Computers & Fluids 101 (2014) 220–232

Contents lists available at ScienceDirect

Computers & Fluids

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

Accuracy analysis of unstructured finite volume discretization schemes for diffusive fluxes

Alireza Jalali *, Mahkame Sharbatdar, Carl Ollivier-Gooch

Department of Mechanical Engineering, The University of British Columbia, 2054-6250 Applied Science Lane, Vancouver, BC V6T 1Z4, Canada

ARTICLE INFO

Article history:
Received 11 February 2014
Received in revised form 29 May 2014
Accepted 5 June 2014
Available online 18 June 2014

Keywords:
Finite volume methods
Diffusive flux
Unstructured meshes
Truncation error
Discretization error

ABSTRACT

Numerical errors form a major source of uncertainty in CFD simulations. This source of error, which is more significant on unstructured meshes, can be alleviated by a careful choice of discretization scheme. In the context of finite volume methods, there are many suggestions for the discretization of diffusive fluxes appearing in viscous flow simulations. In this paper, a wide range of discretization schemes commonly used for diffusive flux approximation in cell-centered unstructured finite volume solvers are compared in terms of discretization and truncation errors. In addition, a novel eigenanalysis tool is proposed to relate these two forms of numerical error to each other and to interpret the error behaviors obtained by each scheme. The error comparisons are performed on unstructured meshes consisting of both isotropic and anisotropic triangles. Our results suggest that adding a solution jump term to the baseline face gradient (determined as the average of two adjacent cell gradients) provides the most accurate approximation for diffusive fluxes. Also, this term produces sufficient damping to stabilize the discretization even on highly skewed meshes.

1. Introduction

The design of aircraft, from commercial and military transport to combat airplanes, depends increasingly on the use of computational fluid dynamics. In particular, computational aerodynamics, which is concerned with the accurate prediction of airplanes' force and moment coefficients, is progressively being used as a practical tool in the design and optimization of aerodynamic objects. The numerical simulation of fluid flows is highly affected by the presence of three error sources: geometry modeling, physical modeling and numerical errors.

Geometry modeling error is unavoidable in those cases where complex geometry can not be accurately modeled. In those circumstances, some minor geometrical features of objects are ignored to facilitate the numerical procedure. As an example, the wing root fillet is often neglected in the CFD simulation of fluid flow around airplanes.

Recent studies have proved that numerical errors are at least as large as physical modeling errors for computations of the flow around transport aircraft. Mavriplis [1] carried out a grid convergence and sensitivity study on a wing-body configuration which was the subject of the second AIAA Drag Prediction Workshop (DPW). Physical modeling errors were quantified based on the sensitivity of computed drag coefficient to the formulation of the viscous terms and to the turbulence model. The results revealed that using the thin-layer approximation near the wall rather than solving the full Navier–Stokes equations affects the drag coefficient by less than 2%. Likewise, the sensitivity to the turbulence modeling was examined by varying the way by which the distance from wall is obtained. The variations in finding this quantity, which is important in Spalart–Allmaras turbulence model [2], rarely influenced the drag coefficient. On the other hand, using topologically different grids with equal resolutions near the trailing edge substantially changed the solution. These observations reinforced the notion that numerical errors are still the dominant source of error in most aerodynamic simulations especially for unstructured flow solvers. In fact, mesh features – including cell size, anisotropy, shape, connectivity, and variations between adjacent cells – can have an adverse interaction with discretization schemes and this interaction may affect the solution accuracy.

One way to control the accuracy of spatial discretization is improving the local quality of cells in an unstructured mesh by node displacement [3]. Although this approach is effective in reducing the local error in the solution, it negates the advantages of automatic unstructured mesh generation. However, the need for precise vertex movement can be alleviated provided that

* Corresponding author. Tel.: +1 6046002324.
E-mail addresses: arjalali@interchange.ubc.ca (A. Jalali), mahkame@interchange.ubc.ca (M. Sharbatdar), cfog@mech.ubc.ca (C. Ollivier-Gooch).

http://dx.doi.org/10.1016/j.compfluid.2014.06.008
0045-7930/

careful attention is given to the discretization schemes used to solve the governing equations.

In the context of finite volume methods, discretization schemes are divided into two major categories: convective and diffusive fluxes. Convective fluxes are those that depend only on the solution at the cell interfaces while diffusive fluxes are dependent upon the interface solution gradient. While there is a clear consensus over the use of flux difference splittings for convective fluxes [4], the choice of diffusive flux discretization and its effect on the accuracy of the solution is less understood.

Diskin et al. [5] compared the accuracy of a limited number of the node-centered and cell-centered discretization schemes for viscous (diffusive) fluxes with the aim of improving turbulent simulations. This comparison was done on a range of grids from regular to irregular composed of arbitrary mixtures of triangles and quadrilaterals. The comparison showed that there is little difference in accuracy between node-centered schemes and the best cell-centered schemes, but poorly-designed cell-centered schemes behave much worse. They also compared the node-centered and cell-centered unstructured finite volume discretization schemes for inviscid (convective) fluxes [6]. The schemes, being different in the least-squares gradient reconstruction, were compared for a constant coefficient convection equation, linear advection. The authors concluded that carefully-designed cell-centered schemes offer the best options for second-order discretization.

Nishikawa [7] introduced a general principle for constructing diffusion schemes which is applicable to various discretization methods, including finite volume, residual distribution, discontinuous Galerkin, and spectral-volume methods. This principle is derived based on a hyperbolic relaxation-system model for diffusion problems and results in a damping term, which is essential for effective high-frequency error damping and, in some cases, for consistency. He also demonstrated for the first time that the diffusion schemes with a finite difference term are special cases of the new damping scheme in the context of finite volume discretizations [8]. Moreover, he extended the damping scheme to the Navier–Stokes equations [9] with an optimal value of the damping coefficient which has already been obtained by Fourier analysis and truncation error analysis for scalar problems.

In this paper, we compare the accuracy of common second-order-accurate discretization schemes for calculating diffusive fluxes on unstructured meshes. In particular, we consider the Poisson equation as a model of viscous discretization and we use variations on the cell-centered finite volume approach. We describe in Section 2 a wide range of discretization schemes which differ in how cell gradients are combined to compute a face gradient; in the presence and form of finite difference correction terms to the gradient; and in treatment of the discontinuity at faces.

The initial source of numerical errors is the truncation error, defined as the difference between the continuous PDEs and the finite discretized equations. Truncation error plays an important role in error quantification since the truncation error can be used to estimate the discretization error that occurs during the approximate numerical solution of differential equations. Discretization error is defined as the difference between the exact solution of governing equations and the numerically approximated solution.

Roy [10] showed that the discretization error can be directly related to the truncation error by the error transport equation where the truncation error serves as a source term to the linearized system of equation whose solution is the discretization error. For a linear problem such as Poisson's equation, it is easy to show that the linear operator applied to the discretization error in the error equation is equivalent to the flux Jacobian. In Section 3, an eigendecomposition paradigm will be introduced where the truncation error is expanded as a combination of linearly independent right eigenvectors of the flux Jacobian matrix. The weights corresponding to the right eigenvectors differ by the choice of discretization scheme and influence the error obtained in the numerical solution. The distribution of these weights are helpful in explaining how the change of discretization scheme results in a smaller or larger discretization error.

In Section 4, the method of manufactured solution is used to compare the discretization and truncation errors produced by different schemes. The eigenanalysis tool is utilized to explain how the choice of schemes affects the eigenvalue spectra and weights associated with the eigendecomposition of error forms. It turns out that each class of discretization method alters these measures in a unique pattern that can be interpreted as the method signature. The accuracy analysis tests are performed on both isotropic and anisotropic unstructured triangular grids with known properties.

2. Discretization scheme

To discretize the flow equations using the finite volume method, the governing equations should be recast in fully conservative form as

$$\frac{\partial \vec{U}}{\partial t} + \nabla \cdot \vec{F} = S \tag{1}$$

in which U denotes the conservative solution vector, F is the flux vector and S represents the source term vector. Assuming the discretized physical domain does not change in time and integrating Eq. (1) over an arbitrary control volume, the 2D finite volume formulation of the governing equations can be written in the form of a surface and an area integral:

$$\frac{dU_i}{\mathrm{dt}} = -\frac{1}{A_{CV_i}} \oint_{CS_i} \vec{F} \cdot \hat{n}\, \mathrm{ds} + \frac{1}{A_{CV_i}} \int_{A_i} S\, dA = -R\left(\vec{U}\right) \tag{2}$$

The right hand side is called the residual which is comprised of the flux integral representing the spatial discretization corresponding to each control volume and the control volume average of the source term. The flux integral is calculated by the Gauss quadrature along the faces of the control volumes. In the present work, we consider the cell-centered approach in which the control volumes are formed as the cells of the mesh. To evaluate the numerical fluxes at the interfaces, primitive variables must be reconstructed to give a series expansion of the solution about the cell's reference point (x_i, y_i) to the desired order of accuracy:

$$\phi_i^R(x,y) = \phi|_i + \left.\frac{\partial \phi}{\partial x}\right|_i (x - x_i) + \left.\frac{\partial \phi}{\partial y}\right|_i (y - y_i) + \left.\frac{\partial^2 \phi}{\partial x^2}\right|_i \frac{(x - x_i)^2}{2} + \left.\frac{\partial^2 \phi}{\partial x \partial y}\right|_i (x - x_i)(y - y_i) + \left.\frac{\partial^2 \phi}{\partial y^2}\right|_i \frac{(y - y_i)^2}{2} + \ldots \tag{3}$$

In Eq. (3), ϕ_i is the value of the reconstructed solution and $\frac{\partial^{k+l}\phi_i}{\partial x^k \partial y^l}$ are its derivatives at the reference point of control volume i. A second-order accurate solution can be achieved by knowing the gradient of the solution at the cell's reference point and reconstructing a linear polynomial in the control volume.

Conservation of the mean within a control volume requires that

$$\frac{1}{A_i} \int_{A_i} \phi_i^R\, dA = \bar{\phi}_i \tag{4}$$

By expanding the left-hand side of Eq. (4) term by term, one can easily show that

$$\frac{1}{A_i} \int_{A_i} \phi_i^R\, dA = \phi|_i + \left.\frac{\partial \phi}{\partial x}\right|_i \bar{x}_i + \left.\frac{\partial \phi}{\partial y}\right|_i \bar{y}_i + \left.\frac{\partial^2 \phi}{\partial x^2}\right|_i \frac{\overline{x^2}_i}{2} + \left.\frac{\partial^2 \phi}{\partial x \partial y}\right|_i \overline{xy}_i + \left.\frac{\partial^2 \phi}{\partial y^2}\right|_i \frac{\overline{y^2}_i}{2} + \ldots \tag{5}$$