

Toward large eddy simulation of turbulent flow over an airfoil

By Haecheon Choi¹

1. Motivation and objectives

The flow field over an airfoil contains several distinct flow characteristics, e.g. laminar, transitional, turbulent boundary layer flow, flow separation, unstable free-shear layers, and a wake. This diversity of flow regimes taxes the presently available Reynolds averaged turbulence models. Such models are generally tuned to predict a particular flow regime, and adjustments are necessary for the prediction of a different flow regime. Similar difficulties are likely to emerge when the large eddy simulation technique is applied with the widely used Smagorinsky model. This model has not been successful in correctly representing different turbulent flow fields with a single universal constant and has an incorrect near-wall behavior.

Germano *et al.* (1991) and Ghosal, Lund & Moin (1992) have developed a new subgrid-scale model, the *dynamic* model, which is very promising in alleviating many of the persistent inadequacies of the Smagorinsky model: the model coefficient is computed dynamically as the calculation progresses rather than input *a priori*. The model has been remarkably successful in prediction of several turbulent and transitional flows.

We plan to simulate turbulent flow over a "2D" airfoil using the large eddy simulation technique. Our primary objective is to assess the performance of the newly developed dynamic subgrid-scale model for computation of complex flows about aircraft components and to compare the results with those obtained using the Reynolds average approach and experiments. The present computation represents the first application of large eddy simulation to a flow of aeronautical interest and a key demonstration of the capabilities of the large eddy simulation technique.

2. Accomplishments

2.1 Code modification

For the simulation of the flow over an airfoil, we have modified the direct numerical simulation code of Choi, Moin & Kim (1993), which was successfully applied to simulation of turbulent flow over longitudinal riblets. The code uses generalized coordinates in two dimensions and a Cartesian coordinate in the third direction. The code is based on the unsteady, three-dimensional, incompressible Navier-Stokes equations along with the equation of continuity. The fully implicit time integration scheme, which uses an approximate factorization method in conjunction with Newton-iterative scheme, allows rather large time-steps and, therefore, reduces the

¹ Present address: Seoul National University, Korea

computational effort for solving the Poisson equation. The code makes use of a multi-grid method in order to accelerate the convergence rate of the Poisson solver.

The convective outflow boundary condition is used at the flow exit, and the unsteady “turbulent” inflow condition is similar to that developed by Lee *et al.* (1992). No-stress or uniform velocity boundary conditions are applied at far-field boundaries far from the airfoil surface. Periodic boundary condition is used in the spanwise direction. Arbitrary velocity boundary condition on the surface of the airfoil is also incorporated in the code for future flow control applications.

The *C*-type mesh configuration is used in this study (Thompson *et al.* 1985). This grid is better than the *O*-type grid for flows with massive separation because the wake region after bluff bodies can be more adequately resolved with the *C*-type grid. The boundary condition along the branch-cut in the *C*-type grid is treated *implicitly*, i.e. periodic boundary conditions are applied along the branch cut, which makes the code significantly more complicated. The commonly used *explicit* interpolation technique using adjacent grids near the branch cut deteriorates the time accuracy due to the time-lagging nature of the explicit method. However, the implicit boundary condition along the branch cut used in this study maintains the overall time accuracy. The use of explicit interpolation method along the branch cut does not accurately predict the Strouhal number for the flow over a cylinder, while the implicit boundary condition predicts the Strouhal number very well compared with the experimental results (see section 2.2).

The development of the dynamic subgrid-scale model (Germano *et al.* 1991; Ghosal, Lund & Moin 1992) is the key inducement for the undertaking of the present computation. The dynamic subgrid-scale model is incorporated in the code (in generalized coordinates).

2.2 Code verification

In order to verify the code, we first applied it to the laminar flow over a circular cylinder at the Reynolds number, $Re = u_\infty d / \nu = 100$, where u_∞ is the upstream velocity, d is the cylinder diameter, and ν is the kinematic viscosity. At this Reynolds number, the flow is periodic, and experimental data are available for comparison.

The flow domain and grid system are shown in figure 1. The flow domain covers about 25 and 20 cylinder diameters in the streamwise and normal directions, respectively. A uniform velocity field, $u = u_\infty$ and $v = 0$, is prescribed at $t = 0$. Here, u and v are the velocity components in the streamwise (x) and normal (y) directions, respectively. An initial random disturbance with magnitude of $0.01u_\infty$ is imposed at the flow field in order to induce vortex shedding behind the cylinder.

Figure 2 shows the time history of the streamwise velocity at several points behind the cylinder. After a transient period ($tu_\infty/d \approx 0 \sim 100$), the flow behind the cylinder shows a periodic behavior. The Strouhal number is calculated from figure 2, $S = d/(Tu_\infty) = 0.163$, where T denotes the period of the flow oscillation. This result is in a very close agreement with the experimental result by Williamson (1989), where $S = 0.164$ (see figure 3).

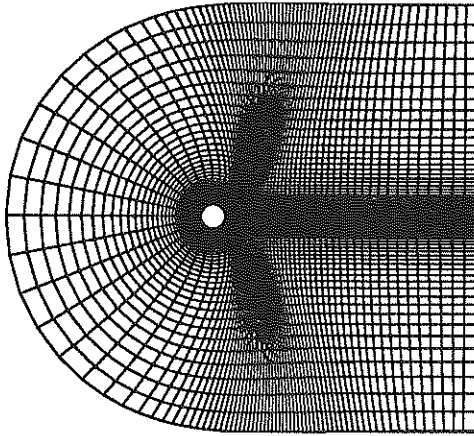


FIGURE 1. Computational domain and grid system. The C -type mesh configuration is used for flow over a cylinder. The branch cut is located along the centerline of the cylinder wake.

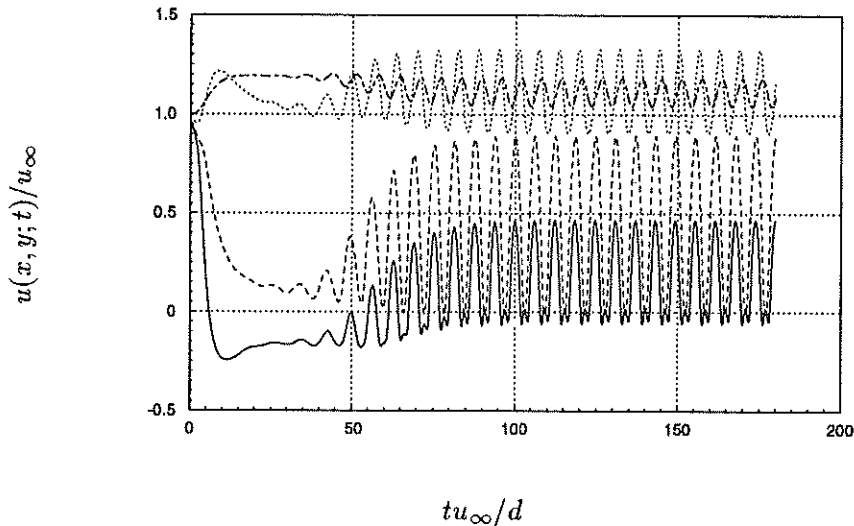


FIGURE 2. Time history of the streamwise velocity at several points ($x = 2.4d$) behind the cylinder: —, $y = -0.21d$; ----, $y = -0.52d$; ·····, $y = -1.1d$; - · - ·, $y = -2.1d$. Here, the position $x = y = 0$ corresponds to the center of the cylinder.

It is interesting to note that the periodic behavior behind the cylinder was not found when an explicit interpolation method using adjacent points was used as

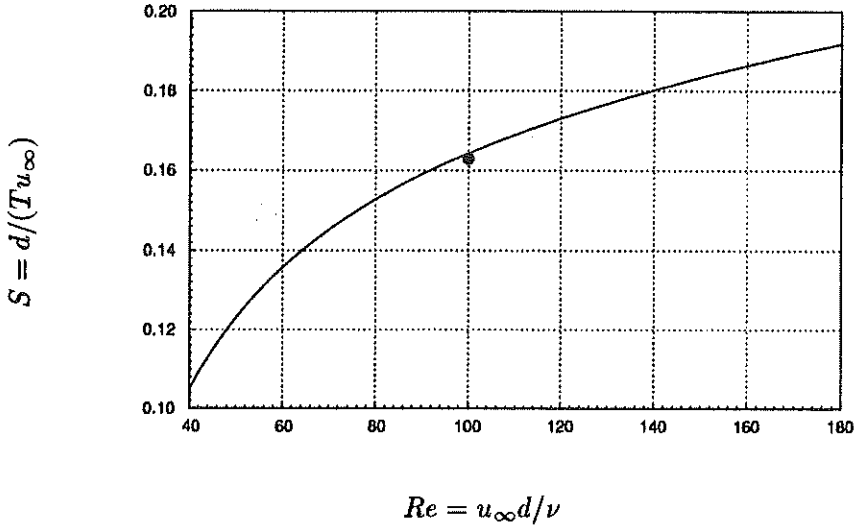


FIGURE 3. Variation of S with Re : —, the least square curve fit from Williamson (1989), $S = A/Re + B + CRe$, where $A = -3.3265$, $B = 0.1816$, $C = 1.6 \times 10^{-4}$; •, the present calculation.

the branch-cut boundary condition. It turns out that the use of the explicit interpolation method along the branch cut indirectly prescribes zero pressure-gradient across the branch cut. Apparently, the absence of oscillating pressure force across the branch cut precludes vortex generation behind the cylinder.

3. Future plans

Future research plan includes two phases.

Phase 1: Laminar test cases will be computed and compared with experimental results. We will also compare laminar flow solutions of the NACA 0012 airfoil with those of a flat-plate airfoil for non-zero angles of attack in order to investigate the effect of the finite thickness of the airfoil on laminar separation characteristics.

Phase 2: We will first simulate turbulent flow over an airfoil with zero angle of attack using the dynamic subgrid-scale model and compare the solution with experimental results and with those by Reynolds averaged turbulence closure models. The angle of attack for maximum lift (or for maximum lift-drag ratio) will be considered subsequently.

REFERENCES

- CHOI, H., MOIN, P. & KIM, J. 1993 Direct numerical simulation of turbulent flow over riblets. *J. Fluid Mech.* **255**, 503.

- GERMANO, M., PIOMELLI, U., MOIN, P. & CABOT, W. H. 1991 A dynamic subgrid-scale eddy viscosity model. *Phys. Fluids A* **3**, 1760.
- GHOSAL, S., LUND, T. S. & MOIN, P. 1992 A dynamical localization model for large eddy simulation of turbulent flows. *CTR Manuscript 139*. Center for Turbulence Research, Stanford U./NASA Ames.
- LEE, S., LELE, S. & MOIN, P. 1992 Simulation of spatially evolving turbulence and the application of Taylor's hypothesis in compressible flows. *Phys. Fluids A* **4**, 1512.
- THOMPSON, J. F., WARSI, Z. U. A. & MASTIN, C. W. 1985 *Numerical Grid Generation - Foundations and Applications*. Elsevier Science Publishing Co., New York.
- WILLIAMSON, C. H. K. 1989 Oblique and parallel modes of vortex shedding in the wake of a circular cylinder at low Reynolds number. *J. Fluid Mech.* **206**, 579.