Mitsubishi Heavy Industries Technical Review Vol. 62 No. 1 (March 2025)

Validation of Applicability of LES Analysis Technology for Accurate Prediction of Large-Scale Unsteady Flow around Aircraft

TAKAAKI YUMITORI[*1] **SHUN TAMURA**[*1]
KAZUHIRO IMAI[*2] **SOSHI KAWAI**[*3]

To understand aerodynamic characteristics in aircraft development, CFD analysis is used in addition to wind tunnel testing. However, when evaluating the maximum lift coefficient, which is important for aircraft aerodynamic performance, unsteady separated flows need to be captured and analyzed, which is still difficult to predict using conventional CFD analysis technology. Therefore, Mitsubishi Heavy Industries, Ltd., in collaboration with Tohoku University, a CoE in this field, is validating the applicability of a CFD solver that implements the latest large-scale LES analysis technology to actual design issues with the aim of significantly improving the capability of unsteady separated flow analysis. In this report, the prediction accuracy improvement of the maximum lift coefficient was confirmed by the comparisons with the flight test results for SpaceJet.

1. Introduction

It has been a long time since computational fluid dynamics (hereinafter referred to as CFD) analysis has been used together with wind tunnel testing to understand aerodynamic characteristics in aircraft development. Recently, both hardware and software have evolved to such an extent that even active discussions on certification by analysis (CbA), which replaces wind tunnel tests and flight tests required for aerodynamics-related certification with CFD analysis, have begun.

On the other hand, the maximum lift coefficient, which is important for the aerodynamic performance of aircraft, is still difficult to predict by CFD analysis. The maximum lift coefficient is an extremely important performance evaluation index that determines the minimum speed required for an aircraft to fly, but under flight conditions where the maximum lift coefficient is obtained, an unsteady separation flow, which is one of the phenomena difficult to capture by conventional CFD analysis, occurs around the wings. To solve this issue, Mitsubishi Heavy Industries, Ltd. (hereinafter referred to as MHI), in collaboration with Tohoku University, a domestic CoE in CFD analysis technology, has been conducting the applicability validation of a CFD solver FFVHC-ACE(1), which realizes highly accurate prediction of the maximum lift coefficient, to product development. This report introduces issues in the conventional CFD and the latest large-scale large eddy simulation (hereinafter referred to as LES) analysis technology to solve them and presents the results of confirming the prediction accuracy improvement by the comparison with the maximum lift coefficient of the flight test result for SpaceJet.

2. Current status and issues of conventional CFDs

An analysis method that has been widely used so far in practical product design and development is the Reynolds-averaged Navier-Stokes (hereinafter referred to as RANS) method, which characteristically solves equations incorporating eddies as an averaged model. Although this method allows for reduced computational resources and easy application to design and development due to the modeling, its inherent inability to predict unsteady separation flow with high accuracy because of eddy averaging has been recognized as a major issue. As an example of the insufficient accuracy of unsteady separation flow prediction using RANS, **Figure 1** shows the results of analysis

---

[*1] Fluid Dynamics Department, Research & Innovation Center
[*2] Research Manager, Special Purpose Systems Research Department, Research & Innovation Center
[*3] Professor, Department of Aerospace Engineering, Tohoku University

to predict the maximum lift coefficient of an aircraft model using RANS by various participants conducted at the High Lift Prediction Workshop (2) held by the American Institute of Aeronautics and Astronautics. This figure shows that in the high angle of the attack region, around which the maximum lift coefficient is obtained, the participants' analysis results varied much more quantitatively and qualitatively than the wind tunnel test data, indicating that the highly accurate prediction of the maximum lift coefficient with RANS is very difficult.

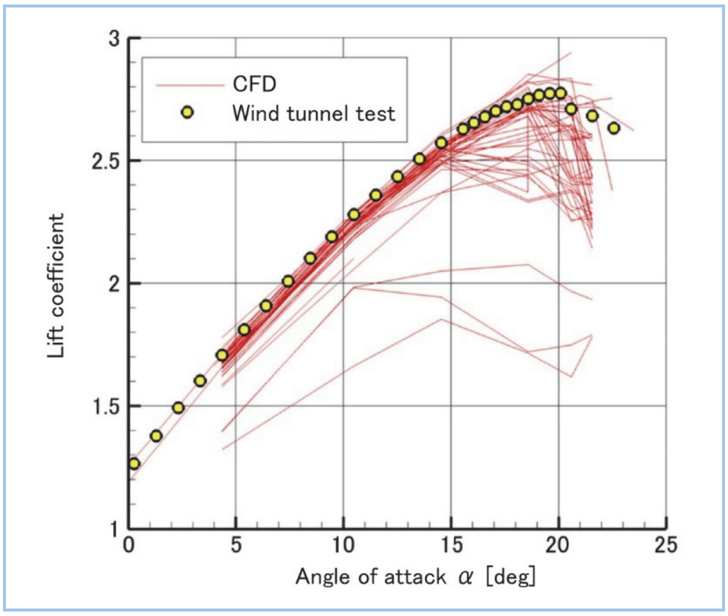

Figure 1 Analysis results of lift coefficient prediction for aircraft geometry

Against such a background, the application of LES, which has higher physical fidelity than RANS, is expected to be applied to the design. Unlike the RANS, which averages eddies, the LES directly captures and analyzes eddies larger than the spatial cell size and models eddies smaller than the cell size, making it highly applicable to various types of turbulent flows. However, until now, the LES has only been applied to basic flow fields of simple geometries and has three major issues that must be resolved before applying it to the design and development of actual aircraft.

The first issue is to achieve both highly accurate predictions and realistic computational resources. The size of turbulent eddies within the near-wall inner-layer of turbulent boundary layer becomes very small, thus the placement of a grid involving a huge number of cells near-wall boundary layer is required to fully resolve them by the LES. In addition, to perform a stable unsteady analysis with very small cells, the time increment of the analysis needs to be extremely small, so it is necessary to keep the number of cells and the computation time from increasing to suppress the computational resources to a realistic level while maintaining the prediction accuracy.

The second issue is to establish a low-numerical dissipation and robust scheme to achieve a highly accurate LES. Conventional numerical schemes include an unphysical parameter called numerical dissipation to make the analysis numerically stable. In the case of LES, the magnitude of this numerical dissipation is of the same order as that of the physical turbulence viscosity, and this contributes to unphysical dissipation in the advection process of eddies and prevents highly accurate analysis. Therefore, an advanced computational scheme that achieves both robustness and low dissipation in the analysis is needed.

The third issue is to reduce the burden of grid generation. Since LES needs to capture turbulence eddies with the grid, the distortion of the grid has a significant impact on the analysis results. Therefore, compared to the grid of RANS, that of LES needs to be isotropic, which increases the difficulty of generating the grid dramatically, especially in the vicinity of the object. In addition, current grid generation tools cannot handle large-scale grid generation involving billions to tens of billions of cells in terms of both hardware and software, and often spend more processing time on grid generation than on the computation itself. Therefore, the load on grid generation must be reduced.

MHI has been introducing and validating FFVHC-ACE, which implements the latest LES analysis technology to solve the above three issues. The next chapter describes the latest technologies that realize large-scale, high-accuracy LES.

3. Key technologies that realize large-scale, high-accuracy LES analysis

The key technologies to solve the three issues mentioned in the previous chapter are the wall-modeled LES (hereinafter referred to as WMLES)(3),(4), the kinetic-energy and entropy preserving (hereinafter referred to as KEEP)(5)~(8) scheme, and the hierarchical equally spaced Cartesian grid(9),(10). Overviews of each are given below.

3.1 WMLES

WMLES is a key technique to achieve both highly accurate prediction and realistic computational resource requirements. LES can be roughly divided into two types depending on how the near-wall region is handled: wall-resolved LES (hereinafter referred to as WRLES), which directly resolves eddies all the way down to the wall, and WMLES, which models the near-wall region. The WMLES developed by Tohoku University is a high-efficiency and high-accuracy model that can capture and analyze the turbulent boundary layer accurately as well as near-wall regions with a very coarse grid, focusing on the fact that the turbulent eddies within the inner-layer of turbulent boundary layer that develop near the wall follow a universal turbulence scaling law. Compared to WRLES, which uses a fine grid (dimensionless parameter $y+\approx 1$ or less, indicating the distance from the wall) to resolve the vicinity of the wall, WMLES can use a very coarse grid ($y+\approx 100$ approximately) for the analysis and allows the time increment, which is limited by the minimum grid width, to be large. For the same analysis as WRLES, WMLES reduces grid scale to about 1/100, analysis time to about 1/100, and computational cost to about 1/10,000 in typical cases, although it also depends on the Reynolds number, thereby realizing high-precision LES with realistic computational resources.

3.2 KEEP scheme

The KEEP scheme is a key technology to capture and analyze turbulence with high accuracy. This scheme has been developed at Tohoku University, as an epoch-making calculation scheme which enables non-numerical dissipation and robust calculation by satisfying the kinetic energy conservation which is not satisfied by the conventional upwind scheme and drastically improving the entropy conservation. **Figure 2** shows an example of analysis using the KEEP scheme. The inviscid Taylor-Green vortex problem shown on the left side of the figure is a test problem in which a large initial vortex breaks down to finer scales over time and the non-dissipative nature of the calculation scheme and computational robustness can be evaluated by following the process of vortex break down. The figure compares the conventional upwind scheme, the central-difference scheme, and the KEEP scheme, with entropy conservation error on the vertical axis and dimensionless time on the horizontal axis. Since this problem is an inviscid flow, even if the vortex breaks down, there is no dissipation due to the physical viscosity, and the entropy is preserved, and the state in which the entropy holds zero is the exact solution. Conversely, an increase in the entropy means that numerical dissipation is occurring, and a decrease in the entropy means that the computation is unstable. This figure indicates that the entropy increases for the upwind scheme, while the entropy immediately decreases for the central-difference scheme. On the other hand, the entropy remains approximately at zero for the KEEP scheme. The right side of the figure visualizes the near-wall eddy structure of the flat turbulent boundary layer for the upwind scheme and the KEEP scheme. This figure shows that the eddies are unphysically dissipated for the upwind scheme, whereas the KEEP scheme is able to capture and analyze fine eddy structures with high resolution. In this way, it is found that the KEEP scheme is a high resolution scheme suitable for LES because it is compatible with robustness and non-dissipation of analysis.

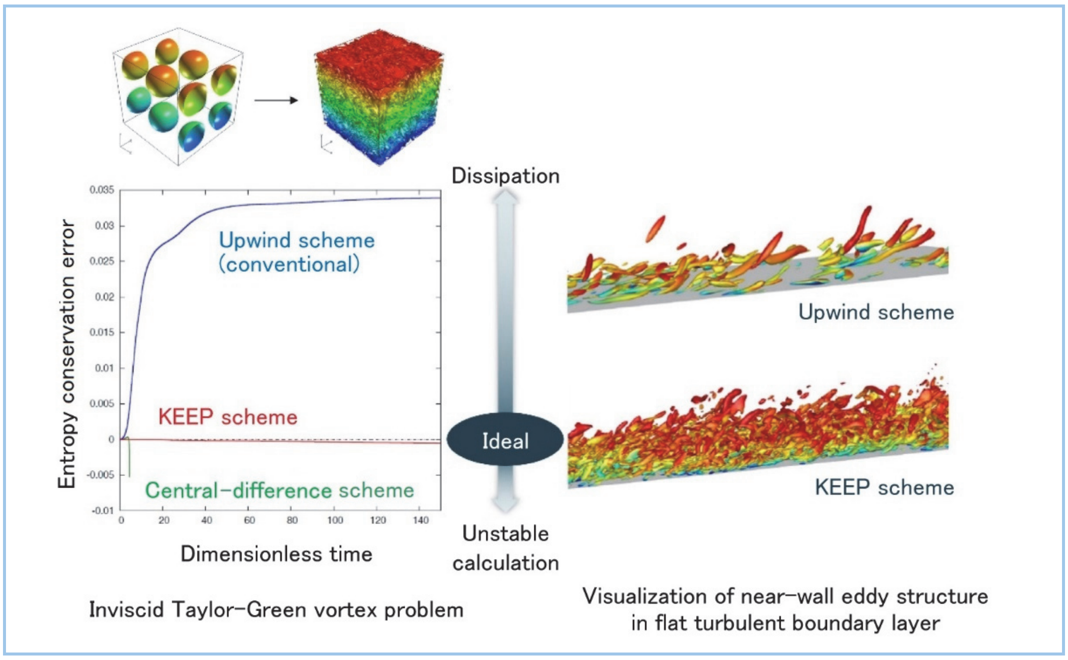

Figure 2 Analysis results of KEEP scheme

3.3 Hierarchical equally spaced Cartesian grid

The hierarchical equally spaced Cartesian grid is a key technology that significantly reduces the grid generation load. **Figure 3** shows a schematic diagram of a hierarchical equally spaced Cartesian grid. Unlike the commonly used method of dividing space with a grid along the surface shape of the aircraft (boundary fitted grid), the hierarchical equally spaced Cartesian grid divides space with equally spaced orthogonal blocks regardless of the surface shape of the aircraft. Then, for regions near walls where high resolution is required, a block is subdivided hierarchically by generating eight further blocks within that block to obtain the final grid. The advantages of this hierarchical equally spaced Cartesian grid are that it produces a high-quality equally spaced and orthogonal grid ideal for CFD and that it can generate a grid completely automatically because there is no dependence on the complexity of the aircraft geometry. Therefore, the grid generation by the user which is necessary for the large-scale LES with boundary fitted grid becomes unnecessary, and the grid generation problem which has hindered the use of the high-fidelity flow simulation in the applied research is solved. Furthermore, since each block contains the same number of cells, the parallelization efficiency to level the computational load is extremely high, thus making the hierarchical equally spaced Cartesian grid possible to handle large-scale computations.

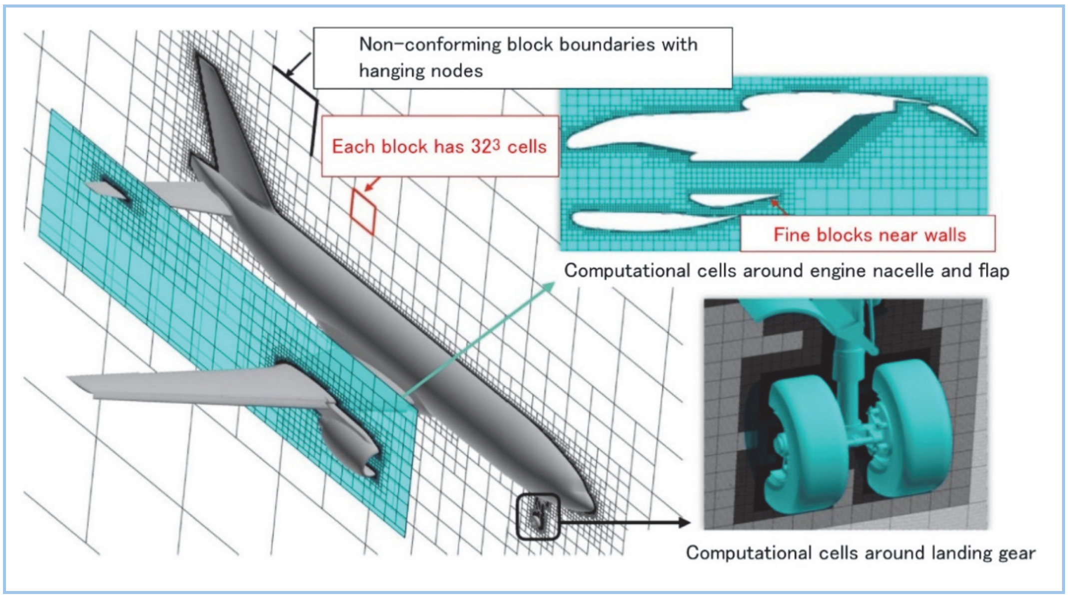

Figure 3 Schematic diagram of hierarchical equally spaced Cartesian grid

The next chapter shows the results of validating the accuracy of analysis with FFVHC-ACE, which implements these key technologies, by comparing with actual aircraft data.

4. Validation of prediction accuracy by comparison with SpaceJet flight test results

We simulated a stall flight test of SpaceJet using FFVHC-ACE, which implements the latest LES analysis technologies to validate its prediction accuracy for the maximum lift coefficient. This analysis was performed using the supercomputer "Fugaku."

Figure 4 shows the shape of the analysis target and the grid distribution in the nearby symmetric cross-section.

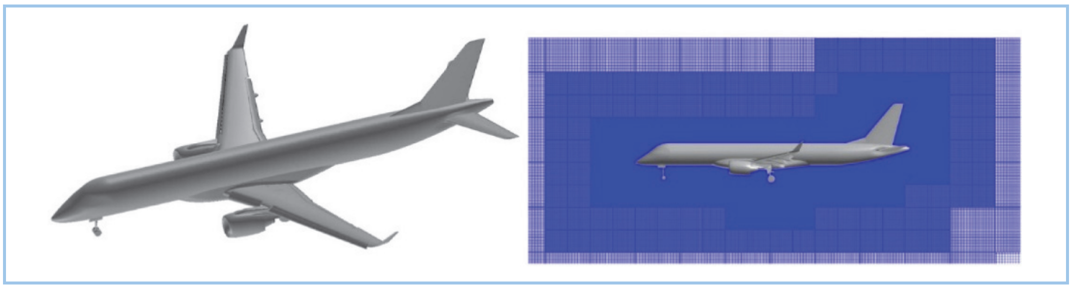

Figure 4 Shape of analysis target and grid distribution in nearby symmetric cross-section

The target configuration is the landing configuration of the SpaceJet, the same as in the flight test. To understand the effect of grid resolution, two grids with different grid sizes were used: Grid 1 containing about 4 billion cells and Grid 2 containing about 20 billion cells. The flight Mach number was about 0.2. To understand the trend of changes in the maximum lift coefficient with the angle of attack, three angles of attack were computed for Grid 1 and two for Grid 2. The required computation time per angle of attack from the start of computation to data acquisition was approximately 2 days for Grid 1 and approximately 1 week for Grid 2, which is within the practically acceptable range.

The number of nodes and cores of Fugaku used for the analysis were 768 nodes/36,864 cores for Grid 1 and 3,840 nodes/184,320 cores for Grid 2 (about 2% of the total computational nodes of Fugaku). It was confirmed that the 20-billion-cell large-scale grid of Grid 2 was able to be generated fully automatically, and the grid generating time was less than 20 minutes, indicating a significant reduction in the grid generation load.

Figure 5 compares the lift coefficients obtained from the flight test results and CFD analysis results. For Grid 1, the CFD analysis results show a good agreement with the test results at the low angle of attack, but a tendency toward underestimation around the angle where the maximum lift coefficient is obtained, at which the lift resulted from the analysis is smaller than that from the test. For Grid 2, on the other hand, the CFD analysis results are closer to the test results compared to the case for Grid 1. In particular, around the angle where the maximum lift coefficient is obtained, the error is as small as 2% or less, confirming the good prediction accuracy.

Next, to understand the factors that cause the difference in the lift coefficient, the streamlines and velocity component along body axis (hereinafter referred to as Ux) distributions for Grid 1 and Grid 2 are plotted in **Figure 6**. In the regions where the streamlines are turbulent and Ux is shown in blue, the flow is separated and Ux is reduced or reversed. Comparison of Grid 1 and Grid 2 shows that Grid 1 shows a larger separation than Grid 2. In the case of Grid 1, it is considered that the pressure on the upper surface of the wing increased due to the reduction in Ux caused by the separation there, resulting in a decrease in the overall lift of the aircraft. These results show that the 4-billion-cell-scale grid is not sufficient and a larger grid is needed for predicting the maximum lift coefficient where the separation plays an important role.

To understand the flow fields around the aircraft, the unsteady turbulent structures for Grid 2 at the angle of attack where the maximum lift is obtained is visualized in **Figure 7**. The turbulent structures are colored by Ux with blue, green, and red indicating higher velocities in that order. This figure shows that fine eddies can be captured and analyzed, including the longitudinal vortices from the slat track rails placed at regular intervals on the leading edge of the wing, the interference between the longitudinal vortices from the nacelle chine and the eddies from the inner side of nacelle, and the eddies flowing from the nose landing gear along the fuselage.

As described above, we validated the prediction accuracy of analysis with the FFVHC-ACE for the maximum lift coefficient, which has been difficult to predict accurately in the past, by comparing with the flight test results, and confirmed the improvement in the prediction accuracy as a result.

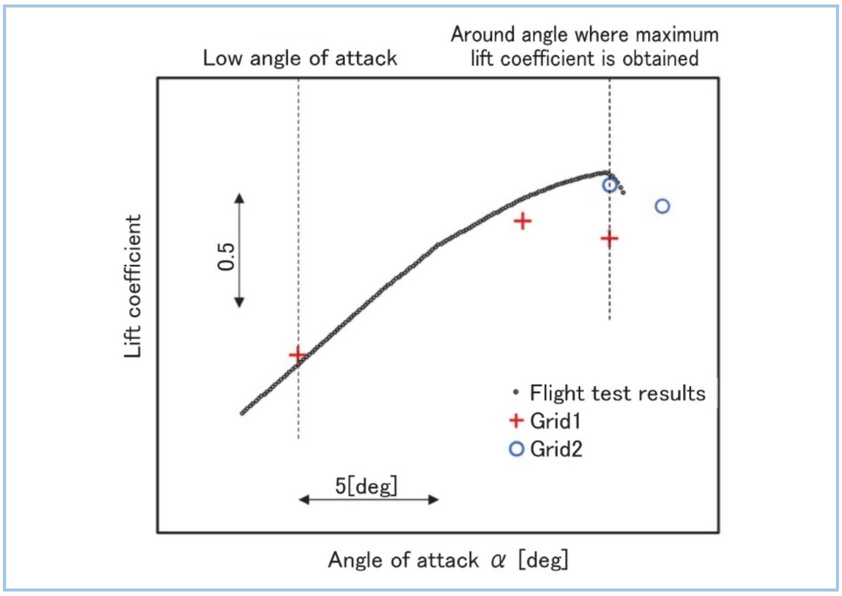

Figure 5 **Comparisons of lift coefficient in terms of angle of attack obtained from flight test results and analysis results**

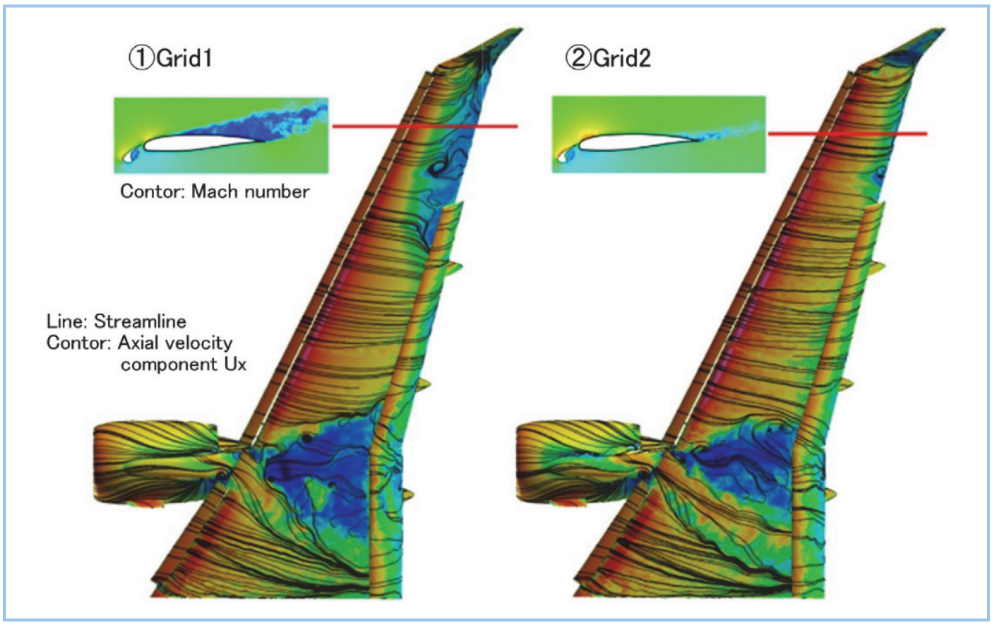

Figure 6 **Surface streamlines and axial velocity and Mach number cross-section distributions for Grid 1 and Grid 2**

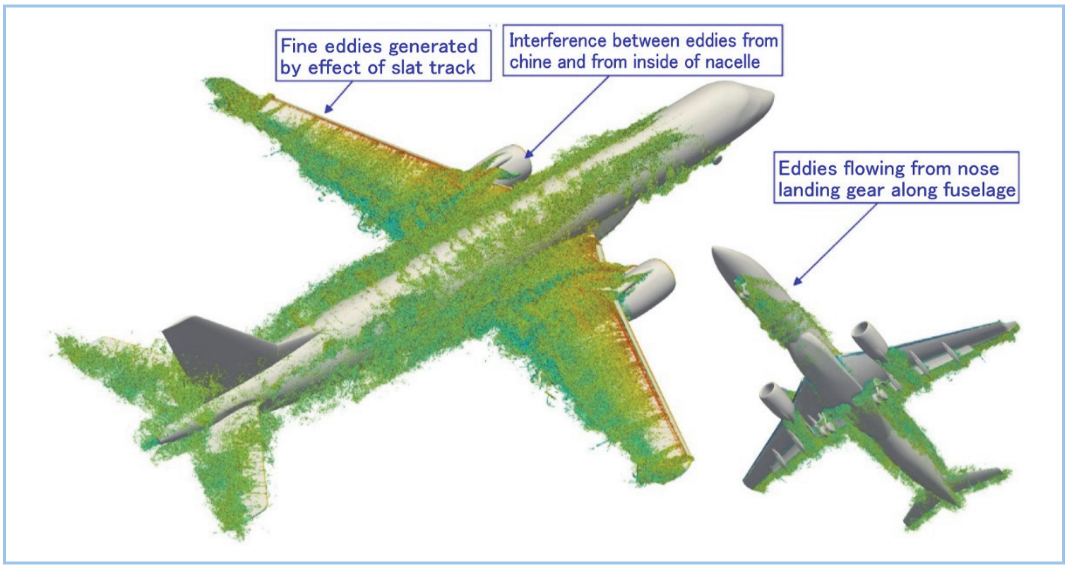

Figure 7　Visualization of turbulent structures colored by axial velocity

5. Conclusion

We validated the prediction accuracy of analysis with the FFVHC-ACE, which implements the latest LES analysis technologies, for the maximum lift coefficient of aircraft, which has been difficult to predict accurately with the conventional CFD, and confirmed as a result that its prediction accuracy was excellent with the error within 2% compared to actual aircraft flight test results, demonstrating the applicability of large-scale LES to actual problems.

Future developments include the application to prediction of transonic buffet and aircraft noise. Accurate prediction of transonic buffet caused by shock wave oscillations interacting with the turbulent boundary layer separation on the wings is important for flight safety. Therefore, the KEEP scheme, which currently does not handle shock waves, needs to be adapted to handle shock waves. It is also important to accurately predict aircraft noise from the standpoint of environmental compatibility. The FFVHC-ACE's features of low numerical dissipation and detailed vortex resolution make it suitable for acoustic analysis, so we consider that continued validation is necessary. We will continue to validate the applicability of the analysis technology through joint research with Tohoku University to contribute to the development of our products.

This work was supported in part by Ministry of Education, Culture, Sports, Science and Technology (MEXT) as “Program for Promoting Researches on the Supercomputer Fugaku” (Leading Research on Innovative Aircraft Design Technologies to Replace Flight Test, JPMXP1020200312) and used computational resources of supercomputer Fugaku provided by RIKEN Center for Computational Science (Project IDs: hp200137, hp210168, hp220160).

References

(1) Asada, Hiroyuki, et al. FFVHC-ACE: fully automated Cartesian-grid-based solver for compressible large-eddy simulation. AIAA Journal 61.8 (2023): 3466-3484.

(2) Rumsey, Christopher L. High-Lift Prediction Workshops: Retrospective, Lessons Learned, and Future Prospects. 34th Congress of the International Council of the Aeronautical Sciences (ICAS). 2024.

(3) Kawai, Soshi, et al. Wall-modeling in large eddy simulation: Length scales, grid resolution, and accuracy. Physics of Fluids 24.1 (2012).

(4) Tamaki, Yoshiharu, et al. Wall modeling for large-eddy simulation on non-body-conforming Cartesian grids. Physical Review Fluids 6.11 (2021): 114603.

(5) Kuya, Yuichi, et al. Kinetic energy and entropy preserving schemes for compressible flows by split convective forms. Journal of Computational Physics 375 (2018): 823-853.

(6) Kuya, Yuichi, et al. A stable and non-dissipative kinetic energy and entropy preserving (KEEP) scheme for non-conforming block boundaries on Cartesian grids. Computers & Fluids 200 (2020): 104427.

(7) Kuya, Yuichi, et al. High-order accurate kinetic-energy and entropy preserving (KEEP) schemes on curvilinear grids. Journal of Computational Physics 442 (2021): 110482.

(8) Tamaki, Yoshiharu, et al. Comprehensive analysis of entropy conservation property of non-dissipative schemes for compressible flows: KEEP scheme redefined. Journal of Computational Physics 468 (2022): 111494.

(9) Nakahashi, Kazuhiro. Aeronautical CFD in the age of Petaflops-scale computing: From unstructured to Cartesian meshes. European Journal of Mechanics-B/Fluids 40 (2013): 75-86.

(10) Lintermann, Andreas, et al. Massively parallel grid generation on HPC systems. Computer Methods in Applied Mechanics and Engineering 277 (2014): 131-153.