Research Paper
International Journal of Aerospace System Engineering
Vol.2, No.2, pp.40-43 (2015)

ISSN 2383-4986

Evaluation of the Aerodynamic Performance of A Hybrid Mesh Based Finite Volume Code

H.W. ZHENG [1,†]
[1] *LHD, Institute of Mechanics, Chinese Academy of Science*
No.15 Beisihuanxi Road, Beijing 100190, China

Abstract : A hybrid mesh based finite volume compressible flow code (PolySim) has been developed recently. Instead of the simple average method for the gradients of variables at each face, the volume average is applied for the calculation of the viscous flux. What is more, an improved Green Gauss method for the calculation of the gradient is also presented. These two techniques will improve both the accuracy and robustness of the code. The aerodynamic performance of this in-house cell centered code is examined by several widely-used bench-mark test cases. These cases include flows over flat plate and RAE 2822 etc. The comparisons on results between calculation and experiment are conducted. They show that the code can produce good numerical results which agree well with the corresponding experiment data.

Key Words : Hybrid mesh, Finite volume, Turbulent model, Lift Coefficient, Drag Coefficient, Friction Coefficient

1. Introduction

Computational fluid dynamics has become an important tool in the engineering design. For example, the prediction of lift and drag play an important role in the engineering design. There are a lot of CFD codes developed so far. However, the performances are quite different due to their different implementations. For example, as reported in Schwoppe and Diskin (2013), the difference of the drag prediction by the cell-centered version and the cell-vertex version may be bigger than 30 drag counts. Hence, it is important to evaluate the performance of a CFD code.

The present in-house code (called Polysim) is a cell-centred finite volume solver based on hybrid mesh. It is developed based on the unstructured grid data architecture for flexibilities of handling complex geometries. Unlike finite difference, the finite volume formulation discretises the integral governing equations on arbitrary shaped control volumes. In this code, the control volume can be any one type of tetrahedron, hexahedron, pyramid and prism. Instead of the simple average method for the gradients of variables at each face, the volume average is applied. What is more, an improved Green Gauss method for the calculation of the gradient at the cell centroid is also developed. These two techniques for the discretization of the viscous flux terms will improve both the accuracy and robustness of the code. The aerodynamic performance of this in-house cell centered code is examined by several widely-used bench-mark test cases.

2. Methodology

Received: June 03, 2015 Revised: Aug. 24, 2015
Accepted: Sep. 12, 2015
† H.W. ZHENG
Tel:+ 86-10-8254-3976,
E-mail: h. zheng@imech.ac.cn

2.1. Governing equations

The compressible flows are solved by the Farve-averaged Navier-Stokes equations

$$\partial_t \int_{\Omega} U dV + \int_{S} \mathrm{F} \cdot \vec{\mathrm{n}} dS = 0$$

where U is the state vector, and F is flux vector,

$$U = \begin{pmatrix} \rho \\ \rho \vec{u} \\ E \end{pmatrix}, F = F^c - F^\upsilon .$$

Here, ρ is the density, $\vec{u}$ is the velocity, $\tilde{\upsilon}$ is the modified viscosity, E is the total energy, P is the pressure, and $[\tau]$ is the stress tensor,

$$[\tau] = (\mu_L + \mu_{tur}) \left[\nabla \vec{u} + \nabla^T \vec{u} - \frac{2}{3} (\nabla \cdot \vec{u}) [\mathrm{I}] \right].$$

2.2. Numerical discretization

The present in-house code (called Polysim) is a cell-centred finite volume solver based on hybrid mesh. The discretization of the inviscid flux terms is quite trival. Hence, in this section, the attention is paid to the discretization of the viscous flux terms. To calculate the viscous flux, one also needs to calculate the flow variables and their gradients on faces. Since we are using the cell-centred method and all flow variables are stored at the cell centers, the estimation of the variables on faces has to be constructed as accurate as possible. The integrals on the faces require the best estimation of all the flow variables on these faces. Usually, a simple average technique is applied.

$$U_{face} = \frac{1}{2} (U_L + U_R), \tag{4}$$

$$(\vec{\nabla} U)_{face} = \frac{1}{2} (\vec{\nabla} U_L + \vec{\nabla} U_R) \ . \tag{5}$$

In the present code, a volume weighted average of the adjacent cell gradients is employed

$$(\vec{\nabla} U)_{face} = \frac{\Omega_L \vec{\nabla} U_L + \Omega_R \vec{\nabla} U_R}{\Omega_L + \Omega_R} .$$

What is more, the gradient in the cell center is calculated by Green-Gauss method

$$(\vec{\nabla} U)_c = \frac{1}{\Omega_c} \sum_f (U_f \vec{n}_f) S_f,, \tag{2}$$

The face state in this calculation employs a new inverse distance weighted average rather than simple average (Eq. (4)),

$$U_f = \frac{\Omega_{Rf} U_L + \Omega_{Lf} U_R}{\Omega_{Lf} + \Omega_{Rf}} . \tag{3}$$

These new technique helps to improve the accuracy of the method as well as the improvement of the robustness of the code.

3. Numerical Cases

In order to evaluate the aerodynamic performance of this in-house cell centered code, several widely-used bench-mark test cases are examined.

3.1. Flat plate

The first case is the turbulent boundary layer flows over a flat plate at Ma number as 4.5. It has been extensively investigated experimentally by Coles (1953). The mesh used in this study is shown in Fig. 2. It consists of 12,808 cells with 7200 brick (hex) cells and 5608 prim cells. The grid was clustered in the stream wise direction at x=0 (the beginning of the flat plate) to resolve flow gradients near the leading edge of the plate and normal to the plate to resolve the boundary layer. The grid points upstream of the leading edge of the flat plate were treated as an in-viscid wall to provide a uniform profile at the leading edge location while the plate itself was modeled using a viscous wall. The symmetric boundary condition is applied to the sidewalls.

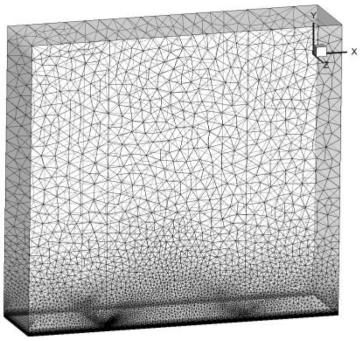
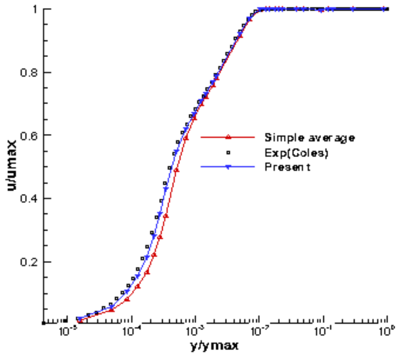
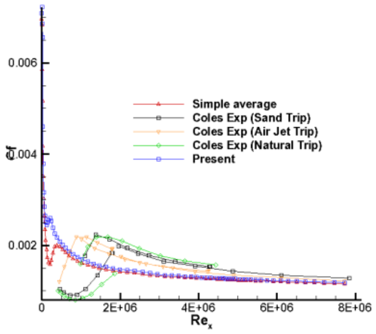


Fig. 1 Mesh

The velocity and skin friction profile are plotted in Fig. 2. From the Fig. 2(a), it could be easily observed that the results produced by the present method (Eqs. 6-8) agree well with the experiment ones by Coles (1953). Besides, it is clearly that the present results are better than those of the simple average method (Eqs. 4-5). Since the current code did not employ any transition model, it can be only used to predict the fully turbulent solution. Hence, results by both methods do not match the experimental skin friction data. However, the trends of both methods are closed to the experiment ones.



(a)



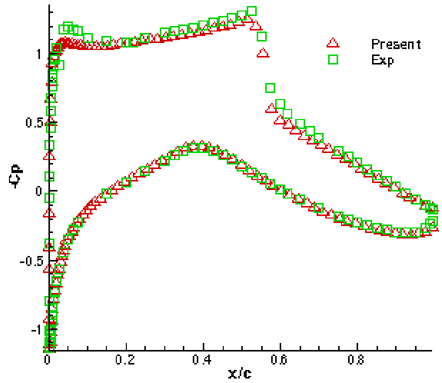
(b)

Fig. 2 Profiles of stream wise (u-) velocity and skin friction

3.2. Transonic flow over RAE2822 airfoil

The transonic flow over a RAE 2822 has been used as a benchmark test by many codes. There are several experimental results (Cook et al., 1979) to be compared. The RAE 2822 airfoil is a supercritical airfoil. The main feature of this flow is that a shock wave is developed on the suction side of the aerofoil. The flow conditions for the numerical simulation are $M_\infty = 0.74$, $\alpha = 2.79^\circ$ and $Re_\infty = 6.5 \cdot 10^6$, based on the chord length. Under this condition, the flow separation is not expected. The simulation is carried without considering of any transition.

Fig. 3 shows the distribution of the pressure coefficient and of the friction coefficient on the airfoil surface, respectively. Both results agree well with the experimental data (Cook et al., 1979).



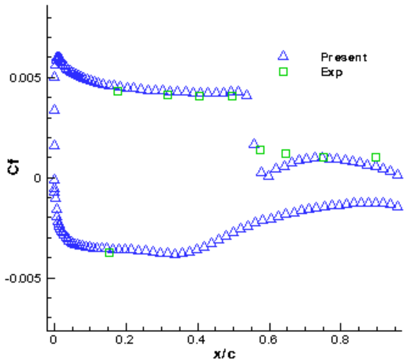


Fig. 3 Profiles of the pressure coefficient and skin friction

The aerodynamic lift coefficient and the drag coefficient are listed in Table 1. It is clearly that they are predicted very accurately, compared with the experimental data. The lift is slightly under predicted, while the drag is slightly over underestimated.

Table 1 The aerodynamic coefficients

	CL	CD
Present	0.79331	0.01754
Experiment	0.803	0.168

4. Conclusions

The evaluation of the aerodynamic performance of a hybrid mesh based finite volume compressible flow code is conducted. The two cases, flows over flat plate and the transonic flow over RAE 2822, are examined. The numerical results agree well with the experimental data. Besides, it shows that the improved Green Gauss method for the gradient calculations and the volume average gradient for the viscous flux helps to improve the accuracy of the results.

Acknowledgement

This research work was supported by the Natural Science Foundation for the Youth of China under Grant 11202220.

References

[1] Schwoppe, A. and Diskin, B., 2013, “Accuracy of the Cell-Centered Grid Metric in the DLR TAU-Code”, New Results in Numerical and Experimental Fluid Mechanics VIII Notes on Numerical Fluid Mechanics and Multidisciplinary Design Volume 121, 429-437, 2013.

[2] Coles, D., “Measurements in the Boundary Layer on a Smooth Flat Plate in Supersonic Flow: I. The Problem of the Turbulent Boundary Layer", Jet Propulsion Laboratory Report No. 20-69, 1953.

[3] Cook, P.H., M.A. McDonald, M.C.P. Firmin, "Aerofoil RAE 2822 - Pressure Distributions, and Boundary Layer and Wake Measurements", Experimental Data Base for Computer Program Assessment, AGARD Report AR 138, 1979.

Authors

H.W. ZHENG

Dr. Zheng obtained his BSc. and MSc. from FUDAN University (Shanghai, China) in 1999 and 2002 respectively. He got his PhD from National University of Singapore in 2008. His major research fields include hybrid turbulence modeling, multi-fluid flows and adaptive mesh generation.