Computers and Mathematics with Applications 72 (2016) 1096–1117

Contents lists available at ScienceDirect

Computers and Mathematics with Applications

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

A high-order finite volume method on unstructured grids using RBF reconstruction

Yilang Liu [a], Weiwei Zhang [a,*], Yuewen Jiang [b], Zhengyin Ye [b]

[a] School of Aeronautics, Northwestern Polytechnical University, No. 127 Youyi West Road, Xi'an 710072, China

[b] Department of Engineering Science, University of Oxford, Oxford, OX1 3PJ, UK

ARTICLE INFO

Article history:
Received 27 January 2016
Received in revised form 29 May 2016
Accepted 19 June 2016
Available online 18 July 2016

Keywords:
Finite volume method
Unstructured grids
High order scheme
RBF reconstruction method
Navier–Stokes equations

ABSTRACT

This paper proposes a high-order finite volume method based on radial basis function (RBF) reconstruction for the solution of Euler and Navier–Stokes equations on unstructured grids. Unlike traditional polynomial K-exact method, RBF method has stronger adaptability for different reconstruction stencils and more flexibility in choosing interpolating points. We expatiate on the detailed process of flow-field reconstruction by using multiquadric (MQ) basis function for the second-order and third-order schemes on unstructured triangular grids. Subsequently, we validate the accuracy order of RBF method through the numerical test case. Furthermore, the method is used to solve several typical flow fields. Compared with traditional K-exact high-order scheme, RBF method is more accurate and has lower numerical dissipation, which can obtain more elaborate and precise results.

1. Introduction

Researchers of computational fluid dynamics (CFD) have kept pursuing the accuracy of numerical simulation and the adaptability to complex configurations. Although finite difference method for high-order scheme performs well and has been successfully used in flow-field numerical simulation, it mainly implement on structured grids [1,2]. Unstructured grids have random data storage, disordered adjacent grid nodes and controllable distribution of elements and nodes. So, they have a natural ability to adapt to complex shapes and can easily control grid distribution to achieve adaptive meshes. Thus, it is significant to develop high order numerical scheme on unstructured grids for both fundamental theory research and practical engineering applications. Although the second-order accuracy schemes have played an important role in aircraft design and application, they still have large numerical dissipation and dispersion. For some complex flow problems, such as wave propagation, vortex-dominated flows including high-lift configuration, helicopter blade vortex interaction, as well as large eddy simulation and direct numerical simulation of turbulence, high-order accuracy schemes must be used to obtain more elaborate and detailed results [3]. Furthermore, as pointed out by Jameson [4], the use of second-order finite volume methods cannot, on realistic meshes, reliably predict complex separated, unsteady and vortex dominated flows, and future research should focus on high-order methods with minimal numerical dissipation for unstructured meshes. However, the development of higher-order finite volume method on unstructured grids is technically difficult [5,6].

The key issue of solving flow equations by finite volume method is to reconstruct the distribution of unknown variables at the interface of the control volumes using the cell averaged state variables. For the second-order scheme, one of the most widely used reconstruction methods is based on the Taylor series expansion first proposed by Barth [7,8]. This method

* Corresponding author.
E-mail address: aeroelastic@nwpu.edu.cn (W. Zhang).

http://dx.doi.org/10.1016/j.camwa.2016.06.024
0898-1221/

assumes that the solution is piecewise linearly distributed over the control volume. It implements a Taylor-series around the neighboring center of grid faces where only the linear term is retained, and then solves the equations by the least-squares approach to obtain the cell center gradient and values at the interface. Frink [9] proposed a simpler linear vertex reconstruction method that did not require explicit evaluation of the center grid variable gradient. Additionally, the gradient at the cell center can be approximated by a simple finite difference after determining the nodal values by inverse distance weighing. The above two methods belong to polynomial flow-field reconstruction method. Barth and Frederickson [10] extend it to high-order finite volume schemes, namely K-exact least-squares reconstruction method, which has been widely applied in computational dynamics [5,6,11–16].

Traditional polynomial reconstruction method is a kind of global fitting method, and the expanded item of Taylor polynomial strictly matches the accuracy order of interpolation method. Fewer interpolation points result in lower numerical accuracy, whereas more interpolation points result in over-fitting, especially for high-order finite volume method on unstructured grids. Unlike polynomial reconstruction method, Lancaster and Salkauskas [17] proposed a moving least-squares (MLS) method which is mainly used in curve and surface fittings. Afterward, CFD researchers successfully introduced it into solving flow-field governing equations [18–20]. MLS method is a local fitting method in which each interpolation base point corresponds with a polynomial shape function and a weight function. It is very convenient to use this method to control the fitting accuracy and smoothness by changing the degrees of shape functions and weight functions. This method has been thoroughly studied by Cueto-Felgueroso [18–20]. Moving Kriging (MK) method, which was first proposed by Gu [21], is also a candidate interpolation technique for flow-field reconstruction. Gu successfully constructed the shape function with MK interpolation instead of MLS and applied it to element-free Galerkin method for solving the weak form of the boundary value problem. Subsequently, the method was widely used in solid mechanic problems [22–25]. Chassaing [26] first introduced the MK interpolation in high-order finite volume method. The successive derivatives of the flow variables in each cell are deduced from the interpolation function, and MK-FV method is proved to be an interesting alternative for the development of high-order methodology for complex geometries. Radial basis function (RBF) interpolation method [27], because of its excellent performance of scattered data fitting and brief mathematical expression, has been widely used in many different domains, such as mesh deformation [28,29], nonlinear aerodynamic modeling [30,31], and partial differential equations solving [32–34]. Sonar [35,36] first applied the RBF interpolation method to finite volume scheme of ENO-type using the thin splines base function. In terms of solving partial differential equations, partial derivatives of equations can be approximately expressed by the linear combination of RBFs, proposed by C. Shu [37], which can be used for numerical solution by finite difference method.

This paper introduces RBF interpolation in flow-field reconstruction for high-order finite volume scheme on unstructured grids. We expatiate on the detailed process of achieving cell-centered high-order scheme for finite volume method by using MQ basis function. It is implemented in terms of three main aspects: the reconstruction of the flow variables at control volume interfaces, the selection of the reconstruction stencil and the treatment of the curved boundary condition. Above this, the accuracy order of RBF reconstruction method is validated through numerical test cases. Furthermore, the method is used to solve several inviscid and viscid typical flow fields, and meanwhile, the computed results are compared with the results from traditional polynomial reconstruction methods.

The outline of this paper is organized as follows. Section 2 briefly describes the finite volume formulation of the governing equations. The high-order discretization algorithm based on RBF reconstruction is detailed in Section 3. Section 4 presents a comprehensive assessment of the scheme by several numerical experiments for inviscid and viscid flows and the concluding remarks are drawn in Section 5.

2. Governing equations and high-order finite volume formulation

The integral form of two-dimensional Navier–Stokes equations can be written as:

$$\frac{\partial}{\partial t}\iint_{\Omega}\mathbf{Q}\,d\Omega+\oint_{\partial\Omega}\boldsymbol{F}(\mathbf{Q})\cdot\boldsymbol{n}\,d\Gamma=\oint_{\partial\Omega}\boldsymbol{G}(\mathbf{Q})\cdot\boldsymbol{n}\,d\Gamma \tag{1}$$

where Ω is the control volume; $\partial\Omega$ is the boundary of control volume; and $\boldsymbol{n}=(n_x,n_y)^T$ denotes the outer normal vector of the control volume boundary. The vector of conservative variables $\mathbf{Q}$, inviscid fluxes $\boldsymbol{F}(\mathbf{Q})=(\boldsymbol{F}_x(\mathbf{Q}),\boldsymbol{F}_y(\mathbf{Q}))$ and viscous fluxes $\boldsymbol{G}(\mathbf{Q})=(\boldsymbol{G}_x(\mathbf{Q}),\boldsymbol{G}_y(\mathbf{Q}))$ are given by the following equations:

$$\mathbf{Q}=\begin{Bmatrix}\rho\\ \rho u\\ \rho v\\ E\end{Bmatrix}\qquad \boldsymbol{F}_x(\mathbf{Q})=\begin{Bmatrix}\rho u\\ \rho u^2+p\\ \rho uv\\ u(E+p)\end{Bmatrix}\qquad \boldsymbol{F}_y(\mathbf{Q})=\begin{Bmatrix}\rho v\\ \rho uv\\ \rho v^2+p\\ v(E+p)\end{Bmatrix}$$

$$\boldsymbol{G}_x(\mathbf{Q})=\mu\begin{Bmatrix}0\\ 2u_x-\dfrac{2}{3}(u_x+v_y)\\ v_x+u_y\\ u\left(2u_x-\dfrac{2}{3}(u_x+v_y)\right)+v(v_x+u_y)+\dfrac{T_x}{(\gamma-1)\,\mathrm{Pr}}\end{Bmatrix}$$