WEST-EAST HIGH SPEED FLOW FIELD CONFERENCE
19-22, November 2007
Moscow, Russia

GRID CONVERGENCE STUDY FOR HIGH-LIFT DEVICES: THREE-ELEMENT AIRFOIL

ZHOU Lei*

** Aeronautic Computing Technique Research Institute (ACTRI)*
P.O.Box 90, Xi'an, SHAANXI, 710068, P.R. China
Email: zhoulei88496335@yahoo.com.cn

Key words: Grid Convergence Study, Three-Element Airfoil

Abstract. In this paper, grid convergence study for an unstructured mesh CFD code WoF90 in simulating the flow around a three element high-lift airfoil (slat, main wing, and flap) is carried out. and the results of both SA turbulence model and SST model are compared and discussed. To estimate convergence order of lift coefficient, generalized Richardson extrapolation method is used. Moreover, a new method is proposed to estimate convergence order of discrete L2 norm of pressure coefficient in entire flow field, it is shown that convergence order of discrete L2 norm of pressure coefficient in entire flow field is comparable to 2.0, theoretical estimated order for linearizing incompressible flow.

1. INTRODUCTION

The use of computational fluid dynamics (CFD) for the prediction of high-lift flow fields on commercial transport wings is becoming more routing way. However, the simulated flow physics are more and more complex. For example, the separate elements of the high-lift configuration shed wakes, whose effects must be accurately modeled by the flow solver.

These wakes often merge with other wakes, or the boundary layers of other elements. To complicate matters further, unlike the cruise condition, high-lift calculations are almost always performed over a wide range of angles of attack. This complicates the grid generation process, because pertinent flow features change as the angle of attack changes. Regions of the grid that may not need to be highly resolved at one angle of attack may be critical at other angles of attack. To develop an aerodynamic design system, the accuracy of CFD for such a complicated flow field has to be clarified.

In Murayama and Yamamoto1 computations were performed using two CFD codes based on different mesh systems (multi-block structured and unstructured mesh) in solving flow field around a three-element (slat, main, and flap) trapezoidal wing with fuselage. The aerodynamic forces were predicted reasonably, even with the unstructured mesh when moderate settings for slat and flap were used. However, the computational scale is so large that the studies of mesh dependency in the wake regions and the difference of the turbulence model are not included.

A simplified two-dimensional (2D) analysis is helpful to study these effects. Many aspects of flow characteristics can be understood in the 2D analysis.

In this paper, 2D grid convergence study for an unstructured mesh CFD code WoF90 in simulating the flow around a three element high-lift airfoil (slat, main wing, and flap) is carried out. To avoid convergence difficulty at higher angles of attack, only lower angle of attack (4° and 8°)are checked, and the results of both SA turbulence model and SST model are compared and discussed.

2. FLOW SOLVER AND TURBULENCE MODEL

An unstructured mesh generator and flow solver, WoF90, is used in this study, and RANS equations are solved in the unstructured mesh using a cell-vertex finite volume method. The unstructured code employs an edge-based approach. The control volumes are non-overlapping and are formed by a dual grid, which is computed from the control surfaces for each edge of the primary input mesh.

In this study, two turbulence models, Spalart-Allmaras one-equation model (SA) and Menter shear stress transport two-equation model (SST) are used to simulate turbulent flows. Meanwhile, In all computation, full turbulence model is used without transition consideration.

3. COMPUTATIONAL CONDITIONS AND MESHES

The configuration used in this study comprises a slat, main airfoil, and single-slotted flap, as in AGARD Report No. 303[2]. The slat and flap angles are set to 25° and 20° , respectively. This configuration was used as a code validation challenge by the CFD Society of Canada[3]. The free stream Mach number is 0.197 and the Reynolds number, Re, is 3.52×10^{6} . In the experiments, transition is tripped on the main element at x/c= 0.125 on both the upper and lower surfaces. On the slat and flap, transition is free. In the computations, The flow was assumed to be fully turbulent.

Although the trailing edges of this model are bluff and very thin, in order to capturing the wakes more accurately, all trailing edges are not sharpened. By an unstructured mesh strategy we can model trailing edge with increasing the number grid points near the blunt trailing edge.

The initial baseline unstructured mesh has 43,676 mesh points. The outer boundary is located 100-chord lengths away from the airfoil. This coarse mesh includes 20 layer quadrangle elements around all solid walls and triangle elements fill up other region.

The computational meshes used in this study are shown in Figure 1 and 2. The minimum spacing in the normal direction to the wing surface is $3\times10^{-6}(\approx 0.01/\sqrt{\mathrm{Re}})$ on unstructured meshes, which satisfies $y^{+}\approx 1$ on most of the surface.

Based on above initial mesh, two progressively larger meshes are generated by fowllowing mesh Refine strategy.

1) All triangle element are divided into four small ones by connect all midpoints of three edges.

2) All quadrangle are divided into two small ones by connect two midpoints on tangent edge of solid wall.
3) The number of boundary layers maintains 20.

The baseline mesh and three refined meshes summarized in Table 1 are used to examine the dependency of numerical accuracy on the density of unstructured meshes. The details are described in the following section.

By properly distributing mesh points in an initial coarse unstructured mesh, the detail in flow physics is obtained. Then a global unified refinement approach is also offered to generate a sequence of three progressively larger meshes, the dependency of numerical accuracy on the density of unstructured meshes is examined. .

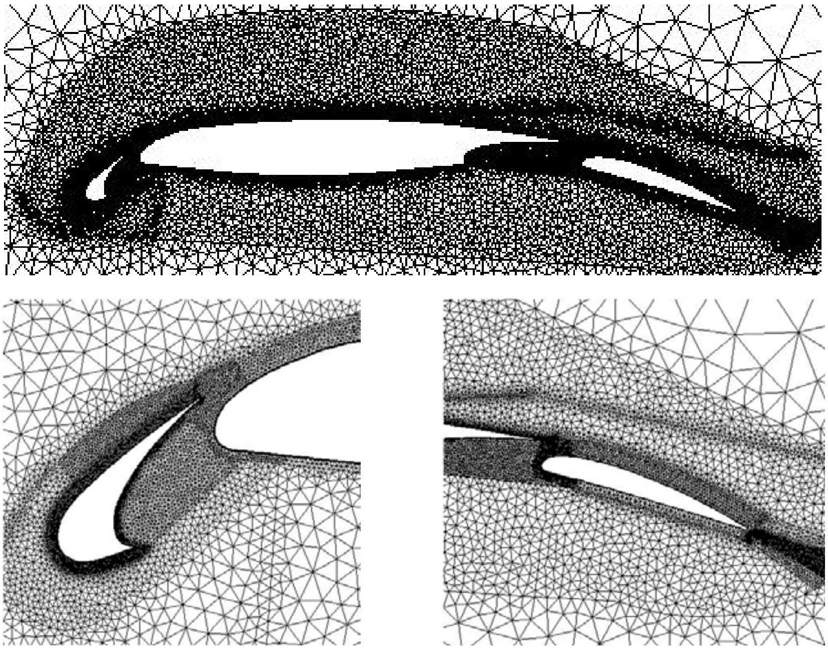

Figure 1. Baseline unstructured mesh

It is well known that verification and validation (V&V) are the primary means to assess accuracy and reliability in computational simulations. Among many errors in CFD application, the discretization error is of most concern to a CFD code user during an application. The examination of the grid convergence of a simulation is a straight-forward method to demonstrate convergence of the spatial discretization error with grid refinement, the method involves performing the simulation on two or more successively finer grids.

To avoid convergence difficulty at higher angles of attack, only lower and medium angle of attack is checked, and the results of both the SA turbulence model and SST model are compared and discussed. By properly distributing mesh points in an initial coarse unstructured mesh, the detail in flow physics is obtained. Then a global unified refinement approach is also offered to generate a sequence of three progressively larger meshes, the dependency of numerical accuracy on the density of unstructured meshes is examined. To estimate convergence order of lift coefficient, generalized Richardson extrapolation method is used. For both $4°$ incidence and $8°$ incidence cases the detailed results are shown in 0.

4. GRID CONVERGENCE STUDY FOR FORCE COEFFICIENTS

In Figure 2 it is shown that the changes in the computed lift coefficient decreases monotonically with increasing grid resolution. Moreover, generalized Richardson extrapolation method[4] can provide higher-order estimate of the lift coefficient value at zero grid spacing from a series of lower-order discrete values when lift coefficient value computed show monotone trend as grid the size of the grid spacing tends to zero.

	Mesh point	CL(SA) 4°	CL(SST) 4°	CL(SA) 8°	CL(SST) 8°
Mesh 1	43676	2.2145	2.2529	2.7901	2.8237
Mesh 2	120973	2.1854	2.2140	2.7491	2.7737
Mesh 3	375638	2.1613	2.1838	2.7167	2.7324
Extrapolation Value	∞	2.0953	2.1184	2.6422	2.6205
Extrapolation Order		0.5495	0.6704	0.6372	0.5545
Experment Value		2.1216	2.1216	2.6823	2.6823

Table 1.Lift Coefficients from CFD Calculations

	Mesh point	CL(SA) 4°	CL(SST) 4°	CL(SA) 8°	CL(SST) 8°
Mesh 1	43676	0.0335	0.0268	0.0370	0.0320
Mesh 2	120973	0.0329	0.0266	0.0365	0.0318
Mesh 3	375638	0.0317	0.0264	0.0346	0.0311
Extrapolation Value	∞	-	0.0260	-	-
Extrapolation Order		-	0.20	-	-
Experiment Value		0.0311	0.0311	0.0324	0.0324

Table 2.Drag Coefficients from CFD Calculations

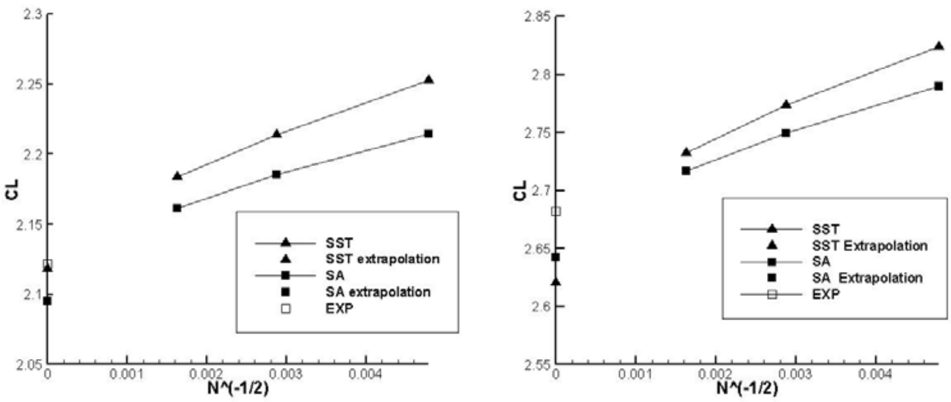

Figure 2. Comparison of computed lift coefficients versus the number of grid points to the -1/2 power at 4° incidence(Left) and 8° incidence(Right)

In short, above results show apparent grid convergence trend of computed lift coefficient by WoF90 code for SA model and SST Model at low and medium incidence.

5. GRID CONVERGENCE STUDY FOR CP DISTRIBUTION

In following discuss, the convergence trend of CP (pressure coefficient) distribution on both viscous wall and entire flow field are investigated. To measuring the CP distribution difference, two norm is defined as follow:

Let $\| Error_{mc}\_CP \|_{L^2(\Gamma)}$ is discrete L2 norm of error between CP solved by coarse grid and middle grid on viscous wall Γ, then:

$$\| Error_{mc}\_CP \|_{L^2(\Gamma)} = \frac{\sqrt{\sum_{p_c \in \Gamma} (CP_c(p_c) - CP_m(P_c))^2}}{\sum_{p_c \in \Gamma} 1} \qquad \forall p_c \in \Gamma \tag{1}$$

Here: P_c is coarse grid point (also is middle grid point) on Γ, $CP_c(P_c)$ means the CP value at point P_c solved on coarse grid, and $CP_m(P_c)$ means the CP value at point P_c solved on middle grid. In the familiar way we denote $\| Error_{fm}\_CP \|_{L^2(\Gamma)}$, the discrete L2 norm of error between CP solved by middle grid and fine grid in viscous wall Γ.

On the other hand, Let $\| Error_{mc}\_CP \|_{L^2(\Omega)}$ is discrete L2 norm of error between CP solved by coarse grid and middle grid on entire flow field Ω, then:

$$\| Error_{mc}\_CP \|_{L^2(\Omega)} = \frac{\sqrt{\sum_{p_c \in \Omega} (CP_c(p_c) - CP_m(P_c))^2}}{\sum_{p_c \in \Omega} 1} \qquad \forall p_c \in \Omega \tag{2}$$

Here: P_c is coarse grid point (also is middle grid point) on Ω, $CP_c(P_c)$ means the CP value at point P_c solved on coarse grid, and $CP_m(P_c)$ means the CP value at point P_c solved on middle grid. In the familiar way we denote $\| Error_{fm}\_CP \|_{L^2(\Omega)}$, the discrete L2 norm of error between CP solved by middle grid and fine grid in entire flow field Ω.

In Figure 3 ， let $\| Error\_CP \|_{L^2(\Gamma)}$ standing for discrete L2 norm of error between CP solved by coarse grid and middle grid (or by middle grid and fine grid) on viscous wall Γ, and let $\| Error\_CP \|_{L^2(\Omega)}$ standing for discrete L2 norm of error between CP solved by coarse grid and middle grid (or by middle grid and fine grid) in entire flow field Ω, it is shown that the both two kind of error norm in the computed pressure coefficient decreases monotonically with increasing grid resolution with almost same speed. Moreover, if we calculate the $\log(\| Error\_CP \|_{L^2(\Gamma)})$ (or $\log(\| Error\_CP \|_{L^2(\Omega)})$) and $\log(N^{-1/2})$, then we can easily obtain new convergence order for $\| Error\_CP \|_{L^2(\Gamma)}$ and $\| Error\_CP \|_{L^2(\Omega)}$, which is shown in table 3.

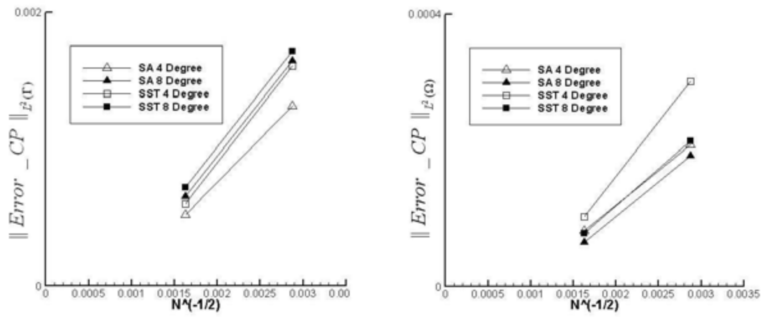

Figure 3. $\| Error\_CP \|_{L^2(\Gamma)}$ versus the number of grid points to the -1/2 power(left), and $\| Error\_CP \|_{L^2(\Omega)}$ versus the number of grid points to the -1/2 power(right)

Error Norm	Turb. Model	Attack angle	Convergence Order
$\| Error\_CP \|_{L^2(\Gamma)}$	SA	4°	1.6323
		8°	1.6305
	SST	4°	1.73985
		8°	1.5274
$\| Error\_CP \|_{L^2(\Omega)}$	SA	4°	1.6556
		8°	1.9210
	SST	4°	1.9061
		8°	1.7843

Table 3.Convergence order for $\| Error\_CP \|_{L^2(\Gamma)}$ and $\| Error\_CP \|_{L^2(\Omega)}$

Note that almost all convergence order shown in table 3 are not far from 2.0, theoretical estimated order for linearizing incompressible flow, thus it means the computed results obtained by these three grids already fall in to asymptotic convergence range

6. CONCLUSIONS AND FUTURE WORKS

In this paper, we get convergence order of lift coefficient by generalized Richardson extrapolation method, and get convergence order of discrete L2 norm of pressure coefficient in entire flow field and on solid viscous wall, all these research works, especially later ones, shown that computed results are convergent at a reasonable speed as grid the size of the grid spacing tends to zero.

Moreover, further grid convergence study should focus on three dimension case of CFD simulation for flow field around high lift configuration, and new grid refinement strategy should designed to reduce amount of grid points of fine grid.

REFERENCES

[1] M. Murayama, K. Yamamoto, and K. Kobayashi, "Validation of Computations Around High-Lift Configurations by Structured- and Unstructured-Mesh", *Journal of Aircraft,* Vol.43, No.2, March-April, 2006.

[2] I. R. M. Moir, "Measurements on a Two-Dimensional Aerofoil with High-Lift Devices", *AGARD Advisory Report* No. 303: "A Selection of Experimental Test Cases for the Validation of CFD Codes", Vol. 1 and 2, 1994, pp. A2-1 – A2-12.

[3] I. Fejtec, "Summary of Code Validation Results for a Multiple Element Airfoil Test Case", *AIAA Paper* 97-1932, 1997.

[4] T. J. Baker, "Mesh generation: Art or science?", *Progress in Aerospace Sciences* 41 (2005) 29 – 63.