NUMERICAL SIMULATION OF VORTICAL FLOWS USING A HIGHLY ACCURATE FINITE DIFFERENCE SCHEME

Katsuya Ishii*, Tomonori Nihei**, Shizuko Adachi**

*ITC, Nagoya University, Nagoya, 464-8601, Japan

**Dept. of CSE, Nagoya University, Nagoya 464-8603, Japan

***RIKEN, Wako, 351-0198, Japan

Summary A combined compact finite difference (CCD) scheme with high accuracy and high resolution is proposed for numerical simulation of incompressible vortical flow fields. The CCD scheme has eighth-order accuracy and spectral-like resolution for the first derivative except for the boundary. The Poisson equation is also solved accurately by using the CCD scheme and the ADI method. The vortical flows in the three-dimensional lid-driven square cavities with different spanwise aspect ratios are studied using this new scheme.

INTRODUCTION

Numerical simulations of vortical flows need high accuracy and high resolution, since vortices with different spatial scales are found in the flow. The combined compact finite difference (CCD) scheme proposed by the authors has high accuracy with comparatively low computational cost and it is a promising scheme for large-scale computation with high accuracy[1]. A class of combined compact difference schemes was proposed by Chu and Fan [2]. In the schemes, the derivatives of a function are evaluated by means of the values and derivatives at the two adjacent points. Our CCD scheme has eighth-order accuracy and spectral-like resolution for the first order derivative in the problem with periodic boundary conditions. The order of computational cost for the CCD scheme is the same as that of the ordinary difference scheme.

In this paper, the incompressible vortical flow field in a three-dimensional lid-driven cavity is considered. We modify the CCD scheme for the incompressible flow fields. For solving the Poisson equation, a new iterative method using the ADI method and the CCD formulation is proposed, and it gives highly accurate numerical solutions of the 3D Poisson equation. We also discuss the CCD scheme with Dirichlet and/or Neumann boundary conditions and the parallelization of the scheme.

The flow in the lid-driven cavity has been extensively studied as a fundamental model for separated flows and for the vortex dynamics in the closed systems with simple geometry and boundary conditions. There are two aspect ratios in the three dimensional flows in finite domains : $\Gamma = d/h$ and $\Lambda = l/h$ where d, h and l are the width, height and span length of cavity respectively . We consider the square cavity ($\Gamma = 1$) with different spanwise aspect ratio Λ's and study the side boundary effect for the overall flow structure and the Taylor-Görtler-like vortex pair in the secondary flow.

NUMERICAL SCHEME

Consider a function $f(x)$ defined on the interval where N grid points are located with a uniform spacing h. Let f_i, f'_i, f''_i and f'''_i be the values of the function and its first, second and third derivatives at i-th grid point x_i respectively. The CCD scheme evaluates these derivatives implicitly from the following relations:

$$f'_i = a_1(f_{i+1} - f_{i-1}) + a_2(f'_{i+1} + f'_{i-1}) + a_3(f''_{i+1} - f''_{i-1}) + a_4(f'''_{i+1} + f'''_{i-1})$$

$$f''_i = b_1(f_{i+1} + f_{i-1} - 2f_i) + b_2(f'_{i+1} - f'_{i-1}) + b_3(f''_{i+1} + f''_{i-1}) + b_4(f'''_{i+1} - f'''_{i-1})$$

$$f'''_i = c_1(f_{i+1} - f_{i-1}) + c_2(f'_{i+1} + f'_{i-1}) + c_3(f''_{i+1} - f''_{i-1}) + c_4(f'''_{i+1} + f'''_{i-1})$$

The formal accuracy of the scheme is determined by the parameters a_j, b_j and c_j. By relaxing the requirement for the highest formal accuracy, we have free parameters. The values of the free parameters are chosen to minimize errors in the wavenumber space for most waves except very short waves [1,3].

The Poisson equation usually needs to be solved in incompressible fluid flow problems, and its finite difference solver generally takes a large portion of the entire computation time. Since CCD schemes implicitly evaluate the derivatives, a method such as SOR, which is often used, is hard to apply to the Poisson equation solver that uses a CCD scheme. Chu and Fan [2] proposed an algorithm for a CCD scheme which solves the 2D elliptic partial differential equation by the ADI method using the modified Poisson equation and the CCD derivatives. However, their method is not effective for the accurate calculation with many grid points. We developed a new ADI method using the original Poisson equation. This algorithm can be parallelized easily.

The incompressible Navier-Stokes equations in conservation form are numerically solved by the well-known MAC method and the fourth order Runge-Kutta method for the time integration. The derivatives in the equations are evaluated by the CCD scheme, in which the first and the second derivatives have eighth-order accuracy except for the boundary and the third derivative has fourth-order accuracy. At the boundary, the first and the second derivatives have sixth-order accuracy. In the case of the cubic cavity(Λ=1) at Re=100 to 400, the numerical results using this new scheme with 33x33x33 uniform grid points are in good agreement with the previous results[4] with 81x81x81 non-uniform grid points. For the study of the wall effect, the 100x100x 100Λ uniform grid system is used in the calculation, in which Λ is a spanwise aspect ratio.

RESULTS

We focus on the flows in a square driven-cavity with spanwise aspect ratios Λ from 1 to 6.55 at the Reynolds number $Re=Ud/\nu$=850, where U is the speed of the sliding upper wall and ν is the kinematic viscosity. In this Reynolds number, we got steady state solutions. We show the streamline using the projected velocity on the vertical plane near the upstream end wall at Λ=4 and Λ=6.55 in Fig.1 and Fig. 2 respectively. While Fig. 1 shows one symmetric flow structure and the upper corner vortex, Fig.2 shows 2.5 cell structures in the central region and the other cell structure near the side wall. These results agree with the experimental result [5]. These patterns are strongly related to the Taylor-Görtler-like vortex pairs in the secondary flow of the overall flow structure. We can discuss the bifurcation of flows in the lid-driven cavity with different spanwise aspect ratios Λ.

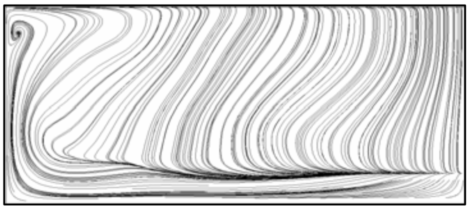

Fig.1 streamlines using the projected velocity near the upstream end wall (left half plane, Λ =4)

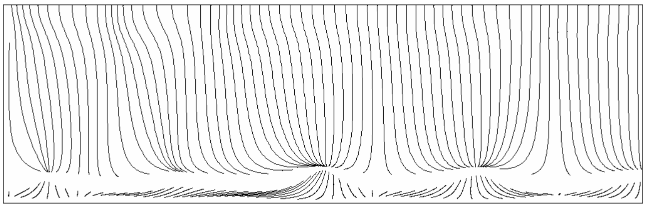

Fig.2 streamlines using the projected velocity near the upstream end wall (left half plane, Λ=6.55)

CONLUDING REMARKS

We have developed the highly accurate combined compact finite difference scheme for the incompressible flow. The application to the flow shows the efficiency of the new scheme. The numerical results clearly show the bifurcation of flow structures in a lid-driven cavity with different spanwise aspect ratios

Reference

[1] Nihei ,T. and Ishii, K:A Fast Solver of the Shallow Water Equations on a Sphere using a Combined Compact Difference Scheme.*J. Comput. Phys.* **187**, 639—659,2003

[2] Chu, P.C. and Fan, C.: A Three-point Combined Compact Difference Scheme, *J. Comput. Phys.* **140**, 370-399,1998.

[3] Lele, S.K.:Compact Finite Difference Schemes with Spectral-like Resolution. *J. Comput. Phys.* **103**, 1992, pp. 16—42, 1992.

[4] Iwatsu, R., Ishii, K., Kawamura, K. Kuwahara, K. Hyun, J. M: Numerical Simulation of Three-dimensional Flow Structure in a Driven Cavity, *Fluid Dyn Res* **5**,173-189, 1989..

[5] Albensoeder, S., Kuhlmann, H. C., and Rath, H.J. : Three-dimensional Centrifugal-flow Instabilities in the Lid-driven-cavity Problem. *Phys. of Fluid* .**13,** 121-135,2001.