DEVELOPMENT OF FAST UNSTRUCTURED CFD CODE "FASTAR"

Atsushi Hashimoto*, Keiichi Murakami*, Takashi Aoyama*, Keiichi Ishiko*, Manabu Hishida, Masahide Sakashita***, Paulus Lahur******
***Japan Aerospace Exploration Agency, ** Ryoyu Systems Co., Ltd.,**
***** Daiko Denshi Tsushin, Ltd, **** Research Center of Computational Mechanics, Inc.**
ahashi@chofu.jaxa.jp

Abstract

In order to develop a fast CFD code, we investigated well-known unstructured CFD codes, and we have determined the target performance and specification of CFD code. This is then followed by the development of a fast unstructured CFD code "FaSTAR". The accuracy of drag prediction with FaSTAR is validated by the DPW4 benchmark problem. The computed drag coefficients generally agree with other results. Since we employ the Cartesian-based unstructured grid generated with HexaGrid, the predicted drag is affected by the choice of discretization method, whether it is cell-center or cell-vertex. The computational speed of present CFD code is 1.8 hour/case with 10 million grid and 100 cores for a standard civil aircraft. Although we have not achieved the requirement of 1hour/case, we believe that it is possible to achieve the target using the multigrid. The preliminary multigrid result shows four times faster convergence with global coarse grid.

1 Introduction

During wind tunnel test campaign, approximately 200 aerodynamic data are obtained every day. However, the present CFD code cannot produce the data at the same speed. It takes several hours for one case by using a supercomputer. Therefore, we need a fast CFD code to improve the productivity. In this study, we develop a new fast CFD code. Since the various CFD codes are developed so far, we investigate their performance and major computational schemes. We determine target performance and specifications of CFD code. Then, we develop the code and carry out several computations to validate it. The accuracy and computational speed of the new code is demonstrated.

2 Target performance and code design

Target performance of new CFD code is as follows,

- ✓ Computational time: 1 hour / case
- ✓ Governing Equation: RANS
- ✓ Grid: Unstructured grid, 10 million cells (nodes)
- ✓ Computational resource: 100 cores of JAXA supercomputer system (JSS)

The performance is determined by requirement of accuracy and computational speed. We studied feasibility considering the present CFD technology level.

Before developing the code, we investigated numerical schemes employed by famous unstructured-grid CFD codes: NSU3D[1], FUN3D[2], TAU[3], USM3D[4], BCFD[5], Edge[6], TAS[7]. Based on this investigation, we determine the computational schemes.

We can select the discretization method from the cell-center and cell-vertex methods for the new code. The data structure is face-based, so that almost all do-loops in the code are face-loop such as computations of gradient, fluxes, and time integration. Basically, the only cell indexes connecting a face (from face number to cell number) are stored. The various Riemann solvers are installed so that they can cover from low-subsonic to hypersonic flows. As turbulence models, major two turbulence models, SA and SST models, are employed. In addition, the explicit algebraic Reynolds stress model (EARSM) is also employed as one of the anisotropic models. We use the multigrid method and GMRES to accelerate convergence. Finally, this code is named "FaSTAR (FAST Aerodynamic Routines)" aiming at the fastest flow solver.

3 Results

We validate the code for drag prediction benchmark problem of DPW4. The aircraft model is NASA-CRM and the three kinds of grid are generated: coarse, medium, and fine grids. Figure 1 shows the medium grid.

Computational conditions are Mach number of M=0.85, Reynolds number of $Re=5.0\times10^{6}$. Here, we compare the cell-center and cell-vertex methods. The drag coefficients at CL=0.5 are computed for the three grids. Figure 2 shows Cp contour computed with medium grid. The grid convergence of CD is shown in Fig. 3 with the three DPW4 results[8]: Gridgen+UPACS, TASmesh+JTAS, HexaGrid+JTAS. As a whole, the both FaSTAR results agree with the other DPW4 results. The results with cell-center are a little higher than the cell-vertex results due to the entropy production at the interface of the Cartesian grids with different size.

Then, DLR-F6 FX2B is computed to measure computational speed. The computational conditions are Mach number M=0.7, Reynolds number $Re=1.5\times10^{5}$, attack angle is $\alpha=0^{\circ}$. SA turbulence model is used. The grid is generated with HexaGrid in 25 minutes at JSS front-end (Fujitsu SPAC Enterprise M9000), and total number of cells is approximately 6.7 million. RANS simulation is carried out using 100 cores of JSS. Convergence acceleration techniques such as multigrid and GMRES are not used in this computation. The aerodynamic coefficients converge at 10,000 steps, and total computational time is 76 minutes. If we calculate the computational time for 10 million grid in a straightforward manner, it is about 1.8 hours. Although this is not as fast as the requirement of 1 hour/case, we believe that it is possible to achieve the target speed using the multigrid. In addition, FaSTAR is found to achieve 1GFlops for 1CPU (10% of theoretical peak) at JSS. Therefore, we could develop a fast flow solver for unstructured grid.

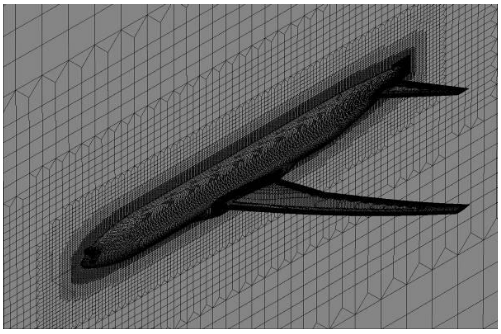

Fig. 1 Grid generated with HexaGrid

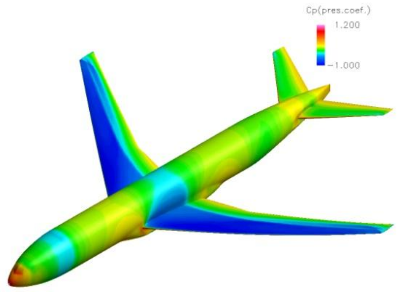

Fig. 2 Surface Cp of NASA-CRM

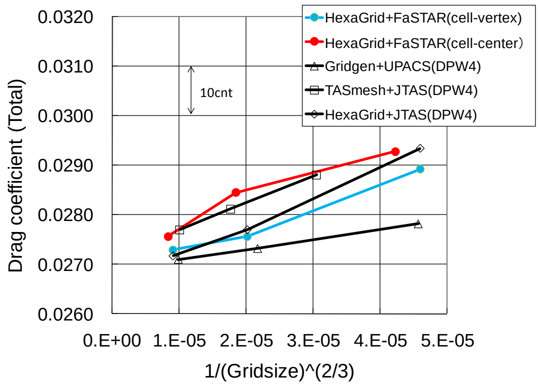

Fig. 3 Grid convergence of CD

Next, the multigrid method is demonstrated. Here, we employ two different coarse grids as shown in Fig. 4(b) and 4(c), where Fig. 4(a) is the original grid. One is zonal coarse grid and the other is global coarse grid. The coarse grids are generated using the octree data structure of Cartesian grid. The wing is ONERA-M6 wing. Mach number is 0.84 and the attack angle is 3.06°. Inviscid flow is assumed in this study. Figure 5 shows the convergence history of drag coefficient. In fact, the speed of zonal coarse grid is almost same as original time integration. (We could not accelerate the convergence with zonal coarse grid.) However, the global coarse

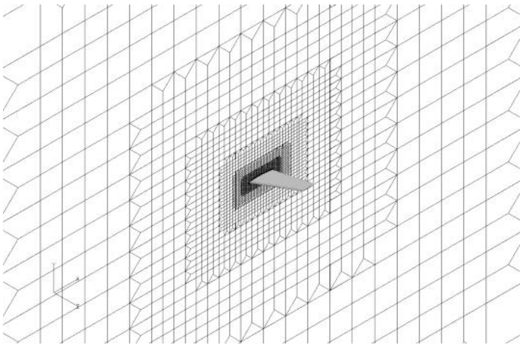

Fig. 4(a) Original Grid

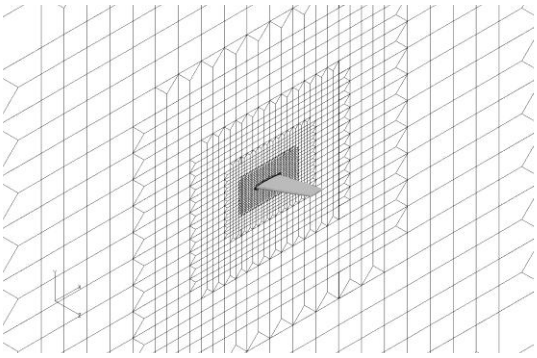

Fig. 4(b) Zonal coarse grid

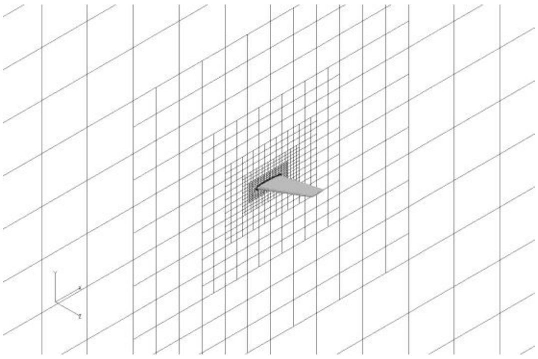

Fig. 4(c) Global coarse grid

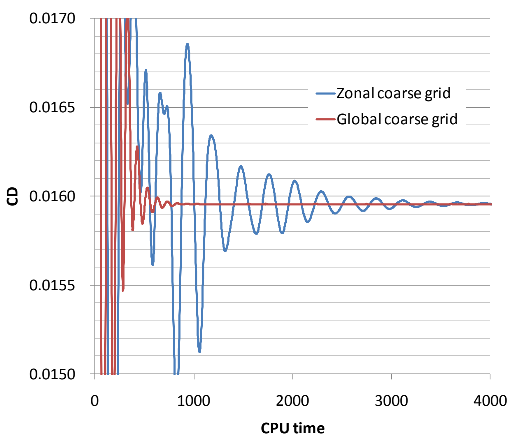

Fig. 5 CD convergence

grid is more effective than zonal coarse grid, and it converges approximately 4 times faster. We believe that it is possible to achieve the target performance using the multigrid.

4 Conclusions

Based on the investigation, we have developed a fast unstructured CFD code "FaSTAR". The accuracy of drag prediction with FaSTAR is validated by the DPW4 benchmark problem. The computational speed of present CFD code is 1.8 hour/case with 10 million grid and 100 cores. Although we have not achieved the requirement of 1hour/case, we believe that it is possible to achieve the target using the multigrid.

References

[1] Mavriplis, D., "Third Drag Prediction Workshop Results Using the NSU3D Unstructured Mesh Solver," Journal of Aircraft, vol. 45, no. 3, 2008, pp. 750-761.

[2] Anderson, W. K., et al., "An Implicit Upwind Algorithm for Computing Turbulent Flows on Unstructured Grids," Computers Fluids, Vol. 23, No. 1, 1994, pp. 1–21.

[3] Gerhold, T., "Overview of the Hybrid RANS Code TAU," MEGAFLOW, Springer–Verlag, 2005, pp. 81–92.

[4] Frink, N. T., "Recent Progress Toward a Three-Dimensional Unstructured Navier-Stokes Flow Solver," AIAA paper 94-0061, 1994.

[5] Mani, M., et al., "A Structured and Hybrid Unstructured Grid Euler and Navier-Stokes Solver for General Geometry," AIAA paper 2004-524, 2004.

[6] Edge Theoretical formulation, FOI dnr 03-2870, 2007.

[7] Nakahashi, K., et al., "Some challenges of realistic flow simulations by unstructured grid CFD", International Journal for Numerical Methods in Fluids, Vol.43, 2003, pp.769-783.

[8] Hashimoto, A., et al., "Drag Prediction on NASA CRM Using Automatic Hexahedra Grid Generation," AIAA paper 2010-1417, 2010.

Copyright Statement