Proceedings of the Seventeenth (2007) International Offshore and Polar Engineering Conference
Lisbon, Portugal, July 1-6, 2007

ISBN 978-1-880653-68-5; ISBN 1-880653-68-0(Set); ISSN 1098-6189 (Set)

Study On a Structured Multi-Block Cartesian Grid Method

Yih-Ferng Peng, Chih-Min Hsieh
Department of Civil Engineering, National Chi Nan University, Taiwan
Institute of Physics, Academia Sinica, Nankang, Taipei, Taiwan, China

ABSTRACT

In this paper, the local grid refinement is focused by using a multi-block technique. The Cartesian grid numerical method is developed for simulating unsteady, viscous, incompressible flows with complex immersed boundaries. A finite volume method is used in conjunction with a two-step fractional-step procedure. The key aspects that need to be considered in developing such a multi-block solver are imposition of interface conditions on the inter-block boundaries and accurate discretization of the governing equation in cells that are with block-interface as a control surface. A new interpolation procedure is presented which allows systematic development of a spatial discretization scheme that preserves the spatial accuracy of the underlying solver. The present multi-block method has been tested by two numerical examples to examine its performance in the two dimensional problems. The numerical examples include flow past a circular cylinder symmetrically installed in a Channel and flow past two circular cylinders with different diameters. From the numerical experiments, the ability of the solver to simulate flows with complicated immersed boundaries is demonstrated and the multi-block approach can efficiently speed up the numerical solutions.

KEY WORDS: Cartesian grid, multi-block, fractional-step.

INTRODUCTION

In the numerical simulation of complex physical phenomena, the crucial requirement is predictability, *i.e.*, that the simulation results remain faithful to the actual physical processes. Errors resulting from a lack of spatial resolution are particularly deleterious. However, over-resolving is computationally expensive. As a result, how to efficiently and effectively solve the partial differential equations which represent the mathematical model of physical problems concerned becomes a subject of active research in numerical analysis (Chacon and Lapenta, 2006, Ding and Shu, 2006).

In general, there are two approaches to obtain accurate solution of PDEs. One approach is to employ high-order numerical method, and the other is to improve the resolution through the computational grid. Mesh refinement is desirable to improve spatial resolution by using uniform or non-uniform grids. The uniform mesh refinement is that the resulting grid evolution equation is generally less nonlinear and less stiff which becomes very efficient in conjunction with the line successive-overrelaxation (SOR) solver. However, the uniform mesh refinement is not perfect for the applications, of which the solution may need different resolutions for different regions. For the well-understood physical problems, a non-uniform mesh can be designed to reflect the resolution requirement of practical problems. For example, for the boundary value problems, fine resolution is typically required for regions near boundaries. But for the evolving interfacial flows or flow field with complicated structures, local refinement techniques are more preferred to locally increase mesh densities in the regions of interest, thus saving the computer resources. Nowadays, local mesh refinement in general plays an indispensable role in the efficient solution of industrial and scientific problems. The strategies of adaptive mesh refinement can fall into structured or unstructured mesh refinement approach. One representative of structured grid approaches is adaptive Cartesian mesh refinement proposed by Berger and Oliger (1984). Their approach is established on regular Cartesian meshes, but arranged hierarchically with different resolutions. At the fine-coarse cell interfaces, special treatment is required for the communications between the meshes at different levels. With regard to the unstructured mesh refinement approach, Zienkiewicz and Zhu (1988) reviewed the state-of-the-art of the automatic mesh refinement strategies in the finite element community and discussed the important role of error estimation and automatic adaptation in the finite element analysis.

In this paper, we develop a structured, multi-block Cartesian grid method for simulating unsteady, viscous, incompressible flows with complex immersed boundaries. One drawback of adopting multi-block method as the local refinement technique is that the resulting multi-block grids are uniform and the corresponding evolution equations is less stiff and very efficient in conjunction with line-SOR. For the numerical simulations of flow past bluff body problems in the present study, we used an Immersed Boundary (IB) method where the solid object is represented by a distributed body force in the Navier–Stokes equations.

The current paper will focus on describing these and other salient features of the numerical methodology, validating the accuracy and fidelity of the approach and demonstrating the capabilities of the solver in some complex configurations.

NUMERICAL METHODOLOGY