GPU-ACCELERATED HIGH-ORDER SOLVER FOR SIMULATING 3D INCOMPRESSIBLE NAVIER-STOKES EQUATIONS IN COMPLEX DOMAIN

Neo Shih-Chao Kao*, **Tony Wen-Hann Sheu**†

* National Taiwan University (NTU)
Department of Engineering Science and Ocean Engineering (ESOE)
10617 Taipei, TAIWAN
e-mail: d97525011@ntu.edu.tw

†National Taiwan University (NTU)
Department of Engineering Science and Ocean Engineering (ESOE)
Center for Advanced Study in Theoretical Science (CASTS)
10617 Taipei, TAIWAN
e-mail: twhsheu@ntu.edu.tw

Key words: Incompressible; CCD; Immersed boundary; CUDA; GPU

Abstract. In this study, a flow solver developed within the framework of direct-forcing immersed boundary (IB) method is applied to solve three-dimensional flow problem in complex domain. The incompressible Navier-Stokes equations are fractionally split into the steps for inviscid Euler, Helmholtz and projection equations which are coupled to each other. The high-order upwinding combined compact difference (UCCD) scheme developed in a three-point grid stencil is used to approximate the first-order spatial derivative terms shown in the momentum equations. The sixth-order symplectic Runge-Kutta (SRK6) time integrator is employed to approximate the temporal derivative term in the inviscid Euler equation so as to numerically retain its embedded Hamiltions and Casimir to get a long-time accurate solution. The artificial IB forcing terms are added to the Helmholtz and projection equations, respectively. The introduced forcing terms are solved iteratively so that the interpolation procedure normally required near the immersed boundary can be eliminated. The main feature of the proposed IB method is that the resulting velocity satisfies the immersed boundary condition and the divergence-free constraint condition simultaneously. For the sake of computational efficiency in solving three-dimensional problems, all the calculations will be performed on a hybrid CPU/GPU platform to accelerate the calculations. The numerical results with good agreement with the available reference solutions and experimental data have been demonstrated. Moreover, the good speedup performance indicates that the developed flow solver is effective and reliable tool for investigating fluid-structure interaction problems

1 INTRODUCTION

The idea of combined compact difference (CCD) scheme, which was original proposed by Chu [1], is employed to approximate the first- and second-order derivatives terms simultaneously, This scheme is more compact and accurate than the classical difference scheme when solving the problems under the same number of stencil points. In this study, the improved CCD (UCCD) scheme developed in a three-point stencil is proposed to solve the flow problem [2]. This scheme not only provides high accuracy but also can suppress velocity oscillations in a convection-dominated flow case.

The IB method was firstly developed by Peskin for simulating the blood flow through the elastic heart valves [3] and has been extensively applied to a wide variety of fluid-structure interaction (FSI) problems. The distinguished feature of the IB method is that the entire calculation can be carried out in fixed Cartesian grids and the well-developed Cartesian grid flow solver can be directly applied. The mesh problem regarding regeneration for moving or deformable boundary problem can be fully eliminated. Therefore, simulation of FSI problems in fixed Cartesian grids becomes possible.

2 METHODOLOGY

In this study, the first-order spatial derivative term is approximated within the following three-point compact framework :

$$a_1 \frac{\partial \phi}{\partial x}\Big|_{i-1} + \frac{\partial \phi}{\partial x}\Big|_{i} = \frac{1}{h}(c_1\phi_{i-1} + c_2\phi_i + c_3\phi_{i+1}) - h\big(b_1 \frac{\partial^2 \phi}{\partial x^2}\Big|_{i-1} + b_2 \frac{\partial^2 \phi}{\partial x^2}\Big|_{i} + b_3 \frac{\partial^2 \phi}{\partial x^2}\Big|_{i+1}\big), \tag{1}$$

$$-\frac{9}{8h}\big(\frac{\partial \phi}{\partial x}\Big|_{i-1} - \frac{\partial \phi}{\partial x}\Big|_{i+1}\big) - \frac{1}{8}\big(\frac{\partial^2 \phi}{\partial x^2}\Big|_{i-1} + \frac{\partial^2 \phi}{\partial x^2}\Big|_{i+1}\big) + \frac{\partial^2 \phi}{\partial x^2}\Big|_{i} = \frac{3}{h^2}\big(\phi_{i-1} - 2\phi_i + \phi_{i+1}\big). \tag{2}$$

The introduced unknown coefficients shown in Eq. (1) can be determined by employing the theory of Taylor series expansion and Fourier transform analysis. The fundamental analysis shows that the UCCD scheme is better than the original CCD scheme.

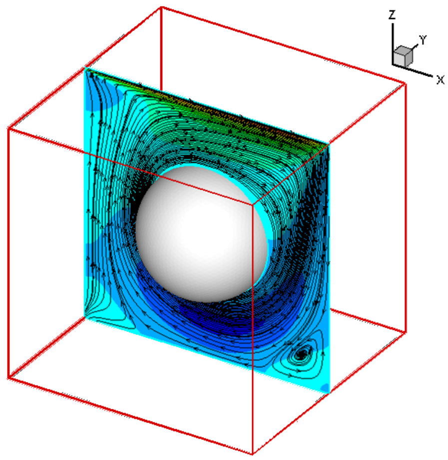
The novel fractional-step algorithm was adopted to solve the fluid flow separately and the symplectic Runge-Kutta time integrator is used to preserve the existing Hamiltonian and Casimir in the Euler equation [4]. The Helmholtz and pressure Poisson equations will be solved by using the fourth-order accurate compact scheme [5].

3 NUMERICAL RESULTS

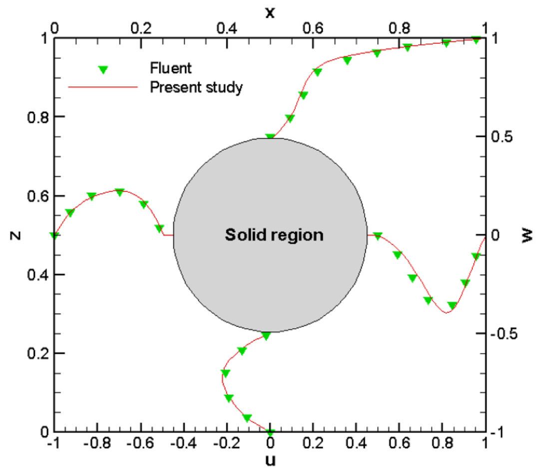
Three-dimensional lid-driven cavity flow, within which there is a sphere with radius 0.2 schematic in Figure 1(a), is solved to validate the developed code. The predicted velocity profiles $u(0.5, 0.5, z)$ and $w(x, 0.5, 0.5)$ show good agreement with the referenced solutions obtained by the commercially available Ansys-Fluent in Figures 1(b).

4 CONCLUSION

In this study, we will provide more numerical results and speedup performance to demonstrate that the developed three-dimensional parallelized code for simulating the three-dimensional incompressible Navier-Stokes equations in complex domain is effective and credible to use and is applicable to investigate the practical flow problems.



(a)



(b)

Figure 1: Schematic and comparison of the predicted velocity profiles of the three-dimensional lid-driven cavity flow problem. (a) Problem schematic; (b) Velocity profiles at Re = 1,000.

REFERENCES

[1] P. C. Chu, C. Fan. A three-points combined compact difference scheme. *J Comp. Phys.* (1998) **140**:370-399.

[2] P. H. Chiu, Tony W. H. Sheu. On the development of a dispersion-relation-preserving dual-compact upwind scheme for convection–diffusion equation. *J Comp. Phys.* (2009) **228**:3640-3655.

[3] C. S. Peskin. Flow patters around heart valves: a numerical method. *J. Comp. Phys.* (1972) **10** 252–271.

[4] Tony W. H. Sheu, C. H. Yu. Development of an incompressible Navier-Stokes solver involving symplectic and nonsymplectic time integrators. *Numer Heat Tranf. B-Fundam.* (2009) **58**:262-286.

[5] A. J. Harfash and H. A. Jalob. Sixth and fourth order compact finite difference scheme for two- and three-dimensional Poisson equations with two methods to derive these schemes, *Basrah J. Sci (A)* (2006). **24(2)**:1–20.