Applied Mathematics and Computation 336 (2018) 368–393

Contents lists available at ScienceDirect

Applied Mathematics and Computation

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

Low-Mach number treatment for Finite-Volume schemes on unstructured meshes

Nicholas Simmonds[a], Panagiotis Tsoutsanis[a,*], Antonis F. Antoniadis[a], Karl W. Jenkins[a], Adrian Gaylard[b]

[a] *Centre for Computational Engineering Sciences, Cranfield University, Cranfield MK43 0AL, United Kingdom*
[b] *Jaguar Land Rover, Banbury Road, Gaydon, Warwick CV35 0RR, United Kingdom*

ARTICLE INFO

Keywords:
Low-Mach treatment
WENO
Unstructured mesh
Finite-Volume

ABSTRACT

The paper presents a low-Mach number (LM) treatment technique for high-order, Finite-Volume (FV) schemes for the Euler and the compressible Navier–Stokes equations. We concentrate our efforts on the implementation of the LM treatment for the unstructured mesh framework, both in two and three spatial dimensions, and highlight the key differences compared with the method for structured grids. The main scope of the LM technique is to at least maintain the accuracy of low speed regions without introducing artefacts and hampering the global solution and stability of the numerical scheme. Two families of spatial schemes are considered within the k-exact FV framework: the Monotonic Upstream-Centered Scheme for Conservation Laws (MUSCL) and the Weighted Essentially Non-Oscillatory (WENO). The simulations are advanced in time with an explicit third-order Strong Stability Preserving (SSP) Runge–Kutta method. Several flow problems are considered for inviscid and turbulent flows where the obtained solutions are compared with referenced data. The associated benefits of the method are analysed in terms of overall accuracy, dissipation characteristics, order of scheme, spatial resolution and grid composition.

1. Introduction

One of the most challenging parts of high-resolution numerical schemes is that they have to maintain adaptivity throughout the solution. Adaptivity, in the sense of identifying regions of sharp gradients, often encountered in compressible flows as well as preventing or eliminating any spurious oscillations that can occur; but at the same time they should be adaptive and achieve high-order of accuracy in smooth regions of the flow. There is a delicate balance between the two requirements and is dependent upon the spatial discretization method, the shock-capturing algorithms, the grid types, the Riemann solvers, the time-stepping algorithms and the integration quadrature rules to name a few.

The numerical methods for unstructured grids have matured and numerous elegant approaches [1–14] and algorithms have been developed in the FV framework for a wide range of applications in Computational Fluid Dynamics. Other state-of-the-art approaches have been developed, such as the Discontinuous Galerkin (DG) [2,11,15–18], Spectral Finite-Volume (SFV) methods [12,19–23], Flux Reconstruction (FR) methods[14,24] that have been successfully applied for various flow problems.

* Corresponding author.
E-mail addresses: n.simmonds@cranfield.ac.uk (N. Simmonds), panagiotis.tsoutsanis@cranfield.ac.uk (P. Tsoutsanis), a.f.antoniadis@cranfield.ac.uk (A.F. Antoniadis), k.w.jenkins@cranfield.ac.uk (K.W. Jenkins), agaylar1@jaguarlandrover.com (A. Gaylard).

https://doi.org/10.1016/j.amc.2018.04.076

For the FV framework, the first class of high-resolution methods developed for unstructured grids included the Essentially Non-Oscillatory (ENO) type schemes [25,26], followed by the WENO type schemes [27–30]. In the WENO case, the high-order accuracy was achieved by non-linearly combining a series of high-order reconstruction polynomials arising from a series of reconstruction stencils. Recently, a class of WENO type methods [8,9] has been successfully extended to hybrid unstructured meshes with various geometrical shapes such as tetrahedrals, hexahedrals, prisms,and pyramids. WENO schemes can achieve very high-order of spatial accuracy across interfaces between cells of different types, and non-oscillatory profiles are produced for discontinuous solutions. This provides greater flexibility to handle complex geometrical shapes in an efficient and accurate manner.

For the majority of the FV numerical methods applied to compressible flows, the dissipation characteristics are proportional to the speed of sound, therefore the low Mach number features are damped by the numerical scheme as noted by Thornber et al. [31]. This is particular important at regions of the flow where the local Mach number is small such in the vicinity of the boundary layer and in vortices arising from shear layers.

There is a wealth of different approaches aiming to improve the dissipation characteristics of numerical methods for compressible flow equations, either by enabling their deployment for very low Mach number flows, or improving their resolution at low Mach number regions [32–42]. In the novel approach of Rieper [41,42], it was shown through an one-dimensional analysis that the right amount of artificial viscosity on each individual characteristic variable is a prerequisite for an upwind scheme to approximate low Mach number flows correctly. A low diffusion preconditioning scheme was developed by Shen et al. [40] using 5th-order WENO scheme, and significant benefits in terms of accuracy and efficiency were noted for low-Mach number flows, as well transonic and supersonic flows.

A thorough analysis of various Roe Riemann solver [43] modifications for low Mach number flows was performed by Li and Gu [38], highlighting the dependence on the order of the coefficient of the velocity difference term and pressure difference term, along with some rules for constructing numerical schemes for all-speed flows. A low-dissipation version of Roe Riemann solver [43] was introduced by Oßwald et al. [39] and compared with the approach of Thornber et al. [31]; the former was modifying only the dissipation term in the numerical flux function, in contrast to the approach of Thornber et al. [31], where the evaluation of the physical fluxes is modified. The latter approach exhibited superior behaviour for the Decaying Isotropic Turbulence (DIT) test problem. Additionally, a non-physical high dissipation of energy was noted when using a tetrahedral mesh for the same test case with a second-order FV scheme. Another novel approach of Qu et al. [33] entailed the development of a new Roe-type scheme labelled RoeMAS, that exhibited high-resolution for low Mach number flows as well as robustness against odd-even decoupling.

The work of Nogueira et al. [32], presents the application of a Moving Least Squares (MLS) FV formulation, in conjunction with a low-Mach number fix and a slope limiter. Grid dependency of the schemes was assessed, demonstrating that even high-order schemes can benefit from the low-Mach number fix of Rieper [41,42]. It was highlighted that the accuracy problem of FV schemes for low Mach number flows can be alleviated by using high-order discretization schemes. The Discontinuous Galerkin (DG) schemes exhibit a similar accuracy problem to the FV schemes as shown by Bassi et al. [44], where it is shown that preconditioning improves accuracy and efficiency of DG schemes in the low Mach number regime.

All of the aforementioned approaches generally involve structured grids, or quadrilateral dominant meshes in 2D. Another new aspect, that was presented by Rieper and Bader [45], is that low Mach number accuracy of FV schemes is dependent on the cell geometry, since when applied on a triangular grid, the accuracy problem disappears. A comprehensive asymptotic analysis of this interesting phenomenon for the first-order Roe scheme [43] revealed that the leading-order velocity component normal to a cell edge does not jump, and that the arbitrary orientation of these triangular cells leaves enough degrees of freedom for the velocity field to represent a physical flow. This study did not include higher-order schemes for triangular meshes. However, the second-order Roe scheme [43] on unstructured triangular grids led to completely wrong results. It was assumed that the reconstruction process prevents the establishment of a continuous normal velocity component introducing the inaccurate pressure field. For higher-order schemes a smoother reconstruction which seems to prevent the jumps of the normal velocity component and with it the accuracy problem was identified.

The work in this paper is a revision of the approach of Thornber et al. [31], since the original implementation of the method as it will be demonstrated can not be extended to unstructured grids. A similar approach is introduced by Oßwald et al. [39], where the Roe Riemann solver was employed, however that study employed these schemes to second-order accurate FV schemes only for tetrahedral meshes. Additionally, it was noticed that extra dissipation was observed over the higher wave number range for the DIT test problem, which was not fully understood. To the best of the authors knowledge this is a first attempt to evaluate the characteristics and the performance of a low-Mach number fix using FV methods for unstructured meshes of various element types for 2D and 3D inviscid and turbulent compressible flows, while also utilising higher-order schemes. In addition, the challenges associated with the modifications are assessed and guidelines are provided for further development of these techniques. The compactness of the proposed scheme following the philosophy of Thornber et al. [31] can be utilised with any Riemann solver in order to remove the Mach number dependence, and improve the resolution at low Mach number regions of the flow. The original LM treatment proposed by Thornber et al. [31] is not directly transferable to any grid-type since different mesh elements have different dissipation characteristics, therefore a unified treatment is implemented that is suitable for all element types and through the computational results obtained the difference between the original treatment, and the modified one are presented. Finally, a desirable feature of this treatment is the efficient implementation in any compressible code, for any numerical scheme that uses a Riemann solver with negligible additional computational expense.