Current Status and Challenge of CFD for Aircraft Developments

Kazuhiro Nakahashi
Department of Aerospace Engineering, Tohoku University

ABSTRACT

Impressive progress in computational fluid dynamics (CFD) has been made during the last three decades. Currently CFD has become an indispensable tool for analyzing and designing aircrafts. Wind tunnel testing, however, is still the central player for aircraft developments and CFD plays a subordinate part. In this article, demands for next-generation CFD are described with an expectation of near future PetaFlops computers. Then, Cartesian grid approach, as a promising candidate for next-generation CFD, is discussed by comparing it with the current unstructured grid CFD. It is concluded that the simplicity of the algorithms from grid generation to post processing of Cartesian mesh CFD will be a big advantage in the days of PetaFlops computers.

1. Will CFD take over wind tunnels?

More than 20 years ago, I heard an elderly physicist in fluid dynamics say that it was as if CFD were just surging in. Other scientists of the day said that with the development of CFD, wind tunnels would eventually become redundant.

Impressive progress in CFD has been made during the last three decades. In the early stage, one of the main targets of CFD for aeronautical fields was to compute flow around airfoils and wings accurately and quickly. Body-fitted-coordinate grids, commonly known as structured grids, were used in those days.

From the late eighty's, the target was moved to analyzing full aircraft configurations [1]. This spawned a surge of activities in the area of unstructured grids, including tetrahedral grids, prismatic grids, and tetrahedral-prismatic hybrid grids. Unstructured grids provide considerable flexibility in tackling complex geometries as shown in Figure 1 [2]. CFD has become an indispensable tool for analyzing and designing aircrafts.

The author has been studying various aircraft configurations with his students using the advantages of unstructured grid CFD as shown in Figures 2.

So, is CFD taking over the wind tunnels as predicted twenty years ago?

Today, Reynolds-averaged Navier-Stokes (RANS) computations can accurately predict lift and drag coefficients of a full aircraft configuration. It is, however, still quantitatively not reliable for high-alpha conditions where flow separates. Boundary layer transition is another cause of inaccuracy. These are mainly due to the incompleteness of physical models used in RANS simulations.

Large Eddy Simulation (LES) and Direct Numerical Simulation (DNS) are expected to reduce the physical model dependencies. But we have to wait for the further progress of computers for the use of those large-scale computations in engineering purposes.

For the time being, the wind tunnel is the central player and CFD plays a subordinate part in aircraft developments.

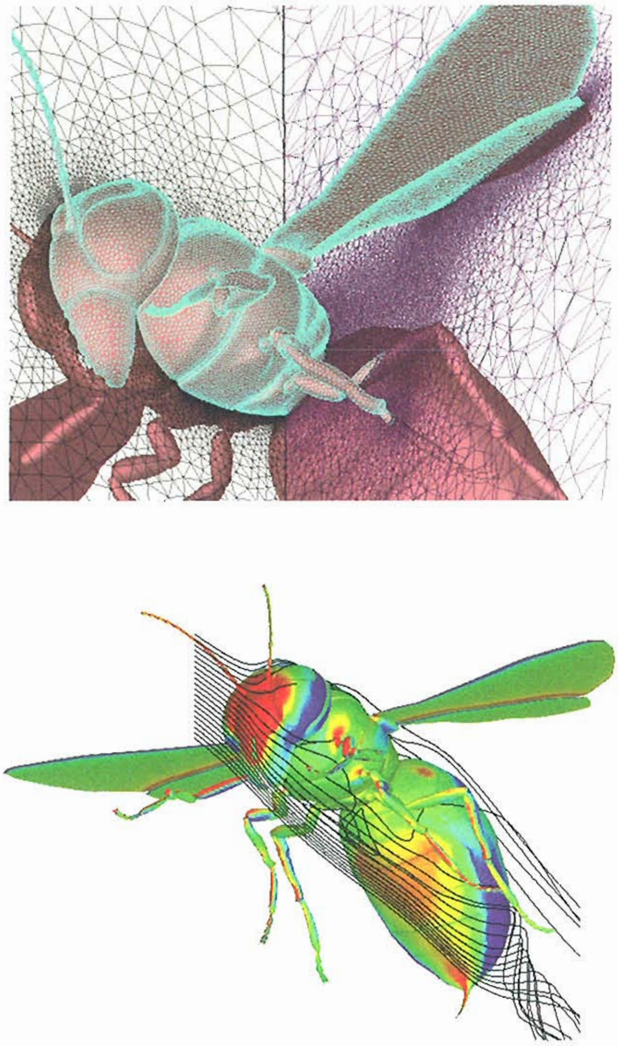


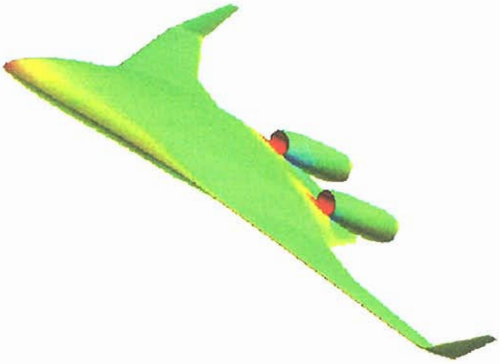
Figure 1: Flow computation around a hornet by unstructured grid CFD [2]

2. Rapid progress of computers

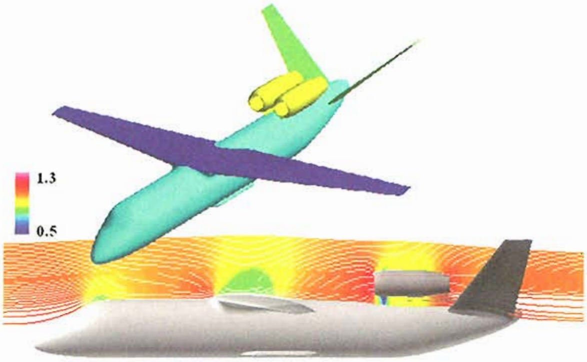
The past CFD progress has been highly supported by the improvements of computer performance. Moor's Law tells us that the degree of integration of a computer chip has been doubled in 18 months. This basically corresponds to a factor of 100 every 10 years. The latest Top500 Supercomputers Sites [7], on the other hand, tell us that the performance improvement of computers has reached a factor of 1000 in the last 10 years as shown in Figure 3. Increase in the number of CPUs in a system in addition to the degree of integration contributes to this rapid progress.

With a simple extrapolation of Figure 3, we can expect to use PetaFlops computers in ten years. This will accelerate the use of 3D RANS computations for the aerodynamic analysis and design of entire airplanes. DNS which does not use any physical models may also be used for engineering analysis of wings.

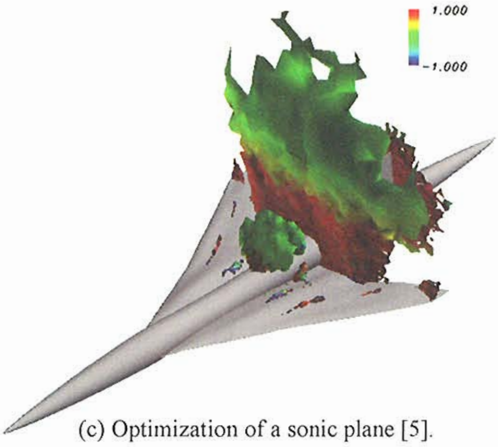
In the not very far future, CFD could take over wind tunnels.



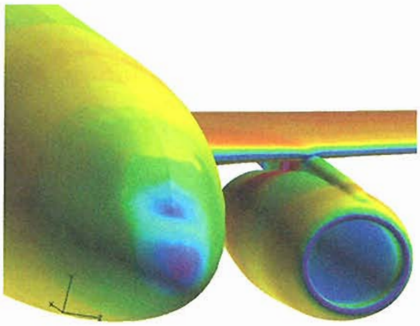
(a) Design of a medium-size Blended-Wing-Body airplane [3].



(b) Design of a quiet jet transport [4].



(c) Optimization of a sonic plane [5].



(d) Optimization of wing-nacelle-pylon configurations [6].

Figure 2: Applications of unstructured grid CFD to design and analysis of various airplanes studied at Tohoku University.

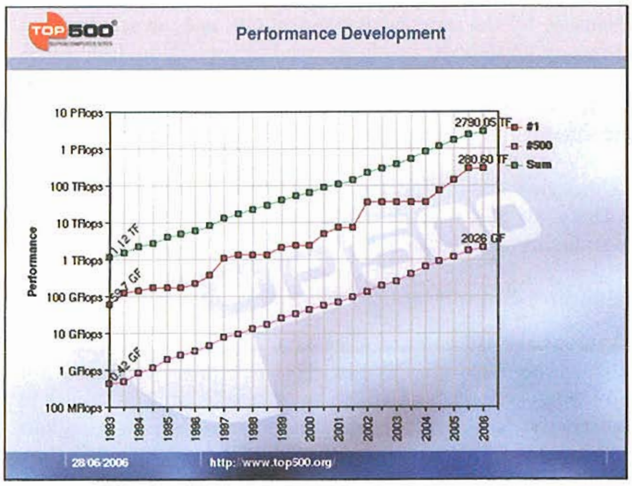


Figure3: Performance development in Top500 Supercomputers [7].

3. Demands for next-generation CFD

So, is it enough for us as CFD researchers to just wait for the progress of computers? Probably it is not.

Let's consider demands for next-generation CFD on PetaFlops computers.

1. *Easy and quick grid generation around complex geometries,*
2. *Easy adaptation of local resolution to local flow characteristic length,*
3. *Easy implementation of spatially higher-order schemes,*
4. *Easy massively-parallel computations,*
5. *Easy post processing for huge data output,*
6. *Algorithm simplicity for software maintenance and update.*

Unstructured grid CFD is a qualified candidate for the demands 1 and 2 as compared to structured grid CFD. However, an implementation of higher-order schemes on unstructured grids is not easy. Post processing of huge data output may also become another bottleneck due to irregularity of the data structure.

Recently, studies of Cartesian grid method were renewed in the CFD community, because of the several advantages such as rapid grid generation, easy higher-order extension, and simple data structure for easy post processing. This is another candidate for the next-generation CFD.

Let's compare the computational cost of uniform Cartesian grid methods with that of tetrahedral unstructured grids. The most time-consuming part in compressible flow simulations is the numerical flux computations. The number of flux computations on a cell-vertex, finite volume method is proportional to the number of edges in the grid. In a tetrahedral grid, the number of edges is at least twice of that of the edges in a Cartesian grid of the same number of node points. Therefore, the computational costs on unstructured grids are at least twice as large as the costs of Cartesian grids. Moreover, computations of linear reconstructions, limiter functions, and implicit time integrations on tetrahedral grids easily doubles the total computational costs.

For higher-order spatial accuracy, the difference of computational costs between two approaches expands rapidly. In Cartesian grids, the spatial accuracy can be easily increased up to the fourth order without extra computational costs. In contrast, to increase the spatial accuracy from second to third-order on unstructured grids can easily increase tenfold the computational cost.

Namely, for the same computational cost and the same spatial accuracy of third-order or higher, we can use 100 to 1000 times more grid points in the Cartesian grid than in unstructured grid. The increase of grid points improves the accuracy of geometrical representation in computations as well as the spatial solution accuracy.

Although the above estimate is very rough, it is apparent that the Cartesian grid CFD is a big advantage for high resolution computations required for DNS.

4. Building-Cube Method

A drawback of uniform Cartesian grid is the difficulty of changing the mesh size locally. This is critical, especially for airfoil/wing computations, where an extremely large difference in characteristic flow lengths exists between boundary layer regions and far fields. Accurate representation of curved boundaries by Cartesian meshes is another issue.

A variant of the Cartesian grid method is to use the adaptive mesh refinement [8] in space and cut cells or the immersed boundary method [9] on the wall boundaries. However, introduction of irregular subdivisions and cells into Cartesian grids complicate the algorithm for higher-order schemes. The advantages of the Cartesian mesh over the unstructured grid, such as simplicity and less memory requirement, disappear.

The present author proposes a Cartesian grid based approach, named Building-Cube method [10]. Basic strategies employed here are; (a) zoning of a flow field by cubes (squares in 2D as shown in Figure 4) of various sizes to adapt the mesh size to local flow characteristic length, (b) uniform Cartesian mesh in each cube for easy implementation of higher-order schemes, (c) same grid size in all cubes for easy parallel computations, (d) staircase representation of wall boundaries for algorithm simplicity.

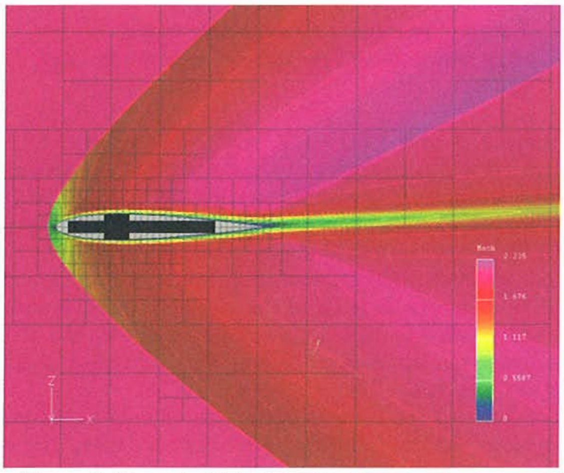


Figure 4: Computed Mach distribution around NACA0012 airfoil at Re=5000, $M_\infty = 2$ and α=3 deg.

It is similar to a block-structured uniform Cartesian mesh approach [11], but unifying the block shape to a cube simplifies the domain decomposition of a computational field around complex geometry. Equality of computational cost among all cubes significantly simplifies the massively parallel computations. It also enables us to introduce data compression techniques for pre and post processing of huge data [12].

A staircase representation of curved wall boundaries requires a very small grid spacing to keep the geometrical accuracy. But the flexibility of geometrical treatments obtained by it will be a strong advantage for complex geometries and their shape optimizations. An example is shown in Figure 5 where a tiny boundary layer transition trip attached to an airfoil is included in the computational model. Figures 6 are the computed pressure distributions which show the detailed flow features including the effect of trip wire, interactions between small vortices and the shock wave, and so on.

The result was obtained by solving the two dimensional Navier-Stokes equations. We did not use any turbulence models, but just used a high-density Cartesian mesh and a fourth-order scheme. This 2D computation may not describe the correct flow physics, since the three-dimensional flow structures are essential in the turbulent boundary layers for high-Reynolds number flows. However, the result indicates that a high-resolution computation using a high-density Cartesian mesh is very promising with a progress of computers.

5. Simplicity is essential for next-generation CFD

CFD, using a high-density Cartesian mesh, is still limited in its application due to the computational cost. The predictions about Cartesian mesh CFD and computer progress in this article may be too optimistic. However, it is probably correct to say that the simplicity of the algorithm from grid generation to post processing of Cartesian mesh CFD will be a big advantage in the days of PetaFlops computers.

6. References

[1] A. Jameson and T. J. Baker, "Improvements to the Aircraft Euler Method," *AIAA Paper* 1987-452 (1987).

[2] K. Nakahashi, Y. Ito, and F. Togashi, "Some challenges of realistic flow simulations by unstructured grid CFD", *Int. J. for Numerical Methods in Fluids*, Vol.43, pp.769-783 (2003).

[3] T. E. Pambagjo, K. Nakahashi, and K. Matsushima, "An Alternate Configuration for Regional Transport Airplane," *Transactions of the Japan Society for Aeronautical and Space Sciences,* Vol.45, No.148, pp.94-101 (2002), and Tjoetjoek Eko Pambagjo, "Application of Inverse Method to Blended-Wing-Body Airplane Design," *Doctor dissertation, Tohoku University,* February 2002.

[4] K. Nakahashi and M. Morita, "Configuration Study of a Quiet Jet Transport Airplanes (in Japanese)," *Proc. of the 34th JSASS Annual Meeting,* pp.284-287 (2003), and M. Morita, "Study of a Quiet Jet Transport (in Japanese)", *Master dissertation, Tohoku University*, February 2004.

[5] W. Yamazaki, K. Matsushima, and K. Nakahashi, "Drag Reduction of a Near-Sonic Airplane by using Computational Fluid Dynamics," *AIAA J.,* Vol.43, No.9, pp.1870-1877 (2005), and Wataru Yamazaki, "Aerodynamic Shape Optimization of a Sonic Plane," *Master dissertation, Tohoku University*, February 2004.

[6] S. Koc, H. Kim and K. Nakahashi, "Aerodynamic Design of Wing-Body-Nacelle-Pylon Configuration," *AIAA*-2005-4856, *17th AIAA CFD Conf.* (2005), and Salim Koc, "Aerodynamic Shape Optimization Method for Complex Aircraft Configurations," *Doctor dissertation, Tohoku University,* July 2005.

[7] Top500 Supercomputers Sites, http://www.top500.org/

[8] M. Berger and M. Oliger, "Adaptive Mesh Refinement for Hyperbolic Partial Differential Equations," *J. Comp. Physics,* Vol.53, pp.561-568 (1984).

[9] R. Mittal and G. Iaccarino, "Immersed Boundary Methods," *Annual Review of Fluid Mechanics,* Vol. 37, pp239-261 (2005).

[10] K. Nakahashi, "Building-Cube Method for Flow Problems with Broadband Characteristic Length," *Computational Fluid Dynamics 2002*, edited by S. Armfield, et. al., Springer, pp.77-81 (2002).

[11] R. L. Meakin and A. M. Wissink, "Unsteady Aerodynamic Simulation of Static and Moving Bodies Using Scalable Computers," *AIAA*-99-3302, *Proc. AIAA 14th CFD Conf.* (1999).

[12] K. Nakahashi, "High-Density Mesh Flow Computations with Pre-/Post-Data Compressions," *AIAA* 2005-4876, *Proc. AIAA 17th CFD Conf.* (2005).

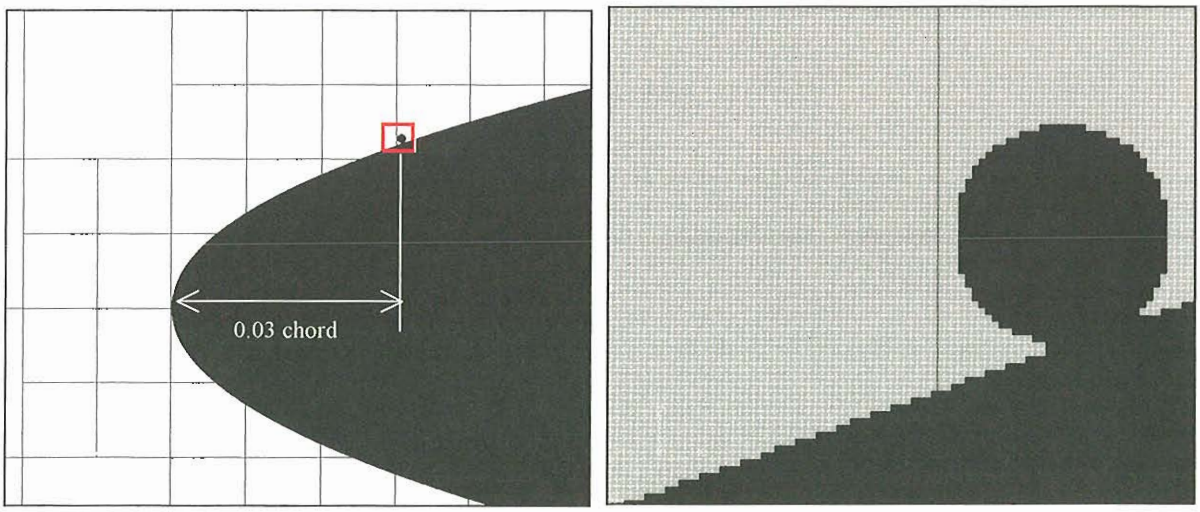


Figure 5: Cube frames around RAE2822 airfoil (left) and an enlarged view of Cartesian grid near tripping wire (right).

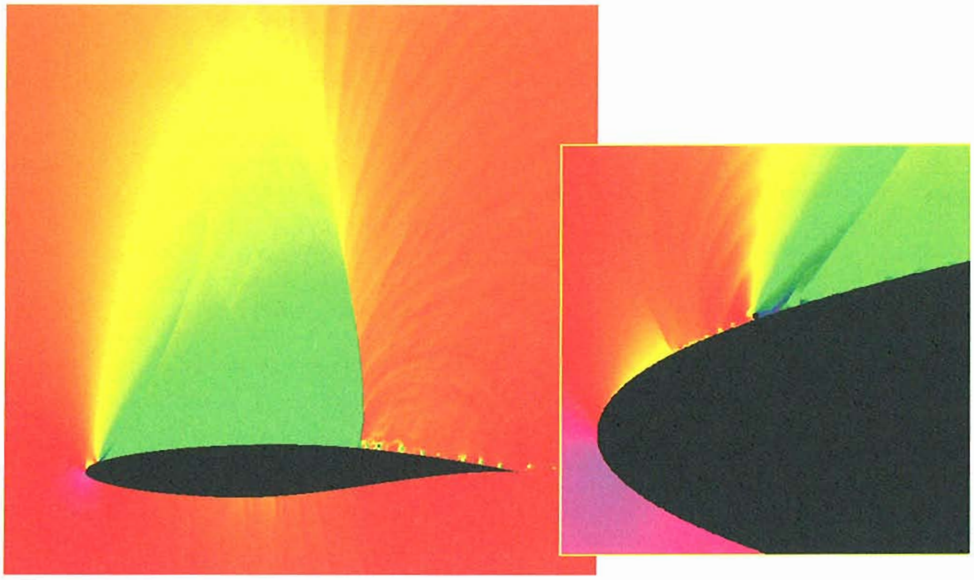


Figure 6: Computed pressure distributions around RAE2822 airfoil at Re=6.5x10^6, M_∞=0.73, α=2.68 deg.