by Benek et al. [7] and a review of recent work on this method was given by Noack and Slotnick [35]. Interpolation is required between the meshes during the solution of the governing equations. These interpolation schemes are typically non-conservative and non-monotone [36], which may be problematic for many applications.

The immersed boundary method was devised by Peskin [40, 41] in order to predict fluid-structure interactions in biological fluid dynamics. In this scheme the boundary is represented as a force term in the governing equations which influences the flow solution in a manner consistent with fluid dynamics. The immersed boundary method has been successfully applied to complex laminar and turbulent flows [14,29,49].

Although the methods described above have been shown to be effective for treating embedded boundaries, in this paper, a new approach is proposed in which a body-fitted multi-block mesh is locally adjusted to arbitrarily embedded boundaries that are not necessarily aligned with the mesh. Not only does this scheme allow for rapid and robust mesh generation involving complex embedded boundaries, it also enables the solution of unsteady flow problems involving bodies and interfaces moving relative to the flow domain. The mesh adjustment algorithm described here has similarities with the Cartesian cut-cell techniques developed by De Zeeuw and Powell [15] and Aftosmis et al. [2] except that the underlying mesh is no longer restricted to a Cartesian mesh. The readjustment of the mesh for moving embedded boundaries follows the approach used in the Cartesian cut cell methods developed by Bayyuk et al. [6] and Murman et al. [34]. Unlike the Cartesian cut-cell method, the mesh adjustment algorithm presented here does not result in the generation of tiny cut-cells which can be restrictive on the time-step and do not permit the construction of consistent and accurate operators for viscous (elliptic) fluxes [12]. This scheme has been implemented within a block-based adaptive mesh refinement (AMR) numerical framework [43]. It has been well established that AMR techniques, which automatically adapt the computational grid to the solution of the governing partial differential equations, are very effective in treating fluid problems with disparate length scales while minimizing computational expense [8,9,15,23,43].