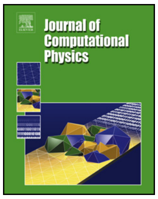
Journal of Computational Physics 362 (2018) 220–242

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp



A novel finite volume discretization method for advection–diffusion systems on stretched meshes

D.G. Merrick, A.G. Malan [*,1], J.A. van Rooyen

Mechanical Engineering Department, University of Cape Town, South Africa

ARTICLE INFO

Article history:
Received 19 August 2016
Received in revised form 23 November 2017
Accepted 12 February 2018
Available online 14 February 2018

Keywords:
Finite volume
Spatial discretization
Accuracy
Incompressible flow
Non-equispaced unstructured meshes

ABSTRACT

This work is concerned with spatial advection and diffusion discretization technology within the field of Computational Fluid Dynamics (CFD). In this context, a novel method is proposed, which is dubbed the Enhanced Taylor Advection–Diffusion (ETAD) scheme. The model equation employed for design of the scheme is the scalar advection–diffusion equation, the industrial application being incompressible laminar and turbulent flow. Developed to be implementable into finite volume codes, ETAD places specific emphasis on improving accuracy on stretched structured and unstructured meshes while considering both advection and diffusion aspects in a holistic manner. A vertex-centered structured and unstructured finite volume scheme is used, and only data available on either side of the volume face is employed. This includes the addition of a so-called mesh stretching metric. Additionally, non-linear blending with the existing NVSF scheme was performed in the interest of robustness and stability, particularly on equispaced meshes. The developed scheme is assessed in terms of accuracy – this is done analytically and numerically, via comparison to upwind methods which include the popular QUICK and CUI techniques. Numerical tests involved the 1D scalar advection–diffusion equation, a 2D lid driven cavity and turbulent flow case. Significant improvements in accuracy were achieved, with L_2 error reductions of up to 75%.

1. Introduction

Finite volume methods are today the most widely used for the solution of the Navier–Stokes equations. Industrial problems are typically computationally demanding with complex geometries being prevalent. This means that the accuracy of the numerical methods used on the required stretched anisotropic computational meshes is of importance. Inherent to computational efficiency is discretization accuracy, with the advection and diffusion terms being of considerable importance. This is the focus of this paper, with specific emphasis to stretched meshes as these are often employed in practice. A vertex-centered finite volume framework will be employed similar to Malan and Lewis [1].

CFD advection discretization methods have to date been intensively researched, with various algorithms being the result. The majority of methods can be expressed within the context of either the Normalised Variable Diagram (NVD) [2], or the Total Variation Diminishing (TVD) scheme [3,4] (though this is not straightforward with the Characteristic Based Split

---

[*] Corresponding author.
E-mail address: arnaud.malan@uct.ac.za (A.G. Malan).

[1] Professor and South African Research Chair in Industrial CFD, Industrial CFD Research Group, Dept. of Mechanical Engineering, University of Cape Town, Rondebosch, 7701 South Africa.

https://doi.org/10.1016/j.jcp.2018.02.025

(CBS) scheme [5,6]). Both of these approaches have associated criteria that must be met by a scheme in order to ensure boundedness. The Convective Boundedness Criteria (CBC) [7] is used for the NVD, while the Flux-Limiting Diagram [8] is used with the TVD scheme. Many schemes created before the popularisation of the two advection philosophies have subsequently been converted to be expressed as either an NVD or TVD scheme. The grouping of schemes into the two approaches has naturally helped analysis and comparison. It is also possible, and fairly simple, to express NVD schemes as TVD schemes, and vice versa.

The above advection discretization methods have historically been dominated by asymmetric, upwind-biased schemes that are defined as either linear or non-linear. Many non-linear schemes were developed by adapting existing linear schemes to conform to developed boundedness criteria. As such, linear schemes form the foundations of many advection discretization schemes. A convenient way of describing a linear method which also enables easy analysis and comparison, is as a member of what is known as the κ-Upwind class [9,10]. Of the many upwind methods developed, some have attained a greater degree of popularity. This paper focuses on those preferred for their accuracy, namely the Cubic-Upwind-Interpolation (CUI, $\kappa = \frac{1}{3}$) [11] and Quadratic-Upwind-Interpolation variants (QUICK, $\kappa = \frac{1}{2}$) [12,13]: CUI is well known to be 3rd-order accurate on equispaced meshes. QUICK is 2nd-order accurate at vertexes. However, both methods deteriorate to 1st-order accuracy on stretched meshes.

The Normalised Variable and Space Formulation (NVSF) scheme [14,15] is, to the knowledge of the authors, the only documented finite volume scheme designed to maintain accuracy on non-equispaced grids. This method is marketed as achieving 3rd-order accuracy at a face, and 2nd-order accuracy at a node, for both equi- and non-equispaced grids. The NVSF method accounts for mesh stretching by calculating a nodal stretching ratio of the three nodes contained within the NVD, and using this ratio to optimise the κ-value.[2] As a result, the accuracy at the face of the NVD is maximised. Interestingly, though understandably, NVSF simplifies to QUICK on an equispaced grid (understandable because QUICK maximises accuracy at a face). In its original form, the NVSF method was applied to only structured meshes with collinear upwind and downwind nodes (the scheme is extended to general unstructured meshes in this paper). In addition, Waterson and Deconinck [16] dismiss the NVSF scheme, stating that it “increases the complexity of the resulting NVD scheme”. While this may be strictly true, it is still possible that the NVSF results in an increase in efficiency. We are therefore strong proponents of the method.

Industrial CFD involves a large range of practical applications, many of which require the use of non-equispaced structured and unstructured meshes. As described above, the vast majority of finite volume advection discretization schemes were, however developed using equispaced meshes as basis. In advancing on this, this paper intends to table a novel philosophy aimed at increasing accuracy with regards to two key aspects of advection–diffusion problems. Firstly, non-equispacing is assumed as fundamental when designing the spatial discretization algorithms. Secondly, consideration is given to treating the advection and diffusion components in a more holistic manner. This stands in contrast to previous work which has taken a mutually exclusive attitude. To this end, this work aims to develop a novel, improved, and industrially relevant CFD advection discretization methodology for incompressible viscous flow systems. The methodology developed will be considered “improved” if:

- It results in a discretization methodology that displays a greater level of efficiency than that of currently used methodologies.
- It maintains both local and global conservation.
- It offers stability similar to that of the QUICK or NVSF methods.

It is worth noting that sparse implicit solution methods are employed in this work, with the solution of the sparse system being computationally dominant.

A methodology’s “efficiency” in CFD is typically taken as the ratio of accuracy to computational cost. This paper will focus on increasing the accuracy. When used as part of an implicit flow solver, however, it should not result in any significant increase in cost per iteration. Once again, in designing an industrially relevant methodology it is important, from an efficiency and parallel processing point of view, that locality of data is maintained. This is interpreted here to mean that only neighbouring nodal data is available when discretizing a face value. Accordingly, the order-of-accuracy (OoA) at the face is employed to develop the new scheme which – dubbed the Enhanced Taylor Advection–Diffusion (ETAD) method. The use of face accuracy as a metric for improved accuracy is viewed as more pragmatic (not a function of element topology), while it naturally results in a conservative scheme.

The section to follow, tables a critical error analysis of three existing methods made popular due their accuracy – these methods are CUI, QUICK and NVSF. Thereafter, the ETAD method is developed, followed by its analysis and evaluation. For this purpose the 1D advection–diffusion and 2D lid-driven cavity test cases are considered for a range of stretching ratios and advection–diffusion ratios (Reynolds for the 2D flow problem). In addition, the separated, turbulent flow over an aerofoil, is finally modelled to assess industrial robustness. A conclusion and appendices completes the paper.

---

[2] For ease of comparison the NVSF method is cast into κ-Upwind format as per A.