SIMULATION OF SUBSONIC FLOW PAST AN AIRFOIL

Kunio Kuwahara
Institute of Space and Astronautical Science
Yoshinodai, Sagamihara, Kanagawa, Japan

Incompressible high-Reynolds-number flows are simulated by solving the Navier-Stokes equations. A finite-difference method with third-order upwinding are employed without using any turbulence model. Also multi-directional method is used. Unsolved problems in computation around an airfoil are attacked and new results are presented, which will make the possibility of computational approach widen substantially. Using the same approach we introduce a concept of an airfoil of negative thickness. Application of this concept is suggested.

INTRODUCTION

Flow around an airfoil is one of the most fundamental problems in aerodynamics. Many simulations have been done but there are some important problems still remain unsolved. Those are very unsteady impulsively started flow and a flow at very high angle of attack and also computation using O-grid. In this paper, those difficult problems are attacked and solved and new results are presented, which will make the possibility of computational approach widen substantially. Using the same approach we introduce a concept of an airfoil of negative thickness. To increase the accuracy, we have developed a new finite-difference scheme named as multi-directional finite-difference method.

COMPUTATIONAL METHOD

The governing equations are the unsteady incompressible Navier-Stokes equations. For high-Reynolds-number flows, time-dependent computations are required owing to the strong unsteadiness.

These equations are solved by a finite-difference method. The numerical procedure is based on the MAC method. The pressure field is obtained by solving the Poisson equation.

A generalized coordinates system is employed, so that enough grid points can be concentrated near the body surface where the no-slip condition is imposed.

False separation occurs if the grid is not well concentrated in the boundary layer, especially near the leading edge, where the flow velocity takes its maximum value.

All the spatial derivative terms are represented by the central difference approximation except for the convection terms. For the convection terms, the third-order upwind difference is used. This is the most important point for high-Reynolds-number computations.

There is another important problem in high-order upwind schemes. That is, the accuracy decreases when the flow direction is not well parallel to one of the coordinate lines. In generalized coordinate system, near the boundary, the flow direction and one of the coordinate lines are almost parallel if the flow is well attached, and this problem is not serious. However, if the flow separates, the flow direction becomes no more parallel to a coordinate line and the problem become very serious

To overcome this problem we introduced the multi-directional upwind method. A non-staggered mesh system is employed where the defined positions of velocity and pressure are coincident.

For the temporal integration of the Navier-Stokes equations, the Crank-Nicolson purely implicit scheme is utilized. This scheme has second-order accuracy in time. These equations and the Poisson equation are iteratively solved at each time step by the successive overrelaxation (SOR) method.

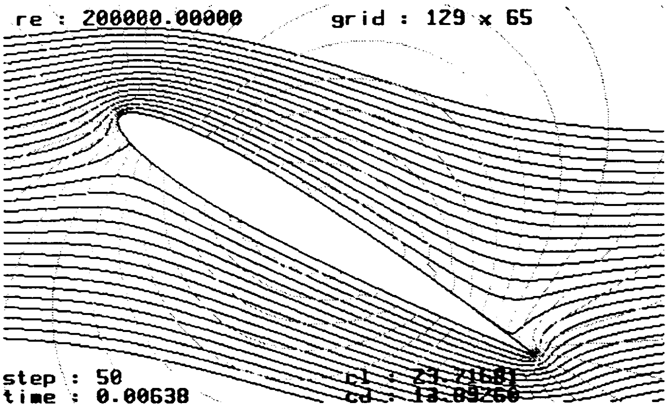
COMPUTATIONAL RESULTS

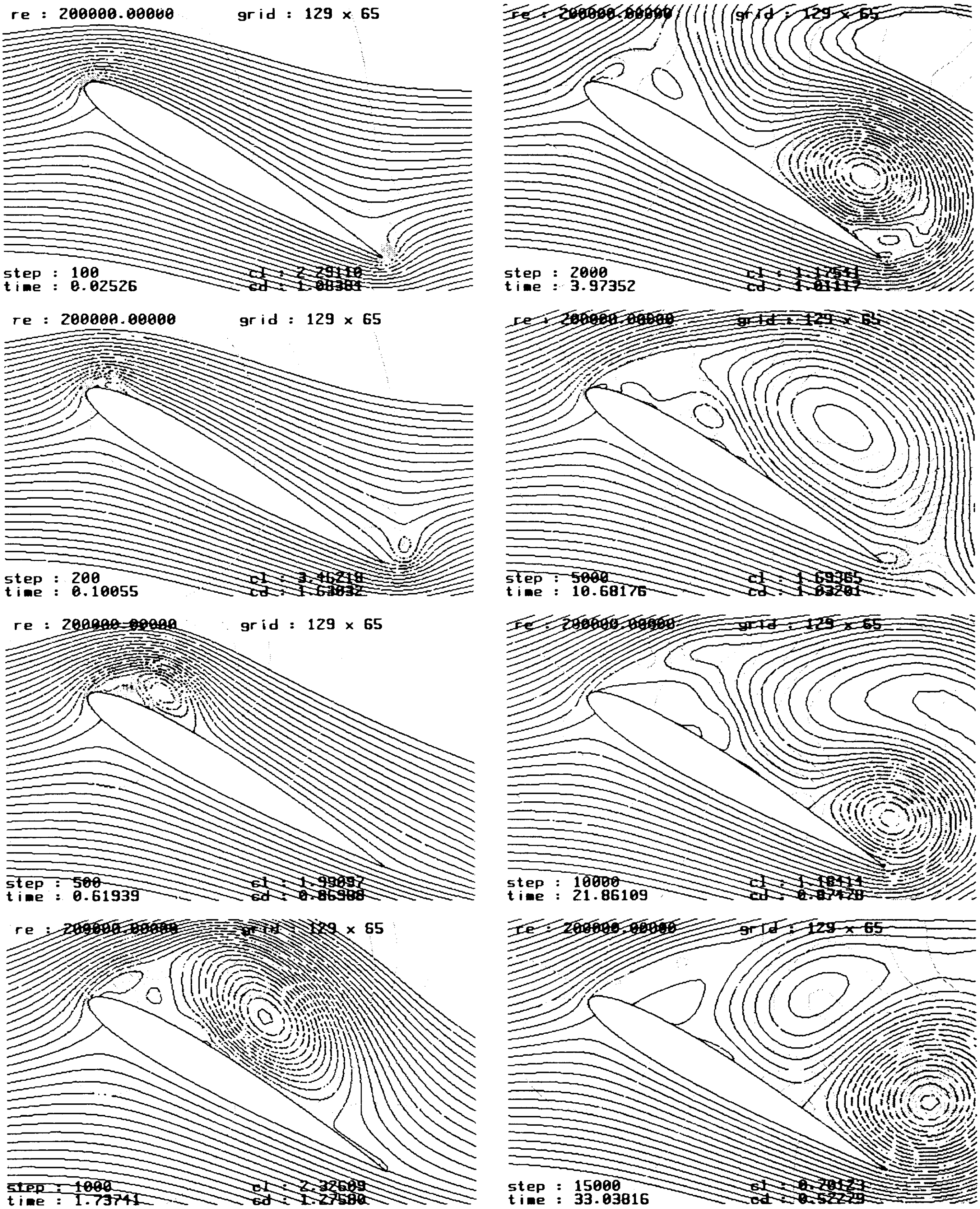
Flow around an airfoil is a standard problem, but unsteady computations have rarely been done. At high angle of attack, the flow becomes very unsteady and to understand the flow well we need an unsteady simulation.

For airfoil simulation, C-grid is usually used to avoid the trailing edge singularity. To make C-grid is not easy for very high angle of attack, and this is another reason of the difficulty to simulate the flow at very high angle of attack. Also C-grid needs unnecessarily concentrated grid points in the near wake region beginning from the trailing edge. This make the computation more unstable.

On the other hand O-grid is, in every sense, much better if the computation converges. The multi-directional finite-difference method makes the computation very stable even near the singular points.

1) The first example is a impulsively started flow around an NACA0012 at angle of attack 30 degrees. Figure 1 shows the development of the flow field. At initial stage almost potential flow is formed. Then, to cancel the singularity at the trailing edge, a small in size but strong vortex appears. Then Kutta condition is satisfied there. At the same time leading separation begins. This separation becomes larger until next trailing separation takes place. Finally Karman vortex street is formed.




re : 200000.00000
grid : 129 x 65
step : 100
time : 0.02526
cl : 2.29110
cd : 1.08381
re : 200000.00000
grid : 129 x 65
step : 200
time : 0.10055
cl : 3.46218
cd : 1.63832
re : 200000.00000
grid : 129 x 65
step : 500
time : 0.61939
cl : 1.99097
cd : 0.86988
re : 200000.00000
grid : 129 x 65
step : 1000
time : 1.73741
cl : 2.32609
cd : 1.27580
re : 200000.00000
grid : 129 x 65
step : 2000
time : 3.97352
cl : 1.17541
cd : 1.01117
re : 200000.00000
grid : 129 x 65
step : 5000
time : 10.68176
cl : 1.69365
cd : 1.03201
re : 200000.00000
grid : 129 x 65
step : 10000
time : 21.86109
cl : 1.18411
cd : 0.87428
re : 200000.00000
grid : 129 x 65
step : 15000
time : 33.03816
cl : 0.20123
cd : 0.52229

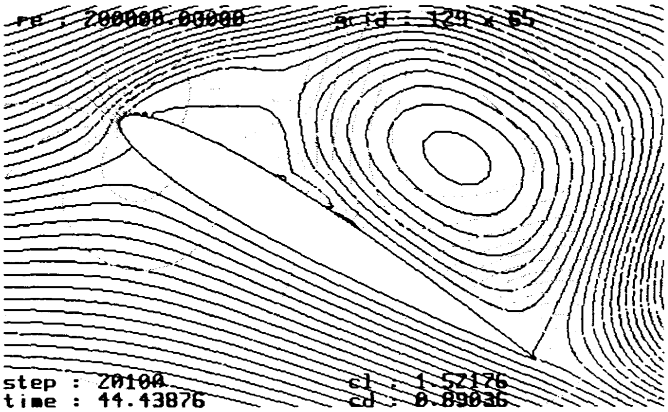


Fig.1 Flow around NACA0012 airfoil at Re=200000, 128*64 O-grid
Angle of attack 30 degrees,
Instantaneous streamlines and pressure contours

In this computation and all of the following ones, the Reynolds number is fixed as 200000. O-grid system is employed.

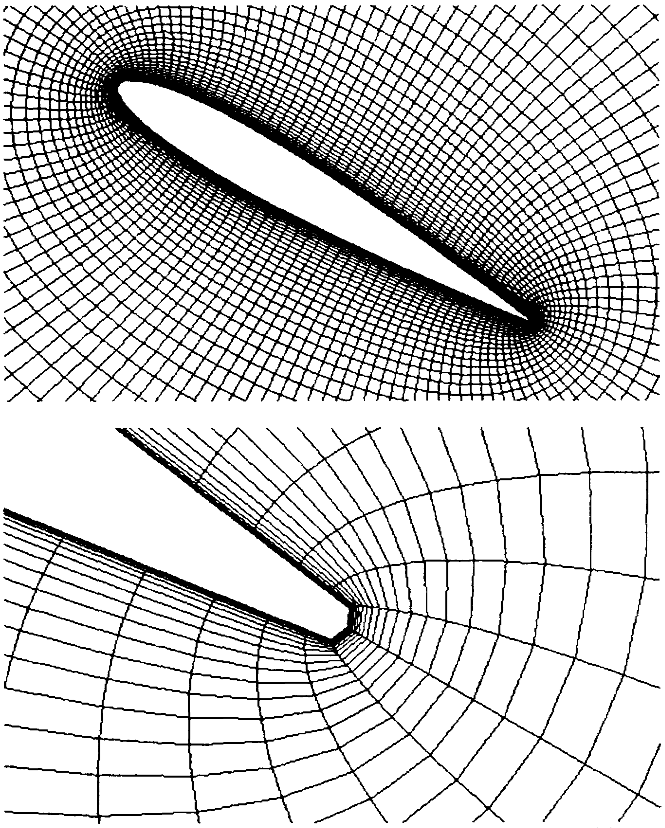


Fig.2 Grid system around NACA0012 airfoil at Re=200000, 128*64 O-grid
Angle of attack 30 degrees,

2) As an extreme case of high angle of attack computation, 90 degree attack angle case is simulated. Even in this case computation is very stable, and reasonable results are obtained.

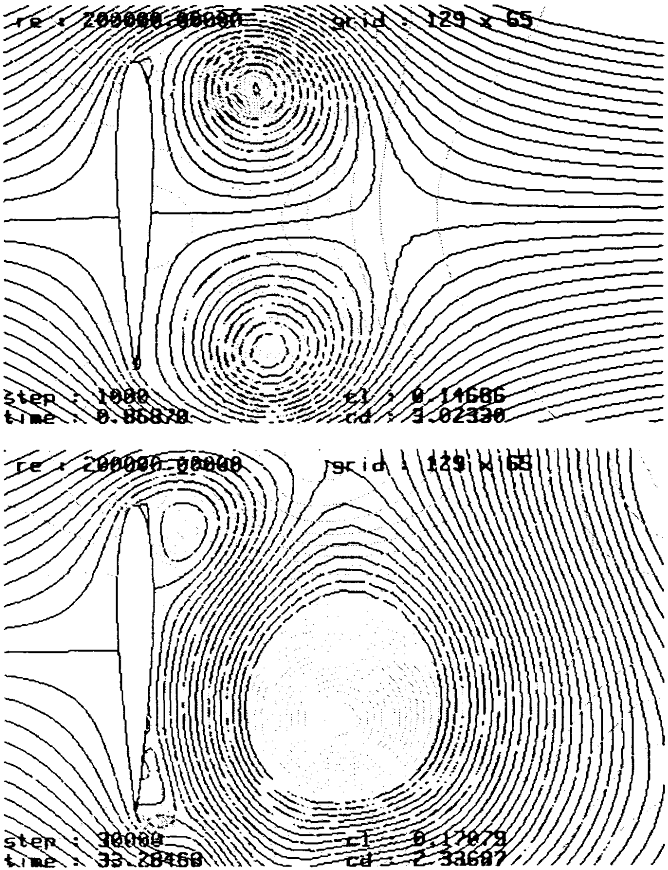
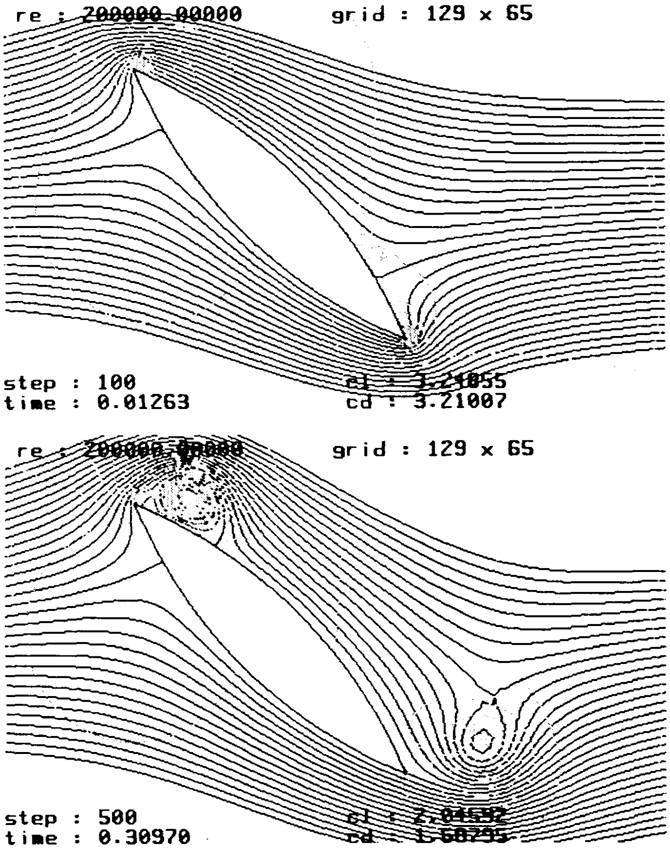


Fig.3 Flow around NACA0012 airfoil at Re=200000, 128*64 O-grid
Angle of attack 90 degrees,
Instantaneous streamlines and pressure contours

3) The above results suggest that even singular leading edge can be handled. Then impulsively started flow around a circular-arc airfoil is computed. Initially almost potential flow is formed, and flow developed as expected. The thickness is 20% and the angle of attack 45 degrees.



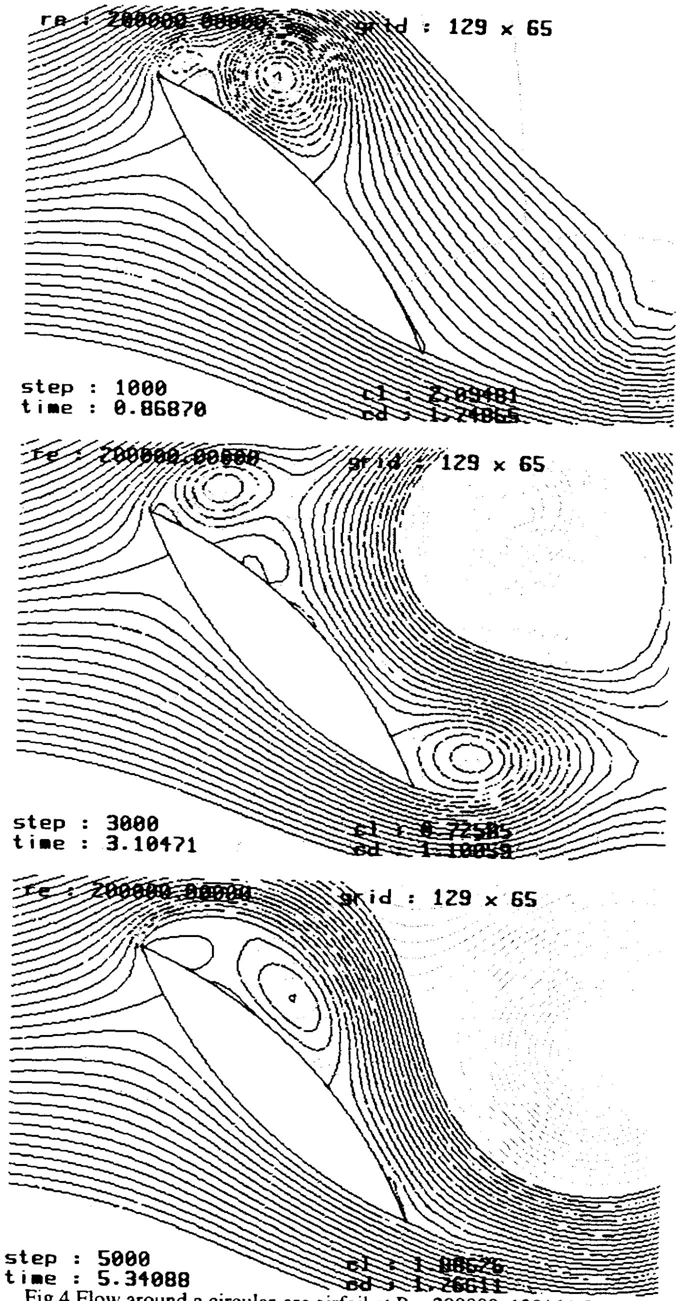
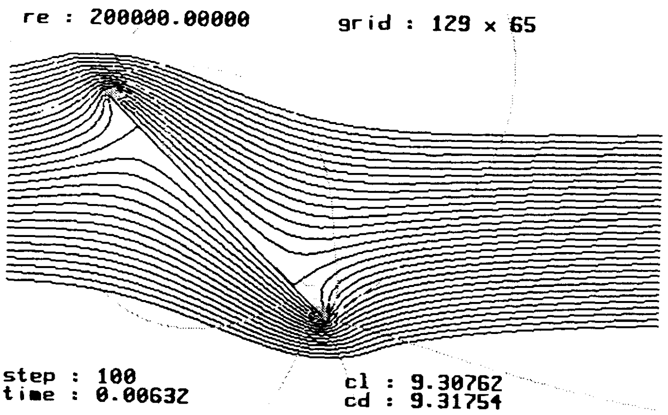


Fig.4 Flow around a circular-arc airfoil at Re=200000, 128*64 O-grid
Angle of attack 45 degrees,
Instantaneous streamlines and pressure contours

4) If circular arc airfoil can be treated, the thickness can be zero. This means that flow around a pure flat plate can be computable. The results is the following:



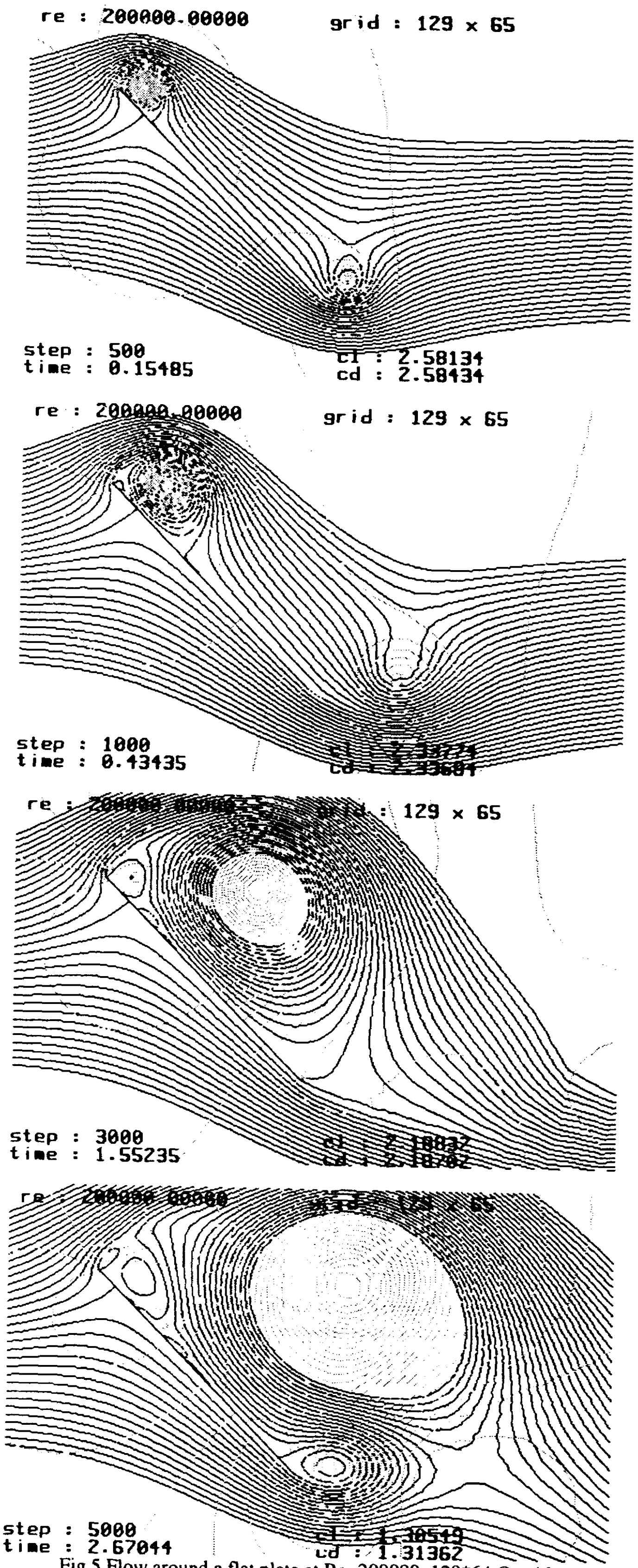


Fig.5 Flow around a flat plate at Re=200000, 128*64 O-grid
Angle of attack 45 degrees,
Instantaneous streamlines and pressure contours

5) Moreover, the thickness can be negative.

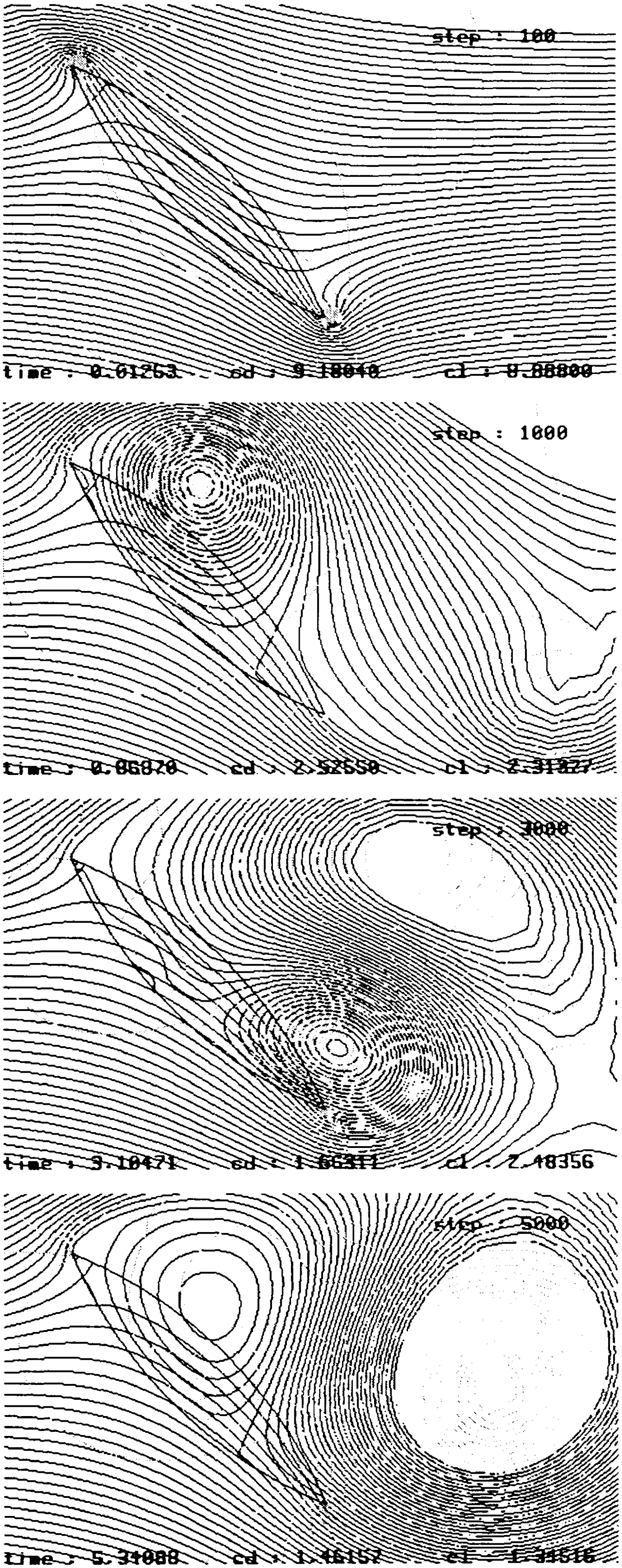


Fig.6 Flow around a circular-arc airfoil of 20% negative thickness at Re=200000, 128*64 O-grid, angle of attack 45 degrees, Instantaneous streamlines and pressure contours

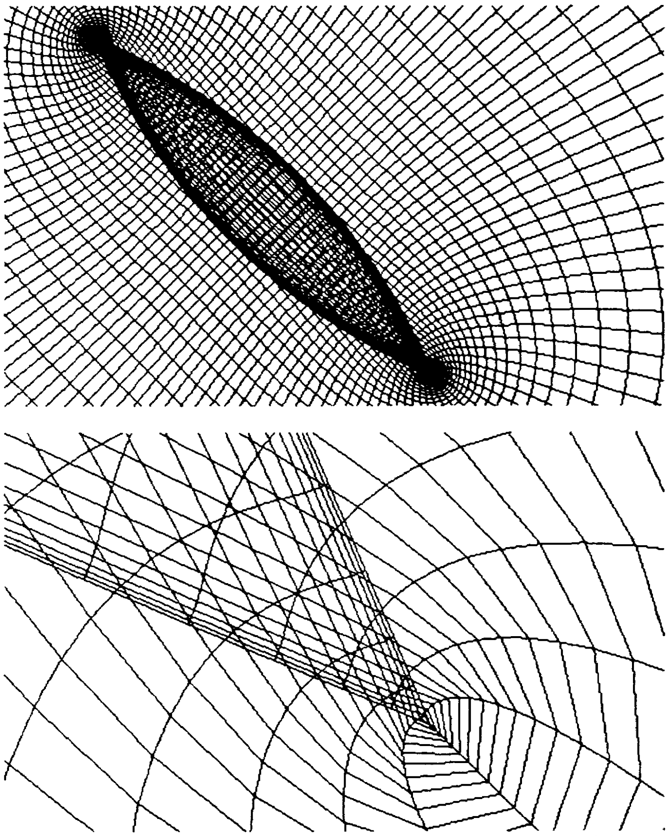
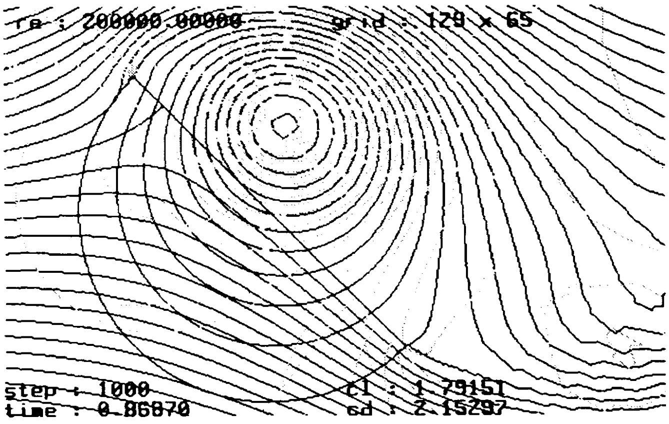


Fig.7 Grid system around a circular-arc airfoil of 20% negative thickness at Re=200000, 128*64 O-grid, angle of attack 45 degrees,

From the figures of the grid system, what negative thickness means can be easily understandable.

6) One of the interesting application of the negative thickness concept is illustrated by the following example. This is the flow past a flat plate but there is no plate in the lee side. There hemicircle is attached in the negative side, therefore oncoming flow interacts strongly with the flat plate but in the lee side flow does not interact with the plate. By pursuing these type of flow we would be able to understand the interaction of flow with an obstacle more deeply.



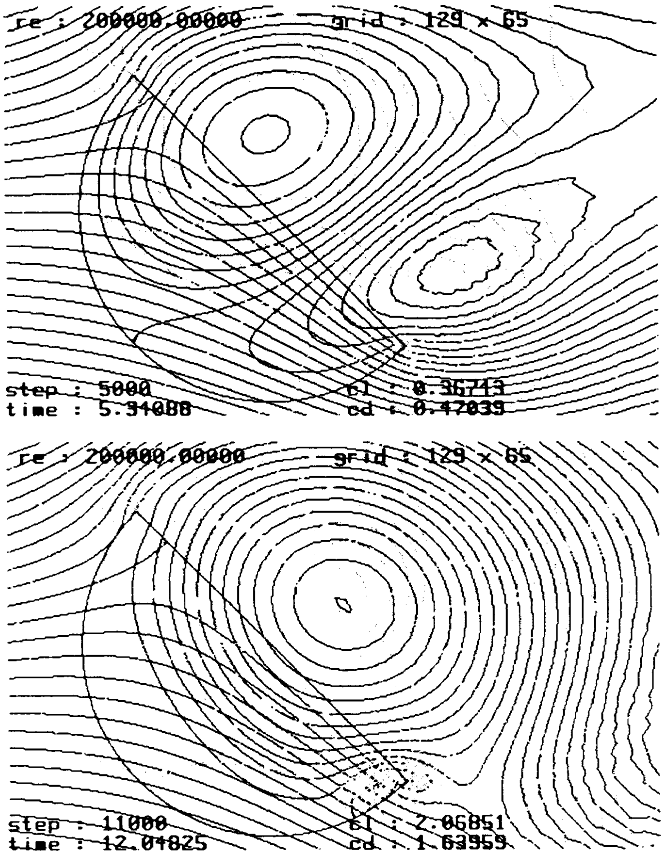


Fig.8 Flow around a flat plate with hemicircle attached at negative side at Re=200000, 128*64 O-grid, angle of attack 45 degrees, Instantaneous streamlines and pressure contours

This flow is very different from that around a simple flat plate. This suggests that even the lee side of a body plays a very important role to determine the wake structure.

CONCLUSIONS

It has been found that ,by using the present finite difference method, most of the difficulties of the incompressible flow around an airfoil has been overcome.

REFERENCES

1) K. Kuwahara: 'Computational Study of Incompressible flow by Finite-Difference Method', Proc. 14th NAL Symposium on Computational Aerodynamics, 1996.