Numerical Simulation of Boundary Layers around a Circular Cylinder

Ema Amalia*, Hirotoshi Kubota**
*Graduate School of Eng., The Univ. of Tokyo, Hongo 7-3-1, Tokyo 113-8656, JAPAN
**Dept. of Aero. and Astro, The Univ. of Tokyo, Hongo 7-3-1, Tokyo 113-8656, JAPAN

Numerical Simulation of Boundary Layers around a Circular Cylinder has been conducted. Besides the purpose of validating some instability modes of boundary layers, the highlight is given to the application of SIMPLE method for that purpose.

Key Words: *Numerical Simulation, Boundary Layers, SIMPLE method*

1. Introduction

In order to understand transition process of three-dimensional boundary layers, a numerical study of boundary layers around a yawed cylinder could be conducted[1], while theoretical study has been conducted and gives a new type of instability named "streamline curvature instability"[2].

In this research, SIMPLE method[3,4] (pressure correction method) is used to do numerical simulation around a circular cylinder which basically the model of a yawed circular cylinder. First, steady condition with velocity that the same with 1 and 2 has been evaluated and then unsteady condition also has been evaluated. Disturbance still not been applied.

2. Methods and Results

The equation used is the incompressible Navier-Stokes equation in generalized coordinate for steady condition and in cylindrical coordinates for unsteady condition. Those equations include momentum equation and continuity equations. The flow is subsonic region.

2.1 Steady Case

For spatial term, two order central difference was used. Free stream velocity used is 30 m/s with Reynolds number 220.000, while grid size used was 200x 100x60 for a half cylinder, The first 25 grids are in about 5 x boundary layers characteristics length. Boundary conditions that used is mixed boundary condition. TDMA was used for solving finite difference equations.

Results of velocity field near surface of circular cylinder could be seen in fig. 1. It could be said that the typical boundary layers could be gotten by SIMPLE method. Fig. 2 gives typical profile of boundary layers near surface of circular cylinder[5].

It is said that people should be careful about numerical diffusion that usually occurred when using SIMPLE method. To deal with this, it is better to use fourth order central difference scheme in treating diffusive term. And also third upwind scheme is also better to used in convective term, especially for high Reynolds number[6].

Although the case evaluated is called "steady case", actually it was used unsteady Navier-Stokes equation, but the time steps that used were just transient steps to get a steady condition. Because the final objective is to get simulations of instability modes, then it is better to use unsteady equations with "really unsteady treatment". Moreover, in order to get understanding of applicability of SIMPLE method to simulate instabilities in boundary layers by using higher order differential scheme, the cylindrical coordinate has been used (paragraph 2.2) to avoid the influence of using the metrics.

2.2 Unsteady Case

Navier-Stokes equation in cylindrical coordinate form was used. Firstly, second order difference scheme was applied. By using this scheme, there is no convergence results could be gotten as could be seen in fig. 3.

Secondly, third order upwind scheme for convective term and fourth order central difference scheme for diffusive term were applied. The convergence results could be gotten as shown in fig. 4 but the velocity distribution near the surface still not give physically correct results.

For both cases, SIMPLE method was basically used, and SOR method was used for evaluating the difference equations.

3. Concluding remarks

For steady condition by using generalized coordinate the method could catch the typical profile of boundary layers, while for unsteady condition by using circular coordinate the method could give convergence results although still not give physically correct results. It is well known that the critical point of using SIMPLE method is whether the iteration could give any physically correct results[4,6]. Proper initial and boundary condition treatments are unique for each cases[4,6]. It means that SIMPLE method could be used for simulation of boundary layers around a circular cylinder although for using higher order difference scheme proper treatments of initial and boundary conditions should be evaluated further. And also, three-dimensional view of velocity field of cylinder would mean after applying disturbance which will be further studies.

References

1) Yokokawa, Y. and Fukunishi, Y., et.al., 1999, *Numerical Study of Excitation of Two Different Instabilities in Three-Dimensional Boundary Layer on a Yawed Cylinder*, IUTAM proceeding (in process of publication by Springer-Verlag).
2) Itoh, N., 1994, Fluid Dyn. Res. 14 pp. 353-366.
3) Patankar, SV, 1980, *Numerical Heat Transfer and Fluid Flow*, Hemisphere Publish. Corp.
4) Arakawa, C., 1994, *Computational Fluid Dynamics for Engineering* (in Japanese), University of Tokyo Press.
5) Floryan, JM, 1991, Prog. Aerospace Sci. Vol. 28, pp. 235-271.
6) Anderson, JD, 1995, *Computational Fluid Dynamics: The Basics with Applications*, McGraw-Hill.

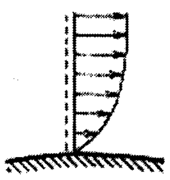

Fig. 2 Typical Boundary Layers Velocity Distribution

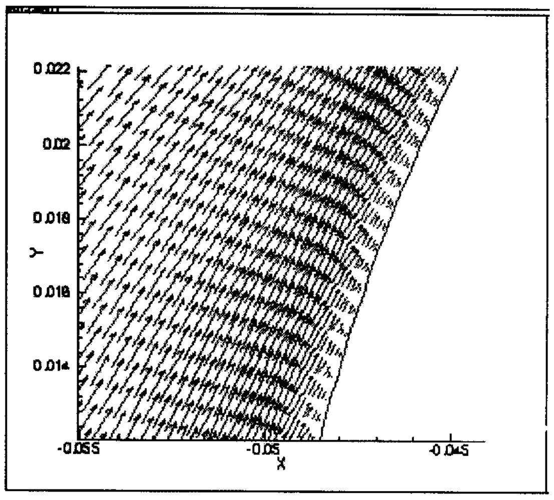

Fig. 1 Velocity field near surface of circular cylinder

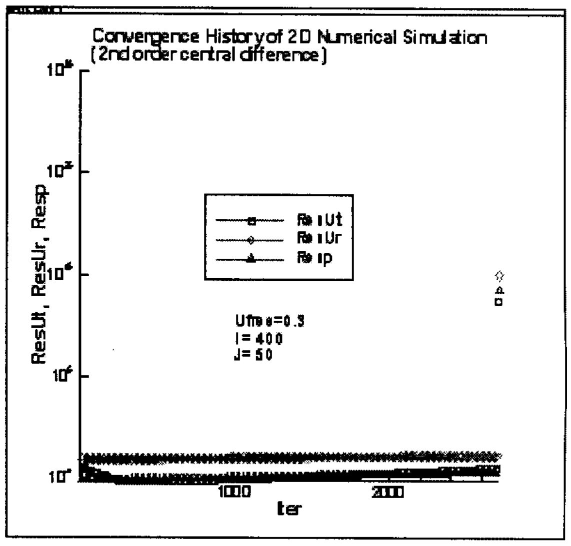

Fig. 3 Convergence History by using second order central Difference

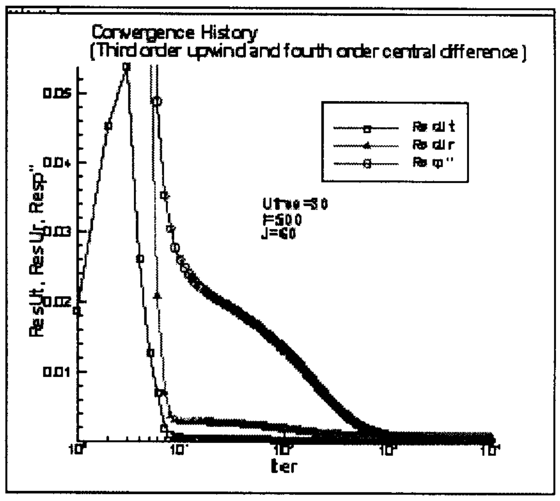

Fig. 4 Convergence History by using Third upwind scheme and Fourth Order Central Difference Scheme