Available online at www.sciencedirect.com
ScienceDirect
Journal of Computational Physics 227 (2008) 6895–6921
JOURNAL OF COMPUTATIONAL PHYSICS
www.elsevier.com/locate/jcp

An object-oriented and quadrilateral-mesh based solution adaptive algorithm for compressible multi-fluid flows

H.W. Zheng, C. Shu *, Y.T. Chew

Department of Mechanical Engineering, National University of Singapore, 10 Kent Ridge Crescent, Singapore 119260, Singapore

Received 26 September 2007; received in revised form 2 March 2008; accepted 31 March 2008
Available online 10 April 2008

Abstract

In this paper, an object-oriented and quadrilateral-mesh based solution adaptive algorithm for the simulation of compressible multi-fluid flows is presented. The HLLC scheme (Harten, Lax and van Leer approximate Riemann solver with the Contact wave restored) is extended to adaptively solve the compressible multi-fluid flows under complex geometry on unstructured mesh. It is also extended to the second-order of accuracy by using MUSCL extrapolation. The node, edge and cell are arranged in such an object-oriented manner that each of them inherits from a basic object. A home-made double link list is designed to manage these objects so that the inserting of new objects and removing of the existing objects (nodes, edges and cells) are independent of the number of objects and only of the complexity of $\boldsymbol{O}(\boldsymbol{1})$. In addition, the cells with different levels are further stored in different lists. This avoids the recursive calculation of solution of mother (non-leaf) cells. Thus, high efficiency is obtained due to these features. Besides, as compared to other cell-edge adaptive methods, the separation of nodes would reduce the memory requirement of redundant nodes, especially in the cases where the level number is large or the space dimension is three. Five two-dimensional examples are used to examine its performance. These examples include vortex evolution problem, interface only problem under structured mesh and unstructured mesh, bubble explosion under the water, bubble-shock interaction, and shock-interface interaction inside the cylindrical vessel. Numerical results indicate that there is no oscillation of pressure and velocity across the interface and it is feasible to apply it to solve compressible multi-fluid flows with large density ratio (1000) and strong shock wave (the pressure ratio is 10,000) interaction with the interface.

Keywords: Adaptive mesh refinement; Compressible multi-fluid; HLLC; Finite volume; Quadrilateral

1. Introduction

The dynamics of compressible multi-fluid flows have many practical applications in engineering, such as shock-bubble interactions [1,2] and underwater explosions [3]. Due to complexity of these problems such as steep material interface, high density and pressure gradients that occur in such flows, accurate predictions

* Corresponding author. Tel.: +65 6874 6476; fax: +65 6779 1459.
E-mail address: mpeshuc@nus.edu.sg (C. Shu).

0021-9991/$ - see front matter
doi:10.1016/j.jcp.2008.03.037

of flow structure of compressible multi-fluid flows pose a major challenge. Thus, it is necessary to apply the adaptive mesh technique to improve the efficiency and accuracy.

In the past decades, intensive research and efforts have been devoted to the development of adaptive mesh refinement technique. As a result, a large number of adaptive algorithms have been proposed and can be categorized as three categories, h-refinement, p-refinement and r-refinement. The basic idea behind adaptive mesh refinement is to increase the density of cells by adding cells in regions where accurate solution is required. h-Refinement increases the mesh resolution by adding elements or vertices to the mesh. For the method of p-refinement, it improves solution by increasing the order of accuracy of the polynomial in each element (or cell). As compared to the previous two methods, r-refinement [4,5] modifies the mesh density through the use of node movement and keeps the number of vertices and elements unchanged. Among these three methods, h-refinement has been widely applied in many areas [6–8].

The popular h-refinement method is the Cartesian grid solver which may be implemented by the nested hierarchical data structure such as quad-tree in 2D [8,9] and octree in 3D [10]. A more general version is to use block adaptive Cartesian meshes [6,11,12] organized in a series of rectangular grid patches. The main drawback of these adaptive Cartesian meshes is their difficulty to be applied to complex geometry. Besides, all these tree-based Cartesian adaptive approaches encounter the difficulties of vectorization (on vector architectures) and tree traversal overhead due to the searching of neighboring cells. Another type of h-refinement method does not use the tree but an unstructured data structure where the connectivity is explicitly stored with the mesh [13–15]. This unstructured feature makes it be easily extended to complex geometry. Besides, as the neighboring references are explicitly stored, the calculation time required for computing quantities involving the neighbors will be lesser than that for the tree-based approach where the recursive tree traverses are required to search the neighboring connectivity.

Although there are many adaptive mesh methods, their extension and applications to compressible multi-fluid flows are still rarely conducted, especially for cases with a large density difference and strong shock wave interaction at the material interface. Recently, Nourgaliev et al. [3] presented the work by combining the ghost fluid method and the block structured adaptive Cartesian meshes. However, it is not easy to be applied to problems with complex geometry due to the use of structured meshes. Besides, as reported by the authors, the coarse-to-fine and fine-to-coarse inter-level transfer operators are very complicated and may violate the stability of the code. Moreover, the ghost fluid method employs the level set method to track the interface. The mass or momentum may not be conserved.

Hence, in order to deal with problems with complex geometry, we adopt the unstructured adaptive technique and the diffuse interface method [2,16] instead of the Cartesian structured mesh and the ghost fluid method. The well-designed data structure for the objects (node, edge and cell) and memory arrangement of the object lists contribute to the fact that the adding, deleting of a certain object is only of the order of $\boldsymbol{O}(\boldsymbol{1})$ as compared to $\boldsymbol{O}(\boldsymbol{n})$ of other cell-edge based methods [15,17]. Besides, due to the hierarchy storage of cell and the separated storage of leaf edges and non-leaf edges in different lists, the edge-based finite volume solver can be easily applied to the current adaptive mesh solver as compared to the cell-based finite volume solver [15,17]. Moreover, the separation of node will reduce the memory requirement of redundant nodes, especially in the cases where the level number is large or the space dimension is three. It will also make the conversion of conservative variables from cell-centroid to the cell-vertex and the extension of adaptive mesh method from 2D to 3D easier and more effective. Hence, from the viewpoint of implementation, our adaptive algorithm is quite simple and efficient.

For the simulation of compressible multi-fluid flows, a well-known difficulty [2,16,18] associated with Euler solvers in the conservative form is the possible appearance of spurious pressure oscillations at material interfaces. Thus, the quasi-conservative form is applied to conquer this problem. Besides, a suitable Godunov-type scheme for multi-fluid flows should resolve the contact wave properly in order to capture the interfaces. In this work, Harten, Lax and van Leer approximate Riemann solver with the contact wave restored, called the HLLC scheme [19–21] is adopted. This method is further extended to the second-order of accuracy by monotonic upwind schemes for conservation laws (MUSCL).

The paper is organized as follows. The finite volume algorithm for compressible multi-fluid flows under unstructured adaptive mesh is given in Section 2. The data structure and the implementation of the adaptive method are proposed in Section 2. The model is extensively validated for single fluid vortex evolution problem,