Computers and Fluids 120 (2015) 111–125

Contents lists available at ScienceDirect

Computers and Fluids

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

Unsteady compressible flow computations using an adaptive multiresolution technique coupled with a high-order one-step shock-capturing scheme

Christian Tenaud [a,*], Olivier Roussel [b], Linda Bentaleb [a,1]

[a] LIMSI, UPR CNRS 3251, Bât 508, Rue John Von Neumann, 91403 Orsay Cedex, France
[b] CORIA, UMR CNRS 6614, Université de Rouen et INSA de Rouen, Avenue de l'Université, BP 12, 76801 Saint Etienne du Rouvray Cedex, France

ARTICLE INFO

Article history:
Received 21 April 2015
Revised 1 July 2015
Accepted 12 July 2015
Available online 6 August 2015

Keywords:
Adaptive mesh refinement
Multiresolution
Finite volume
Shock-capturing scheme
Compressible flow
Shock-wave

ABSTRACT

This article deals with the development of adaptive multiresolution coupled with a one-step shock-capturing scheme for the numerical simulation of unsteady compressible flows in the transonic and supersonic regimes with high frequency oscillations. The discretization of the convective terms is based on a coupled time and space approach by using a one-step (OS) scheme, developed following the Lax–Wendroff approach by correcting the successive modified equations. A monotonicity preserving (MP) criterion is added in order to locally relax the TVD constraints for such schemes. The adaptive strategy relies on the Harten cell-average multiresolution analysis, with a dynamical data structure organized as a graded tree that dynamically evolves in time. We apply the method to several prototype test-cases of shock-wave propagation interaction. We validate this approach on 2D inviscid advection of a vortex. We then present 2D viscous test-cases of shock-shear layer interactions and a 3D spherical Riemann problem to demonstrate the capability of the present method. Results demonstrate that 7th order OSMP schemes coupled with adaptive grid refinement gives very accurate results in comparison with more classical schemes applied on a single grid. We then propose an appropriate MR threshold parameter value that ensures accurate results while achieving drastic gains on the CPU time and memory usage.

1. Introduction

In the high speed flow regime, many aerodynamic configurations involve interactions between shock waves and turbulence such as, for instance, within air intakes, compressor or turbine configurations where shock wave/turbulent shear layer (*e.g.* boundary layer) interactions occur. An accurate prediction of such interactions is of importance in effective design of transsonic or supersonic vehicles since they greatly affect the aerodynamic loads. At the present time, it is commonly admitted that advanced numerical simulations (mainly LES) are powerful tools for accurate predictions of shock wave turbulent shear layer interactions, including large-scale flow phenomena [18,23]. In these approaches, the quality of the solutions depends not only on the capability of the numerical scheme associated with the sub-grid modelings in LES but also on the ability of the computational grid to capture the governing dynamical process. In fact, when dealing with shock waves, LES computations must however use numerical schemes which can both represent small scale structures with the minimum of numerical dissipation, mainly to minimize the interaction with the sub-grid scale model, and capture discontinuities with robustness [15,42]. Nevertheless, some phenomena could not be accounted by sub-grid modeling and accurate schemes coupled with locally very fine grid are needed to recover a high quality of the solution. For instance, according to the theoretical developments in the Linearized Interaction Approximation, the shock-wave/turbulence interaction phenomenon requires the correct prediction of the shock wave deformation occurring at small scales. These small scale shock deformations which could not be accounted by LES modeling, need locally very fine grid. The production of vorticity through baroclinic effect is also a phenomenon largely encountered in real flow physics that could not be accounted by sub-grid modeling and needs accurate numerical scheme on grid tightened in the production regions. These examples show that it is necessary to dynamically refine the grid locally, in the regions where the unsteady phenomena occur. It then motivates the introduction of self-adaptive discretization, as the solution may be over-resolved in

* Corresponding author. Tel.: +33 1 69 85 80 87; fax: +33 1 69 85 80 88.
E-mail addresses: Christian.Tenaud@limsi.fr (C. Tenaud), Olivier.Roussel@coria.fr (O. Roussel), linda.bentaleb@yahoo.fr (L. Bentaleb).
[1] Present address: Bertin Technologies, 10 B avenue Ampère, 78180 Montigny le Bretonneux, France.

http://dx.doi.org/10.1016/j.compfluid.2015.07.025
0045-7930/

large subsets of the computational domain when using equidistant fine grids. Therefore, to be efficient in terms of CPU time and memory usage, a mesh refinement method must be employed to save grid points in smooth regions and to concentrate them in the regions where phenomena (discontinuity, vorticity production, ...) occur. To be adequate for the DNS or LES approaches, the mesh refinement techniques must be based on multi-resolution analysis(MRA) that provides error estimates on the solution. This paper aims at contributing to the development and assessment of mesh refinement techniques based on error estimates coupled with high-order shock capturing schemes to capture small scale mechanisms, encountered in many aerodynamic flows, that have no concern with classical subgrid scale models.

In the literature, most numerical integrations that have been developed up to now can be divided into two classes: on one hand, the coupled space-time methods ; on the other hand, the methods based on separate time and space discretizations.

Most separate time-space methods are based on high-order multi-stage Runge-Kutta (RK) time integrations. At each stage, a high-order space discretization is applied, which usually contains in the flux computations a limiting procedure to prevent spurious oscillations. While a lot of work on separate time-space methods is still under investigation, the most commonly used of these space schemes are the ENO/WENO family [22,35–37]. It has been shown that these schemes are very accurate in smooth regions, capture very well the shock-waves but show a too diffusive behavior in the vicinity of contact discontinuities. Investigations have been undertaken to get better predictions using WENO-based schemes [1,21,31,32]. Moreover, these schemes are very expensive in terms of CPU time.

Coupled time-space schemes are usually developed following the Lax–Wendroff approach. Among them, the one step (OS) schemes have been developed, first for 1D (linear and non-linear) scalar equations, and then extended to multi-dimensional systems of non-linear equations (Daru and Tenaud [14,15]). Such schemes have a minimal stencil, and optimal non-oscillatory conditions, based on monotonicity-preserving constraints, can easily be implemented. These accurate numerical schemes offer a compromise between high accuracy in smooth regions and an efficient shock capturing technique. They provide very accurate results, which compare well to high-order separate time-space classical schemes, at a lower cost [15].

Besides the numerical scheme, the quality of solutions also depends on the capability of the computational grid to capture the governing dynamical mechanisms. In that sense, adaptive techniques for problems exhibiting locally steep gradients or shock-like structures have been developed since the end of the 1970s. Historically, adaptive methods like multi-level adaptive techniques(MLAT) (Brandt [9]) or adaptive mesh refinement (AMR) (Berger et al. [3–5]) were the first to achieve this goal, using a set of locally refined grids where steep gradients of high truncation errors are found. However, the data compression rate is high where the solution is almost constant, but remains low where the solution is regular. To overcome this difficulty, adaptive multiresolution methods, based on Harten's pioneering work [20], have been developed for 1D and 2D hyperbolic conservation laws (Cohen et al. [13], Müller et al. [19]). They have then been extended to 3D parabolic problems (Roussel et al. [34]). First simulations of 3D supersonic flows in the laminar regime using adaptive multiresolution methods were performed by Bramkamp et al. [7,8], with separate RK/ENO time-space discretizations. It has been shown in these papers that a high compression rate can be reached for solutions with inhomogeneous regularity. For an overview on adaptive multiresolution techniques, we refer to the books of Cohen [10] and Müller [26].

This paper aims at evaluating in practical situations the capability of the multiresolution adaptive technique coupled with a one-step shock capturing scheme to recover elementary physical mechanisms by achieving gains in both CPU time and memory usage compared to single grid computations. Numerical simulations are conducted on both inviscid and viscous compressible flows with high frequency oscillations in the transonic and supersonic regimes. As far as there could exist a competition between the discretization error of the scheme and the perturbation error introduced by the MR technique, we use several approximation orders of the OSMP scheme on several grids. The question that arises is: is it better to employ a low order (at least 2nd order) scheme on a very refined grid than use a high order scheme on a coarse grid? Through comparisons with 2nd and 3rd order schemes, we then explore the efficiency of a high order scheme coupled with the MR technique. We then propose an appropriate MR threshold parameter value that ensures accurate results, while achieving drastic gains on the CPU time and memory usage.

The paper is organized as follows. After a brief review of the governing equations (Section 2), we present in Section 3 the numerical approach used in this work: the so-called OSMP scheme (based on coupled time and space integration with MP constraint). Section 4 is dedicated to detail the multiresolution procedure. The evaluation of the method is then presented in Section 5 on several numerical results, 2D and 3D inviscid configurations and 2D configurations where viscous effects are present. Finally we conclude and present perspectives for future works.

2. Governing equations

We consider the Navier–Stokes equations expressed in dimensionless form using Cartesian coordinates:

$$\mathbf{w}_t + \nabla \cdot (\mathbf{f}^E(\mathbf{w}) - \mathbf{f}^V(\mathbf{w}, \nabla \mathbf{w})) = 0, \tag{1}$$

where $\mathbf{w} = (\rho, \rho\mathbf{u}, \rho E)^t$ is the vector of the conservative variables, using the classical notations, and $\mathbf{f}^E(\mathbf{w})$ and $\mathbf{f}^V$ are the Euler and the viscous fluxes respectively:

$$\mathbf{f}^E = \begin{pmatrix} \rho\,\mathbf{u} \\ \rho\,\mathbf{u}\otimes\mathbf{u} + \dfrac{P}{\gamma M_0^2}\,\mathbb{I} \\ \rho\,\mathbf{u}\,E + \mathbf{u}\dfrac{P}{\gamma M_0^2} \end{pmatrix}, \tag{2}$$

$$\mathbf{f}^V = \begin{pmatrix} 0 \\ \sigma \\ \mathbf{u}\cdot\sigma + \boldsymbol{\Psi} \end{pmatrix}, \tag{3}$$

with the strain rate tensor

$$\sigma = \frac{\mu}{\mathrm{Re}}\left(\nabla\mathbf{u} + \nabla^t\mathbf{u} - \frac{2}{3}\nabla\cdot\mathbf{u}\,\mathbb{I}\right),$$

and the heat flux

$$\boldsymbol{\Psi} = \frac{\mu}{(\gamma-1)\,\mathrm{Re}\,\mathrm{Pr}\,\mathrm{M}_0^2}\,\nabla T.$$

In addition, a perfect gas law is needed:

$$\frac{P}{\gamma\,M_0^2} = (\gamma-1)\left[\rho\,E - \frac{1}{2}\,\rho\,\mathbf{u}\cdot\mathbf{u}\right], \tag{4}$$

$$T = \frac{P}{\rho}, \tag{5}$$

with ρ the fluid density, $\mathbf{u}$ the velocity vector, P the static pressure, T the static temperature, E the total energy per unit of mass and μ the dimensionless dynamic viscosity.

These equations are written in dimensionless form using the reference values of the density (ρ_0), the velocity (v_0), and the length scale (L_0). The Reynolds number is based on the reference values: