Computers & Fluids 70 (2012) 126–135

Contents lists available at SciVerse ScienceDirect

Computers & Fluids

journal homepage: www.elsevier.com/locate/compfluid

Shock capturing schemes with local mesh adaptation for high speed compressible flows on three dimensional unstructured grids

Vinh-Tan Nguyen *, Hoang Huy Nguyen, Matthew Antony Price, Jaik Kwang Tan

Institute of High Performance Computing, 1 Fusionopolis Way, #16-16 Connexis, Singapore 138632, Singapore

A R T I C L E I N F O

Article history:
Received 16 February 2012
Received in revised form 7 August 2012
Accepted 9 September 2012
Available online 2 October 2012

Keywords:
Computational fluid dynamics
Shock capturing
Mesh adaptation
Unstructured grids
Solution adaptivity

A B S T R A C T

This paper contributes towards a more complete approach to capture very strong shocks in various applications of high speed compressible Navier–Stokes flows including blasts and explosions using second order finite volume method on unstructured grids. The HLLC Riemann solver is employed to solve for fluxes at cell interfaces with second order approximation of local Riemann states, thus obtaining second order accuracy. In order to stabilize solutions due to high order approximation of solutions in the presence of discontinuities, several strategies are presented in this work. Slope limiters are first explored on unstructured grid to maintain monotonicity of the solution reconstruction following local extremum diminishing (LED) or total variation diminishing (TVD) criteria. The hybrid HLLC/HLLE scheme is appended to eliminate shock instabilities in very strong shock cases. To improve resolution of shocks, a local mesh adaptation scheme is used to increase mesh resolution in areas of high gradients. The scheme only regenerates mesh locally and is proven to be robust and efficient for capturing of unsteady shock propagation applications. Comparisons on the accuracy and performance of different methods on various applications are drawn to suggest a more robust and efficient method for capturing shocks on unstructured grids.

1. Introduction

High speed compressible flows are fundamental to many engineering problems involving hypersonic flows, blasts and explosions, shock wave propagations. A wide range of numerical methods have been developed for simulations of compressible flows in the presence of very strong shocks. In particular, second order finite volume methods on unstructured grids have become desirable computations in many industrial applications. In these high speed compressible flow applications, strong shocks in the solutions present a formidable challenge in resolving discontinuities. It is then essential to derive an effective approach to accurately capture flow features including strong shocks.

In capturing shocks, Godunov-type schemes [4] has been popular for its conservative treatment of resolving discontinuities. Under a basic assumption of piece-wise constant solution across elements in the Godunov-type methods, intercell fluxes need to be constructed from the current solution in order to compute the solution in the next stage. The intercell fluxes are solutions to the so-called local Riemann problem defined at cell interfaces. The local Riemann problems may be solved analytically, if desired. However, existence of exact solutions to the Riemann problems is not always trivial; in many cases, iterative methods are employed to solve for intercell fluxes. Alternatively, solutions to local Riemann problems can be solved approximately with the original motivation that it can provide the solutions more cheaply than exact solvers. Since the pioneering work by [10,11], the research on approximate Riemann solvers has received considerable progress over the last few decades and formed a foundation for extensive studies on shock capturing.

The combination of Godunov-type schemes with approximate Riemman solvers are widely adopted and become a standard for benchmarking of any new schemes. Among those Roe's scheme [10], Harten–Lax–van Leer (HLL) schemes [3] and its variants are popular choices in many computations. Despite huge success over a large class of problems, there are fundamental deficiencies of the Godunov-type framework, especially for very strong shock applications. In [1] it reported and classified failures of various approximate Riemann solvers on multidimensional problems. The limitations of the Riemann solvers to shock-capturing properties was catalogued according to their failing modes including expansion shock, negative internal energy, slowly-moving shock, the carbuncle phenomenon, kinked Mach stem, and odd–even decoupling. Details of these problem can be found in the reference cited therein [1] in which the author proposed a strategy of using combined fluxes with more dissipative flux such as HLLE in the shock area. In [2], the so-called shock instability in multidimensional

* Corresponding author. Tel.: +65 6419 1591; fax: +65 6467 4350.
E-mail address: nguyenvt@ihpc.a-star.edu.sg (V.-T. Nguyen).

0045-7930/$ - see front matter © 2012 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.compfluid.2012.09.011

flows and the nonexistence of a solution for strong receding flows are major problems in such failings of approximate Riemann solvers. The failures were attributed to the dissipative term in mass fluxes defined in [2]; thus it was suggested that making the pressure contribution in dissipative terms vanish by controlling the magnitude of wave speed forms a shock-stable scheme.

Having mentioned the failures of approximate Riemann solvers in the framework of Godunov-type schemes, it is worth noting that low resolution is also considered one of the biggest shortcomings of such schemes. For first order methods, it requires very fine grids for better resolutions, particularly for shock wave interactions. Higher order accurate methods are thus desirable for numerical simulations of high speed compressible flows. Extension of first order finite volume methods to higher order is a favourable option since it only requires one to employ higher order reconstructions of solutions before feeding them to local Riemann solvers. While second order finite volume schemes have been widely adopted in many industrial codes, higher-than-second-order methods remains a formidable challenge for large scale problems, especially the construction of such schemes on unstructured grids. There has been much progress in this research direction over the last few decades including the development of essentially non-oscillatory (ENO) methods, discontinuous Galerkin methods [13] and their variants. However, it is necessary to stabilize the solution to eliminate oscillations due to high order approximations as discontinuities develop in the solutions. In many cases, the inherent diffusion from numerical discretizations is not sufficient and a special treatment is required to prevent oscillations from growing in the solutions. Among various treatments, artificial viscosity and limiters are frequently employed to stabilize shocks and discontinuities. In using artificial viscosity [12], numerical dissipation is directly added into systems to help stabilize the discretization. The approach has been widely used with successful results for shock capturing in various discretization methods; however, adding viscosity certainly reduces stability of the system especially for very strong shock cases. Alternatively, in using limiters, the solution is stabilized by reducing the interpolating order across the shocks; therefore it may affect the order of accuracy in the vicinity of the shock front. Essentially, employment of stabilizing approaches will introduce deficiencies to high order discretizations. Despite considerable progress in deriving high order methods for high speed compressible flows, one has yet found robust and efficient remedies fully addressing all the above issues related to this type of applications.

In this work, we first review our numerical method of finite volume discretization for compressible Navier–Stokes flows on unstructured grids. In the next section, a brief description of the method is given. The edge-based vertex-centered finite volume approach requires the construction of dual mesh by connecting edge midpoints, element centroids, face centroids in such a way that only one grid node is contained in each control volume. Edge-based data structure together with median dual control volume efficiently facilitate discretization of inviscid and viscous fluxes of compressible Navier–Stokes equations. In particular, HLLC flux scheme is employed as an approximate Riemann solver for solving flux functions at volume interfaces. The HLLC scheme is appended by a second order approximation of local Riemann states; thus obtaining second order accuracy. The second order reconstruction in turn makes the scheme unstable for strong shocks and discontinuities. In the next section, various treatments to stabilize the solutions are proposed and implemented including various slope limiters and a hybrid HLLC approach to overcome instability of the second order method for strong shocks. These remedies essentially reduce the approximation to first order accuracy in the vicinity of discontinuities. Therefore, we proposed an robust and effective mesh adaptive approach to increase the resolution in areas of steep gradients. In our proposed approach, a solution sensor is first designed to detect regions of high gradient; subsequently a robust adaptivity procedure was applied to regenerate three dimensional grid in those regions. Employing this local mesh adaptivity strategy, it is able to enhance the resolution of solutions in the presence of strong shocks with affordable computational cost.

2. Unstructured grid finite volume discretization

2.1. Governing equations

The unsteady inviscid compressible flows are governed by the time-dependent, Euler equations on a three-dimensional Cartesian domain $\Omega \subset \mathbb{R}^3$, with surface $\partial\Omega$, can be expressed in integral form as

$$\int_\Omega \frac{\partial \boldsymbol{U}}{\partial t}\,\mathrm{d}\boldsymbol{x} + \int_{\partial\Omega} \boldsymbol{F}_j n_j\,\mathrm{d}\boldsymbol{x} = \boldsymbol{0}, \tag{1}$$

where the conventional summation is employed and n_j is the outward unit normal vector to $\partial\Omega$. The unknown vector of the conservative variables and inviscid flux tensor are given by

$$\boldsymbol{U} = \begin{pmatrix} \rho \\ \rho u_1 \\ \rho u_2 \\ \rho u_3 \\ \rho\epsilon \end{pmatrix}, \quad \boldsymbol{F}_j = \begin{pmatrix} \rho u_j \\ \rho u_1 u_j + p\delta_{1j} \\ \rho u_2 u_j + p\delta_{2j} \\ \rho u_3 u_j + p\delta_{3j} \\ u_j(\rho\epsilon + p) \end{pmatrix}. \tag{2}$$

Here ρ denotes the fluid density, u_i the i'th component of the velocity vector and ϵ the specific total energy. The system is closed by assuming the gas to be calorically perfect, thus setting

$$p = \rho RT, \tag{3}$$

and

$$\epsilon = c_v T + \frac{1}{2} u_k u_k, \tag{4}$$

where R is the real gas constant and $c_v = c_p - R$ is the specific heat at constant volume. In this expression, c_p is the specific heat at constant pressure. Throughout this work, the ratio of the specific heats,

$$\gamma = \frac{c_p}{c_v}, \tag{5}$$

is set to $\gamma = 1.4$ which is the value for air at standard conditions.

2.2. Edge-based vertex-centred unstructured FV method

The computational domain Ω is subdivided into a set of non-overlapping tetrahedral elements using a Delaunay mesh generation process with automatic point creation. To enable the implementation of a cell vertex finite volume solution approach, a median dual mesh is constructed by connecting edge midpoints, element centroids and face centroids such that only one node is present in each control volume. Each edge of the grid is associated with a segment of the dual mesh interface between the nodes connected to the edge. The dual mesh interface inside the computational domain surrounding node I is denoted Γ_I, while the parts of the dual situated on the computational boundary are termed Γ_I^B, so that $\Gamma_I \cap \Gamma_I^B = \emptyset$. The triangular facets which define the control volume interface surrounding node I are denoted by Γ_I^K. In three dimensions, the segment of the dual mesh associated with an edge is a surface. This surface is defined using triangular facets, where each facet is connected to the midpoint of the edge, a neighbouring element centroid and the centroid of an element face connected to the edge. With this dual mesh definition, the control