Computers & Fluids 86 (2013) 483–489

Contents lists available at ScienceDirect

Computers & Fluids

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

Technical note

An investigation of mesh-less calculation for compressible turbulent flows

M. Namvar, A. Jahangirian *

Department of Aerospace Engineering, Amirkabir University of Technology, 424 Hafez Avenue, Tehran, Iran

ARTICLE INFO

Article history:
Received 6 June 2012
Received in revised form 1 August 2013
Accepted 8 August 2013
Available online 19 August 2013

Keywords:
Mesh-less method
Turbulent compressible flows
Taylor Least Square approximation
Green–Gauss approach

ABSTRACT

An investigation is carried out for mesh-less calculation of compressible turbulent flows. The capabilities of the Taylor Least Square method for calculation of spatial derivatives are evaluated by using different turbulence models. Results show that good agreement with analytical solution can be obtained when regular point distribution is used. However, numerical experiments show that using irregular point clouds within the domain may lead to inaccurate results. So an alternative approach is developed and its accuracy is investigated by solving various laminar and turbulent flows at transonic flow conditions.

1. Introduction

In spite of significant progress in theory and practice of flow solution for complex geometries, difficulties associated with mesh generation have remained as a major problem. A growing interest is then observed in using alternative methods which can resolve mesh generation difficulties. Mesh-less scheme refers to methods that use distribution of points instead of domain discretization by mesh.

The Finite Point Method proposed by Onate et al. [1] is formulated using the approximation techniques such as, *Least Square* (LSQ), *Weighted Least Squares* (WLS) or *Moving Least Squares* (MLS) to construct the derivatives. The WLS can be used in the forms of, *Taylor Least Square* (TLS) or *Polynomial Least Square* (PLS). In both methods the derivatives at any point are constructed by using points inside an influence region called neighboring points. Thus, mesh-less method does not need a mesh in a way like finite element or finite volume methods. PLS method involves expanding a polynomial function from the cloud points [2–4] while the TLS uses a Taylor series expansion instead of polynomial functions [5–10]. Katz and Jameson [11] compared TLS and PLS methods and showed that TLS is more efficient for a wide variety of flows. Sridar and Balakrishnan [10] used an upwind mesh-less solver for simulation of subsonic and transonic laminar flows. Hashemi and Jahangirian [8,9] used an implicit method with central discretization of convection terms to solve laminar viscous flows. They showed the superiority of their method compared with the CUSP (Convective Upwind and Split Pressure) method. Some researchers used hybrid solver including mesh-less method near the solid wall and a finite volume flow solver far from the viscous regions [5,6,8]. Most of these methods used an unstructured Cartesian mesh outside the viscous region and point cloud near the wall. Thus, there is not any cut-cell near the surface because point clouds are used in this region instead.

Despite the progress has been made for complex flow computations using different mesh based finite-volume and finite-element methods, only a few mesh-less approaches have been used for practical problems of high Reynolds number turbulent flows. Manicrisna and Balakrishnan [12] used an upwind mesh-less scheme together with algebraic Baldwin–Lomax turbulence model for flow computations. They also reported difficulties in the boundary layer region with TLS method.

The main objective of the present study is to extend the applicability of the mesh-less scheme presented in Ref. [9] to high Reynolds number turbulent flows. Different turbulence models are applied and the efficiency of the method is compared with experimental and alternative analytical data.

2. Governing flow equations

The non-dimensional differential form of the compressible Reynolds-Averaged Navier–Stokes (RANS) equations in two dimensions can be expressed in the conservative form as:

$$\frac{\partial w}{\partial t} + \frac{\partial f^I}{\partial x} + \frac{\partial g^I}{\partial y} = \frac{M_\infty}{\mathrm{Re}_\infty}\left[\frac{\partial f^v}{\partial x} + \frac{\partial g^v}{\partial y}\right], \tag{1}$$

* Corresponding author. Tel./fax: +98 21 64543223.
E-mail address: ajahan@aut.ac.ir (A. Jahangirian).

0045-7930/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.compfluid.2013.08.002

where w is the vector of conserved variables, f^I and g^I are the inviscid fluxes defined as:

$$w = \begin{pmatrix} \rho \\ \rho u \\ \rho v \\ \rho E \end{pmatrix}, \quad f^I = \begin{pmatrix} \rho u \\ \rho uu + P \\ \rho vu \\ \rho Eu + Pu \end{pmatrix}, \quad g^I = \begin{pmatrix} \rho v \\ \rho uv \\ \rho vv + P \\ \rho Ev + Pv \end{pmatrix}. \tag{2}$$

In the above notation ρ, u, v, P and E are the non-dimensional density, velocity components, pressure and total energy. Viscous flux terms f^V and g^V are defined as:

$$f^V = \begin{pmatrix} 0 \\ \tau_{xx} \\ \tau_{xy} \\ u\tau_{xx} + v\tau_{xy} - q_x \end{pmatrix}, \quad g^V = \begin{pmatrix} 0 \\ \tau_{xy} \\ \tau_{yy} \\ u\tau_{xy} + v\tau_{yy} - q_y \end{pmatrix}, \tag{3}$$

The Navier–Stokes equations are completed by the perfect gas equation of state:

$$P = \rho RT, \quad E = e + \frac{u^2 + v^2}{2}, \quad e = \frac{R}{\gamma - 1}T \tag{4}$$

2.1. Discretization of governing flow equations

The mesh-less algorithm is applied directly to the differential form of the governing equations. Here Taylor series Least Square method is used to discretize the equations as following:

$$\left.\frac{\partial \phi}{\partial x}\right|_i = \sum_{j=1}^{m} a_{ij}\Delta\phi_{ij}, \quad \left.\frac{\partial \phi}{\partial y}\right|_i = \sum_{j=1}^{m} b_{ij}\Delta\phi_{ij}, \tag{5}$$

where $\Delta\phi_{ij} = \phi_j - \phi_i$ and ϕ could be any function.

The Navier–Stokes equations can be discretized in space as:

$$\frac{\partial \mathbf{w}_i}{\partial t} + \sum_{j=1}^{m}\left(a_{ij}F^I_{ij} + b_{ij}G^I_{ij}\right) = \frac{M_\infty}{\mathrm{Re}_\infty}\sum_{j=1}^{m}\left(a_{ij}F^v_{ij} + b_{ij}G^v_{ij}\right). \tag{6}$$

The dissipative nature of the viscous terms presented in this equation is not sufficient to damp instabilities to construct a stable scheme. One may use the mid-point of edge ij instead of point j and add diffusive terms D by defining the mid-point inviscid flux as [6]:

$$F^I_{ij+1/2} = \frac{1}{2}\left(F^I_i + F^I_j\right) - \frac{1}{2}D_{ij+1/2}, \tag{7}$$

So Eq. (6) is rewritten as:

$$\frac{\partial \mathbf{w}_i}{\partial t} + \sum_{j=1}^{m}\left(a_{ij}F^I_{ij+1/2} + b_{ij}G^I_{ij+1/2}\right) = \frac{M_\infty}{\mathrm{Re}_\infty}\sum_{j=1}^{m}\left(a_{ij}F^v_{ij+1/2} + b_{ij}G^v_{ij+1/2}\right) - D_{j+1/2}. \tag{8}$$

To discrete the transient term an explicit four-stage Runge–Kutta scheme is used [8]. For computational efficiency the dissipation term D is calculated only at the first and third stages of the Runge–Kutta scheme. To accelerate the convergence, the local time stepping and implicit residual averaging are used in the present work [8,9].

2.2. Turbulence modeling

The first turbulence model which is implemented in mesh-less form is the algebraic Prandtl turbulence model [13]. The turbulent viscosity μ_t is calculated in this model as:

$$\mu_t = \rho\ell^2\left|\frac{\partial u}{\partial y}\right|. \tag{9}$$

More information about this method can be found in Ref. [13]. The turbulence equation is discretized in the same way as done for mean flow equations by the TLS approach. As can be seen from the above equation there is only one derivative calculation in the algebraic turbulence model that can be constructed as:

$$\mu_t = \rho\ell^2\left|\frac{\partial u}{\partial y}\right| = \rho_i\ell^2\sum_{j=1}^{m}|b_{ij}(u_j - u_i)|. \tag{10}$$

In addition to algebraic turbulence model, the one-equation Spalart–Allmaras (SA) turbulence model is discretized by mesh-less method.

$$\frac{\partial \tilde{v}}{\partial t} + \underbrace{u\frac{\partial \tilde{v}}{\partial x} + v\frac{\partial \tilde{v}}{\partial y}}_{convection} = \underset{production}{c_{b1}\tilde{S}\tilde{v}} + \underbrace{\frac{1}{\sigma}[\nabla.((\tilde{v}+v))\nabla\tilde{v} + c_{b2}(\nabla\tilde{v})^2]}_{diffusion} - \underbrace{c_{w1}f_w\left(\frac{\tilde{v}}{d}\right)^2}_{destruction}. \tag{11}$$

More details and the constants which are used in this equation can be found in Refs. [14,15]. In the SA turbulence model the discretization of convection and diffusion terms can be done in the following form.

2.2.1. Convection term

$$u\frac{\partial \tilde{v}}{\partial x} + v\frac{\partial \tilde{v}}{\partial y} = u_i\sum_{j=1}^{m}a_{ij}(\tilde{v}_j - \tilde{v}_i) + v_i\sum_{j=1}^{m}b_{ij}(\tilde{v}_j - \tilde{v}_i). \tag{12}$$

2.2.2. Diffusion term

$$\frac{1}{\sigma}\left[\nabla\cdot((\tilde{v}+v))\nabla\tilde{v} + c_{b2}(\nabla\tilde{v})^2\right] = \frac{1}{\sigma}\left[\left[\sum_{j=1}^{m}a_{ij}((\tilde{v}+v)_j - (\tilde{v}+v)_i) + \sum_{j=1}^{m}b_{ij}((\tilde{v}+v)_j - (\tilde{v}+v)_i)\right]\nabla\tilde{v} + c_{b2}(\nabla\tilde{v})^2\right], \tag{13}$$

where

$$\nabla\tilde{v} = \sum_{j=1}^{m}a_{ij}(\tilde{v}_j - \tilde{v}_i) + \sum_{j=1}^{m}b_{ij}(\tilde{v}_j - \tilde{v}_i). \tag{14}$$

Other terms of this model are computed by simple algebraic formulations. The boundary conditions for turbulence models at the solid wall can be set to $\mu_t = 0$ in algebraic and $\tilde{v} = 0$ in SA turbulence model. On the far field boundary, the turbulence variables are extrapolated if the flow is outgoing or set to their free stream value for incoming flow. The free stream value of eddy viscosity is defined as $\mu_t = 0.1\,\mu$ in algebraic turbulence model and $\tilde{v} = 3\frac{\mu_\infty}{\rho_\infty}$ that suggested by Rumsey and Spallart [15]. The initial turbulence variables are set to free stream turbulence values.

3. Turbulent flow computation over flat plate

The investigation is carried out by solving standard turbulent flow problem over a flat plate at Mach number 0.2 and Reynolds number one million. Two dimensional zero pressure-gradient flat plate case is solved on a series of regular point clouds. Three point clouds are constructed called the coarse, medium and fine point clouds with the size of 74×50, 137×97 and 224×200, respectively. The spacing of the first layer of points to the solid wall is selected to yield the amount of y^+ levels well less than 1. For example on medium point cloud this distance is 1×10^{-6}. The point cloud spacing in x-direction near the leading edge is about 0.002 that grows when approaching to the end of the plate. There are 112