Simulation of a Flow around an Airfoil with a Circulation Boundary Condition

○Kunio Kuwahara (ISAS) and Satoko Komurasaki (Nihon Univ.)

Key Words: Airfoil simulation, The incompressible Navier-Stokes equations, Subsonic flow, Circulation, Stall

Abstract

In the present paper, a simulation of a subsonic flow around airfoil NACA0012 is attacked. Computations are done at angles of attack 8° to 18° and at the Reynolds number of 10^6 based on the chord length. In this computation, the incompressible Navier-Stokes equations are solved by the multi-directional finite-difference method with using an O-type grid system. For high Reynolds number flow, no explicit turbulence models are employed, but a third-order upwind scheme is adopted. On the far boundary, circulation is corrected consistently, and a periodic boundary condition is used in the spanwise direction. Results show that circulation correction on the far boundary has effect on the flow field, especially, it increases the value of Cl at low angles of attack.

1 Introduction

A flow around an airfoil is one of the most fundamental problems in aerodynamics. Many simulations have been done but some important problems still remain unsolved.

In the present paper, as one of those unsolved problems, a simulation of a subsonic flow over an airfoil near the stall angle, is attacked. Most successful simulations of this kind at high Reynolds numbers are based on the third-order upwind formulation[1]. To increase the accuracy, we employ the multi-directional finite-difference method [2, 3].

Kuwahara and Komurasaki computed a 3-d subsonic flow around an airfoil under a free-slip boundary condition[4] and a periodic boundary condition[5], in the spanwise direction. In these papers, the main flow properties were captured and Cl agreed well with the experimental values, without any explicit turbulence model.

In the present paper, we develop the previous paper[5] with consistent circulation control on the far boundary.

2 Computational method

The governing equations are the 3-d incompressible Navier-Stokes equations and the equation of continuity as follows:

$$\mathrm{div}\boldsymbol{u} = 0 \tag{1}$$

$$\frac{\partial \boldsymbol{u}}{\partial t} + \boldsymbol{u}\cdot\mathrm{grad}\boldsymbol{u} = -\mathrm{grad}p + \frac{1}{Re}\Delta\boldsymbol{u}. \tag{2}$$

where $\boldsymbol{u}, p, t$ and Re denote the velocity vector, pressure, time and the Reynolds number respectively. For high-Reynolds-number flows, time-dependent computations are required owing to the strong unsteadiness.

The numerical procedure is based on the projection method. The pressure field is obtained by solving the following Poisson's equation:

$$\Delta p = -\mathrm{div}(\boldsymbol{u}\cdot\mathrm{grad}\boldsymbol{u}) + \frac{D^n}{\delta t} \tag{3}$$
$$D = \mathrm{div}\boldsymbol{u},$$

where n is the time step and δt is the time increment. D^{n+1} is assumed to be zero, but D^n is retained as a corrective term.

The equations are discretized based on the multi-directional finite-difference method. Space derivatives are discretized using second order central difference approximation with the exception of the convective terms. For the convective terms, a third-order upwind scheme is used to stabilize the computation. It has been found to be the most suitable for high Reynolds number flow computations. The second-order Crank-Nicolson implicit scheme is used for time integration. A body-fitted coordinates system O-grid is employed, and grid points can be concentrated near the body surface. The number of grid size is 129*65*65 (fig.1). For airfoil simulation, C-grid is usually used to avoid the trailing edge singularity. To make C-grid is not easy for high angles of attack, and this is another reason of the difficulty to simulate the flow at high angles of attack. Also C-grid needs unnecessarily concentrated grid points in the near wake region beginning from the trailing edge. This makes the computation unstable. On the other hand O-grid is, in every sense, much better if the computation converges.

Figures 2 (a) and (b) show circulation profiles at angles of attack in case with and without circulation correction on the far boundary. In previous computations, velocity on the far boundary was given as steady free stream because it is not easy to give velocity condition on the boundary for O-grid. In this case, circulation on the far boundary is always zero. However, it is not consistent that circulation is always zero in computational domain for airfoil flow because starting vortices generated at the trailing edge never disappear even after long time. In this computation, on the far boundary, velocity is corrected consistently and circulation does not become zero.

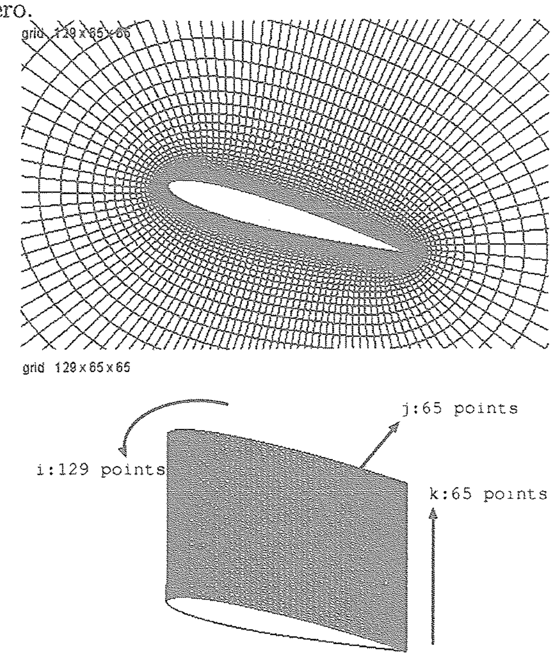
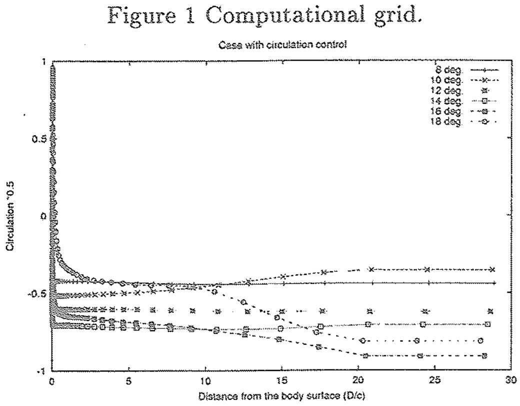
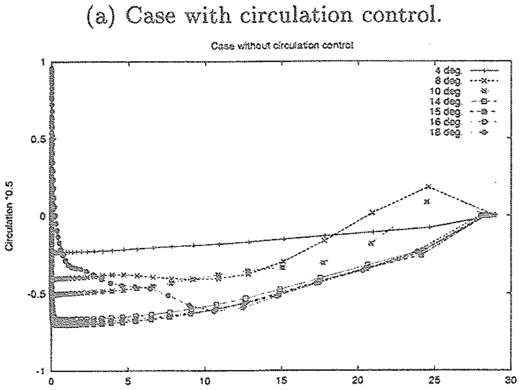


Figure 1 Computational grid.



(a) Case with circulation control.



(b) Case without circulation control.

Figure 2 Circulation control on the far boundary.

3 Results

Three-dimensional flows around NACA0012 airfoil are simulated at Reynolds number of 10^6. Fully developed two-dimensional flow is used as an initial condition for 3-d computation to save the computation time.

Computational results are visualized in figs.3-6. Figure 3 shows pressure distribution on the body surface Cp at angles of attack 14°, 16° and 18°. Section lift coefficients Cl are given in fig.4. They are computational values in case with and without circulation control on the far boundary, and experimental values[6], at angles of attack.

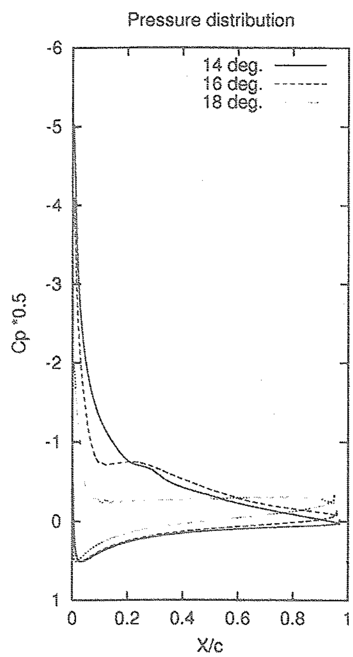


Figure 3 Pressure distribution.

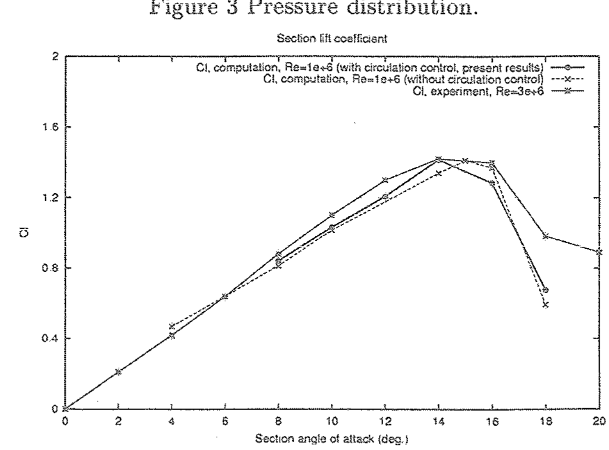
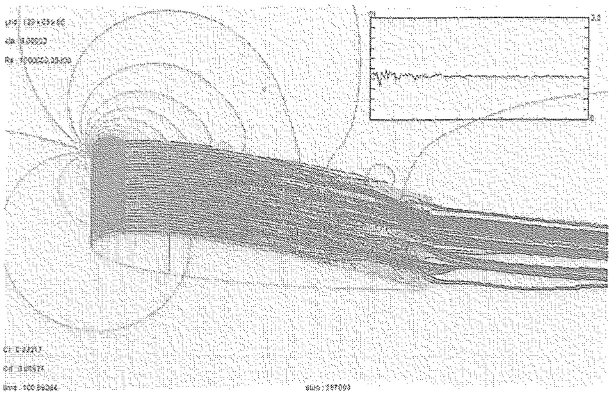
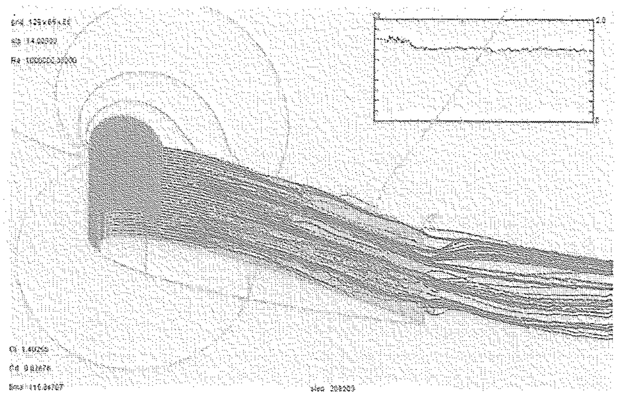


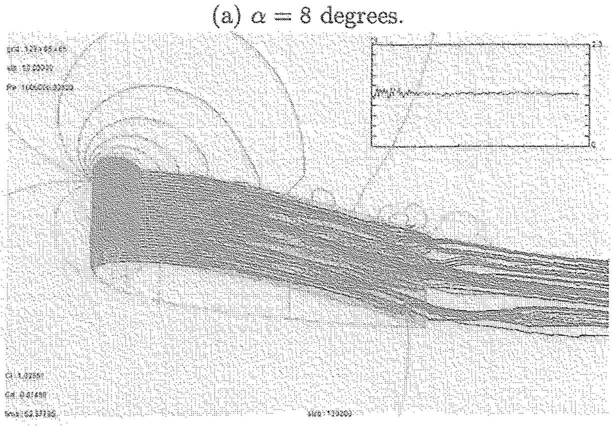
Figure 4 Section lift coefficients.



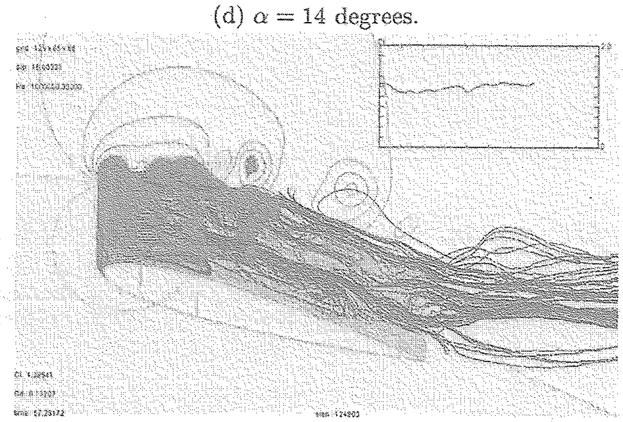
(a) $\alpha = 8$ degrees.



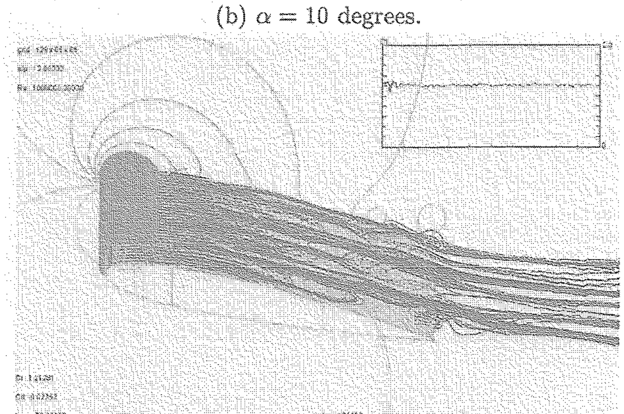
(d) $\alpha = 14$ degrees.



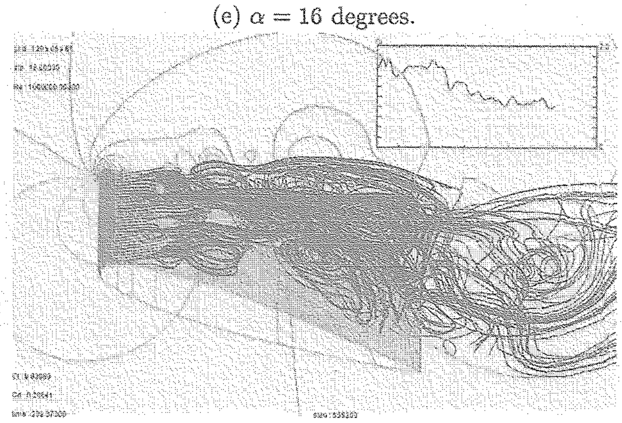
(b) $\alpha = 10$ degrees.



(e) $\alpha = 16$ degrees.

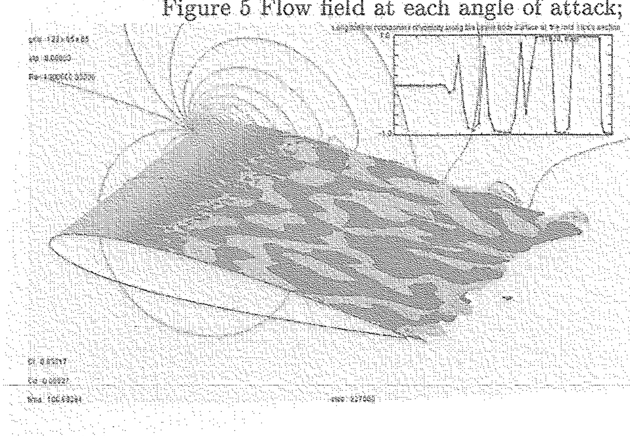


(c) $\alpha = 12$ degrees.

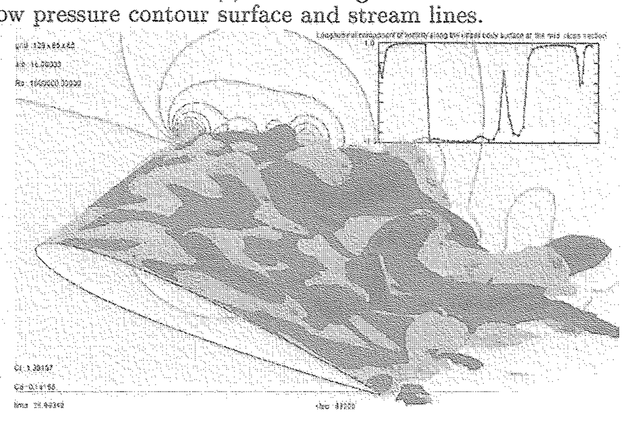


(f) $\alpha = 18$ degrees.

Figure 5 Flow field at each angle of attack; a low pressure contour surface and stream lines.



(a) $\alpha = 8$ degrees.



(b) $\alpha = 16$ degrees.

Figure 6 Longitudinal component of vorticity.

Figures 5 and 6 show instantaneous flow field at angles of attack. In fig.5, a contour surface of low pressure, pressure shading on the body surface and stream lines from the leading edge are expressed. Figure 6 gives longitudinal component of vorticity, at angles of attack 8 and 16 degrees, with using two equi-surfaces of a positive and a negative values. In fig.6, these equi-surfaces show a characteristic pattern at angle of attack 8 degrees.

4 Conclusion

In the present paper, it was shown that circulation correction on the far boundary had effect on a flow field. In other words, under a circulation-correction condition, the value of lift coefficient increased at lower angles of attack and decreased at higher angles.

References

[1] Kawamura, T. and Kuwahara, K., 1984, "Computation of high Reynolds number flow around a circular cylinder with surface roughness", AIAA Paper 84-0340.

[2] Suito, H., Ishii, K. and Kuwahara, K., 1995, "Simulation of Dynamic Stall by Multi-Directional Finite-Difference Method", AIAA paper 95-2264.

[3] Kuwahara, K., 1999, "Unsteady Flow Simulation and Its Visualization", AIAA Paper 99-3405.

[4] Kuwahara, K. and Komurasaki, S., 2001, "Direct Simulation of a Flow around a Subsonic Airfoil", AIAA Paper 2001-2545.

[5] Komurasaki, S. and Kuwahara, K., 2004, "Implicit Large Eddy Simulation of a Subsonic Flow around NACA0012 Airfoil", AIAA Paper 2004-0594.

[6] Abbott, I. H. and Von Doenhoff, A. E., 1959, "Theory of Wing Sections", Dover Pub., pp462.