Improvement of Aircraft Design Technology by Developing Aerodynamics Simulation Code for Whole Aircraft Configuration using High-Order Unstructured Mesh Method

Project Leaders

Kazuhiro Nakahashi Department of Aerospace Engineering, Tohoku University
Keisuke Sawada Department of Aerospace Engineering, Tohoku University
Kazuomi Yamamoto Aviation Program Group, Japan Aerospace Exploration Agency
Yuichi Shimbo Nagoya Aerospace Systems, Mitsubishi Heavy Industries, Ltd.

Authors

Kunihiko Watanabe The Earth Simulator Center, Japan Agency for Marine-Earth Science and Technology
Takanori Haga Graduate School of Engineering, Tohoku University
Akihisa Masunaga Nagoya Aerospace Systems, Mitsubishi Heavy Industries, Ltd.

The aerodynamics simulation code for complete aircraft configuration using high order SV method has been developed for the purpose of improving aircraft design technology. The developed code is first validated for transonic flow computation over ONERA-M6 wing, and is then applied to a simulation around the JAXA high-lift configuration model. The obtained results of the Euler computation are compared with the available wind-tunnel data. Fairly good agreements are obtained in these comparisons although viscous effects are all neglected in the calculation. In particular, the computed results of the JAXA high-lift configuration model indicate that trailing vortices from various high lift devices are clearly captured even in the downstream region, where vortices are likely to be vanished due to inherent numerical viscosity in the conventional unstructured mesh methods. This clearly demonstrates that the present high order unstructured mesh method is capable of capturing various flow features quite accurately while it retains the desired geometrical flexibility. It is also shown that the developed code achieves high computing performance on the Earth Simulator and exhibits a potential for future large scale computations.

Keywords: aircraft, whole aircraft simulation, design technology, industry-government-academia collaboration

1. Introduction

During landing and take-off phase in aircraft operation, complicated flow features due mainly to turbulence appear that can cause aircraft safety issues, and can also cause environmental issues in the area nearby airport. To improve the performance of aircraft, and also to alleviate the environmental load, aircraft design methodology should be sophisticated using various newly developed prediction methods.

Computational fluid dynamics (CFD) was emerged in 1980s as a leading area in computational physics, and has already been recognized as an indispensable engineering tool in aircraft industry. The modern CFD methods can easily consider various complicated geometries such as complete aircraft configuration even with high-lift-devices (HLD) fully deployed. This has been made practical owning to the rapid progresses in mesh generation techniques, particularly those for unstructured mesh methods. Indeed, unstructured mesh methods have offered flexible grid generation for 3D complicated geometries, and extend the applicability of CFD in real aircraft design cycles. However, one numerical issue has been recognized as critical for those unstructured mesh methods that the spatial accuracy of the commonly used finite volume method remains at most second order due to several reasons. In order to resolve vortices and boundary layer separation near HLD the use of low dissipative high-order schemes is really required.

Recently, new high-order unstructured mesh methods such as discontinuous Galerkin (DG) method[1] and spectral volume (SV) method[2-4] have attracted attentions, because these methods are shown to achieve higher order spatial accuracy rigorously on unstructured mesh. In this report, we focus on upgrading the existing unstructured mesh method by employing the SV method to explore possible impact on the aerodynamic design methodology for modern aircraft. At first, the developed code is validated for transonic flowfield over ONERA-M6 isolated wing, and is then applied to obtain inviscid flowfield around JAXA high-lift configuration model. The obtained results are compared with the available wind-tunnel data. Parallel performance of the developed code on the Earth Simulator is also indicated.

2. Parallelized SV unstructured mesh method

In the SV method, the computational domain is divided into non-overlapping tetrahedral cells called spectral volumes (SVs), and each SV is further sub-divided into a structured set of sub-cells called control volumes (CVs). These CVs serve as the stencil to perform high-order data reconstruction inside each SV. The dependent variables assigned in each CV are updated independently as solution unknowns. The number of CVs in one particular SV is determined from the degree of polynomial function that approximates the dependent variable in SV. Therefore, as shown in Fig. 1, the number of CVs becomes four and ten for linear and quadratic reconstruction, respectively. Detailed descriptions of the SV discretization can be found in the literatures[3-5].

The developed SV code has been parallelized using Message Passing Interface (MPI) library. Assume that the computational domain is decomposed into sub-domains. Each sub-domain is then assigned to a processor element. Data communication through inter-domain boundary occurs by MPI calls. Because data reconstruction is solely accomplished within each SV, coupling of neighboring cells occurs only through the numerical flux function determined at the common SV interface. This compact feature of the SV discretization facilitates parallelization of the solution algorithm. Note that data transfer between processors is only needed for those SV elements that face at the inter-domain boundary.

The total computational cost of the SV method is sharply increased as the number of CVs is increased. Required computational resource of the SV method is obviously greater than that for conventional methods because we need to update the dependent variables assigned in each CV. Therefore, it may be fair to compare the computational cost of the SV method with that of the conventional method for the case of having computational cells as many as the total number of CVs instead of the total number of SVs.

3. Transonic flow over ONERA-M6 wing

The developed code is validated for the transonic flowfield over ONERA-M6 wing by solving the Euler equations. This is a standard CFD validation case for external flows. The freestream Mach number is $M = 0.84$ and the angle of attack is 3.06 deg. Figure 2 shows the pressure contours plotted on the root plane as well as on the wing surface. This computation is made by the second order SV method using

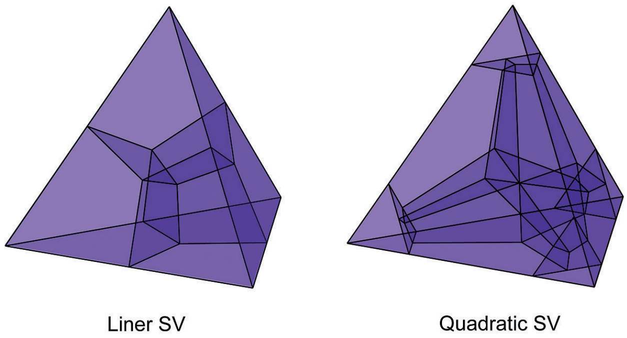

Fig. 1 Partitions in a SV. (Linear and quadratic partitions for second and third order data reconstructions.)

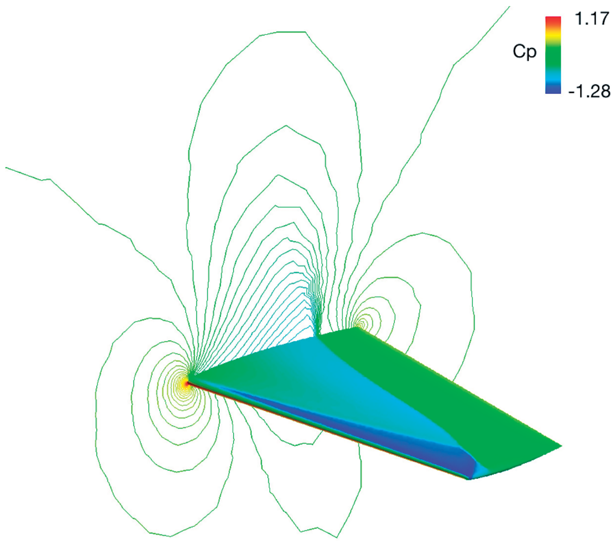

Fig. 2 Pressure contours on the root plane as well as on the surface of ONERA-M6 wing.

tetrahedral cells (SVs) as many as about 1 million. One can find that a lambda shaped shock wave pattern develops on the upper surface of the wing. Although not shown here, it is confirmed that the computed pressure profiles at several span wise cross sections agree well with the corresponding experimental data.

4. Inviscid flow past JAXA high-lift configuration model

The developed code is applied to obtain inviscid flowfield over a realistic aircraft configuration deploying HLD. The wind tunnel test for this model was conducted at the low-speed wind tunnel in JAXA [6]. The aims of the wind tunnel test were to provide detailed experimental data for CFD validation attempted in the CFD workshop [7], and also to understand the flow physics pertinent to HLD.

Figure 3 shows the unstructured mesh system for a wing-body configuration having nacelle-pylon, leading-edge slat, inner double-slotted fowler flap, and outer single-slotted fowler flap without flap-track-fairing. The surface mesh is generated using the direct surface triangulation method [8],

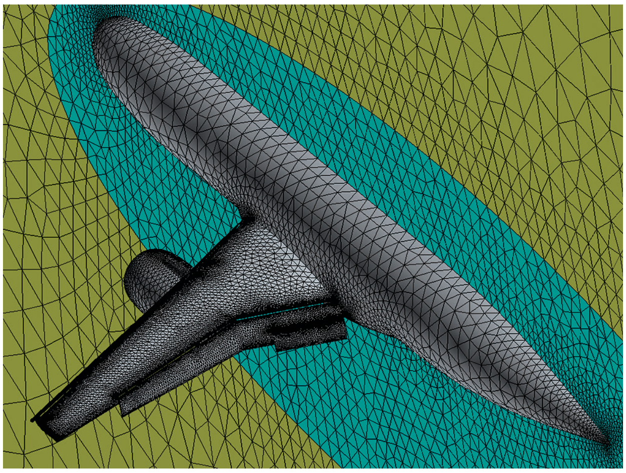

Fig. 3 Computational mesh around JAXA high-lift configuration model.

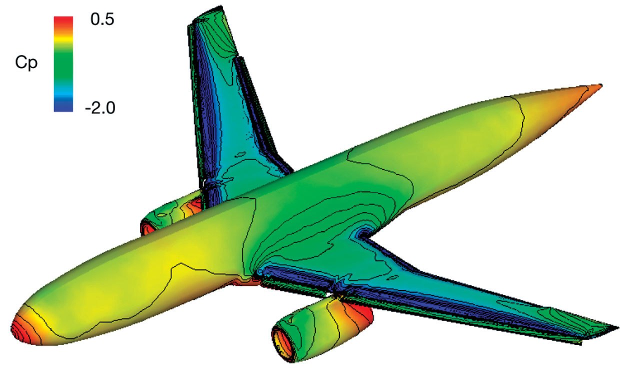

Fig. 4 Pressure contours on the surface of JAXA high-lift configuration model.

and then volume meshing is conducted using the Delaunay method [9]. The number of tetrahedral cells is 2.33 million (0.41 million grid points). We refer this as the coarse mesh. The freestream Mach number is M = 0.175(60 m/s) and the angle of attack is 4.0 degrees, both corresponding to the design condition.

Figure 4 shows the pressure contours on the model surface obtained by the third order SV method. In Fig. 5, the pressure profiles at the cross sections of 24.5% semi span are compared with the results given by Tohoku University Aerodynamics Simulation (TAS) code [10] which is a cell-vertex finite volume scheme. The formal accuracy of the TAS code is second order in space. For comparison, TAS computation is also conducted using a fine mesh which has 74.7 million hybrid cells (13 million grid points). In this case, the computational mesh and results were provided by JAXA. One can find that the Cp profile given by the present SV method using the coarse mesh agrees fairly well with the experimental data. Note that the computed Cp profile given by the TAS code using the coarse mesh shows insufficient expansion on the upper surface resulting in far smaller lift coefficient, whereas the computed Cp profile of the TAS code using the fine mesh gives a good agreement both with the experimental data and that given by the third order SV method using the coarse mesh. In Fig. 6, the vorticity contours obtained by the present SV method are plotted on several cross sections to illustrate the spatial evolution of wake vortices from HLD. It should be noted that the spatial evolution of vortices from HLD is well captured even on this relatively coarse mesh.

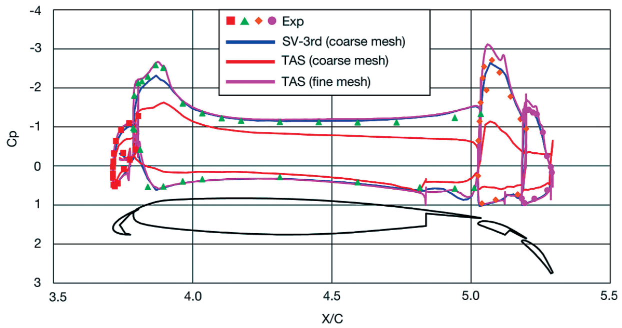

Fig. 5 Comparison of pressure coefficient at the cross sections of 24.5% semi span.

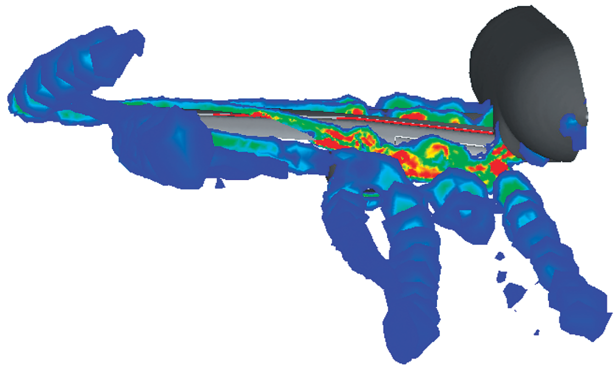

Fig. 6 Vorticity contours on several cross sections in the downstream region.

The SV computation of the flowfield for the JAXA high-lift configuration model needs about 50 hours to obtain the converged solution using 40 nodes of the Earth Simulator (ES). For this case, the sustained computing performance is 1.296 TFLOPS (theoretical peak is 2.560 TFLOPS) using 40 nodes, and 11.067 TFLOPS (theoretical peak is 32.768 TFLOPS) using 512 nodes. The longer computing time needed for this case is caused by the explicit time integration using the Runge-Kutta scheme.

The simulation of the flowfield around the realistic aircraft configuration with HLD fully deployed is carried out for the first time using the high-order accurate SV method. The obtained results obviously indicate the potential of high-order CFD method. Viscous effects will be included in our next study, where convergence acceleration to steady solution will be made by utilizing the implicit SV code based on

LU-SGS algorithm. In the future study, we will attempt to obtain unsteady flowfield with resolving small scale vortices in turbulence. Although this certainly requires the use of more powerful computer and efficient algorithm such as hp adaptation, we expect that such detailed computation will give the required aerodynamic coefficients with sufficient accuracy for aircraft design.

5. Summary

The aerodynamics simulation code for complete aircraft configuration using high order SV method has been developed. The developed code is first validated for transonic flow computation over ONERA-M6 wing, and is then applied to the simulation of the flowfield around the JAXA high-lift configuration model. The computed pressure profiles are shown to agree well with the corresponding experimental data. It is also shown that the trailing vortices from high lift devices installed on JAXA high-lift configuration model are clearly captured even in the downstream region. High computing performance of the present SV code on the Earth Simulator is shown to exhibit the potential of high-order accurate CFD method for future large scale computations.

References

[1] B. Cockburn, and C.-W. Shu, "The Runge-Kutta discontinuous Galerkin method for conservation laws V: multidimensional systems," *Journal of Computational Physics*, Vol.141, 1998, pp.199–224.

[2] Z. J. Wang, and Y. Liu, "Spectral (finite) volume method for conservation laws on unstructured grids II: extension to two-dimensional scalar equation," *Journal of Computational Physics*, Vol.179, 2002, pp.665–697.

[3] Z. J. Wang, L. Zhang, and Y. Liu, "Spectral (finite) volume method for conservation laws on unstructured grids IV: extension to two-dimensional systems," *Journal of Computational Physics*, Vol.194, No.2, 2004, pp.716–741.

[4] Y. Liu, M. Vinokur, and Z. J. Wang, "Spectral (finite) volume method for conservation laws on unstructured grids V: Extension to three-dimensional systems," *Journal of Computational Physics*, Vol.212, No.2, 2006, pp.454–472.

[5] T. Haga, N. Ohnishi, K. Sawada, and A. Masunaga, "Spectral volume computation of flowfield in aerospace application using Earth Simulator," AIAA Paper, 2006–2823.

[6] T. Ito, H. Ura, Y. Yokokawa, H. Kato, M. Mitsuo, and K. Yamamoto, "High-lift device testing in JAXA 6.5m × 5.5m low-speed wind tunnel," AIAA Paper, 2006–3643.

[7] T. Haga, N. Ohnishi, K. Sawada, A. Masunaga, and N. Uchiyama, "Flow simulation around JAXA high-lift configuration model using high-order unstructured method," 44th Aircraft Symposium, 2006. (in Japanese)

[8] Y. Ito and K. Nakahashi, "Direct surface triangulation using stereolithography data," *AIAA J.*, Vol.40, No.3, 2002, pp.490–496.

[9] D. Sharov and K. Nakahashi, "A boundary recovery algorithm for Delaunay tetrahedral meshing," 5th Int. Conf. on Numerical Grid Generation in Computational Fluid Simulations, 1996, pp.229–238.

[10] D. Sharov and K. Nakahashi, "Reordering of hybrid unstructured grids for lower-upper symmetric Gauss-Seidel computations," *AIAA J.*, Vol.36, No.3, 1998, pp.484–486.

「全機シミュレーションによる安全性・環境適応性の向上を目指した民間航空機設計技術の開発」に係わる共同調査

プロジェクトリーダー
中橋　和博　　東北大学　航空宇宙工学専攻
澤田　恵介　　東北大学　航空宇宙工学専攻
山本　一臣　　宇宙航空研究開発機構　航空プログラムグループ
真保　雄一　　三菱重工業株式会社　名古屋航空宇宙システム製作所

著者
渡邉　國彦　　海洋研究開発機構　地球シミュレータセンター
芳賀　臣紀　　東北大学　大学院工学研究科
増永　晶久　　三菱重工業株式会社　名古屋航空宇宙システム製作所

民間航空機の安全性及び環境適応性の向上に関し、特に離着陸時のフルフラップ形態における機体まわりの流れ場の正確な把握（飛行安全性）、流れの乱れが発生源となる騒音の推定（周辺環境負荷軽減）、機体から流れ去る気流の後方への影響把握（運航管制安全性）などが重要な技術課題として挙げられる。これら機体まわりの流れ場は、主に乱流現象に支配される複雑な流れとなっており、航空機の飛行安全性・運航管制安全性を確保し、かつ周辺環境への負荷を軽減するためには、予測手法を改善し、より高度な航空機設計技術を開発する必要がある。そこで、本調査では、これらの技術課題に対して、地球シミュレータによる航空機全機まわりの空力シミュレーションを実施し、その有効性を確認する。

本年度は、より高精度な計算手法としてSpectral volume（SV）法を導入した空力シミュレーションを実施した。SV法は、複雑形状への形状適応性の高い非構造格子法において、従来困難であった空間精度の高次化が可能な計算手法として近年研究が進められている最新の計算手法である。はじめに、簡易主翼形状に対する巡航速度域（遷音速流れ）のEuler解析を実施し、移植したコードの妥当性を確認した。次に、航空機全機まわりの空力シミュレーションとして、宇宙航空研究開発機構（JAXA）で実施された風洞試験（JAXA高揚力装置風洞模型）に対するEuler解析を実施した。翼面上の圧力分布を風洞試験データと比較した結果、妥当な解析結果が得られることを確認した。特に、従来手法では捉えることが難しい、スラット端・フラップ端などから発生した渦が機体後方に流れ去る様子をより鮮明にシミュレートしており、空力騒音や後流渦の源となる翼端渦の現象把握に極めて有効であることが判った。また、本コードは、地球シミュレータ上でスケーラビリティの高い演算性能を達成しており、航空機全機まわりの空力シミュレーションに地球シミュレータが非常に有効であることを確認することができた。

キーワード：航空機, 丸ごとシミュレーション, 設計技術, 産官学連携