Direct Numerical Simulations of wall bounded turbulent flows using general pressure equation

Xiaolei Shi[a], Tzu-Hsuan Chiu[a], Chao-An Lin[*,a]

[a] *Department of Power Mechanical Engineering, National Tsing Hua University, Hsinchu 30013, TAIWAN*

The simulations of incompressible flows (INS) encounter the decoupling between pressure and velocity. To advance the pressure field, numerical schemes which involve solving an additional equation such as the Poisson equation for the pressure correction might be used. Solving the elliptic linear algebra system introduced by pressure Poisson equation is usually the most severe obstacle for both reducing the computational cost or taking advantage of highly parallelism hardware like graphics processing unit (GPU) due to the limitation of parallel efficiency or memory bounded operations.

Instead of solving the pressure Poisson equation, a fully explicit approach with locality stencils for the simulation of incompressible flow is desired. Chorin [1] proposed the artificial compressibility method (ACM), in which artificial compressibility was introduced into the continuity equation as shown in eq.1a,

$$\frac{\partial P}{\partial t} + \frac{1}{\delta}\nabla \cdot \mathbf{u} = 0 \tag{1a}$$

$$\frac{\partial P}{\partial t} + \frac{1}{\mathrm{Ma}^2}\nabla \cdot \mathbf{u} = \frac{\gamma}{\mathrm{RePr}}\nabla^2 P \tag{1b}$$

where δ is the inverse of the square of the artificial speed of sound C_s, i.e. $C_s^2 = 1/\delta$. The AC method was used to obtain steady incompressible flows, so the intermediate results before the final steady-state solutions using large time-step does not satisfy the divergence-free condition. Recently, Toutant [2] derived the general pressure equation (GPE) based on the isothermal limit and low Mach number assumption, as shown in eq.1b. Despite the successes of many low Reynolds number flow simulations using GPE [3], the capability of the GPE method to compute turbulent flows, which to the author's knowledge has not been explored. Therefore, the GPE based method is used to perform the direct numerical simulation (DNS) of turbulent lid-driven cavity (LDC) flow of $\mathrm{Re} = 3200$ and fully developed turbulent flow through a square duct of $\mathrm{Re}_\tau = 360$.

*Corresponding author.
Email address: `calin@pme.nthu.edu.tw` (Chao-An Lin)

The governing equations for the simulation of incompressible flow by general pressure equation [3] and the momentum equation are given by

$$\frac{\partial P}{\partial t} + \frac{1}{\mathrm{Ma}^2}\nabla\cdot\mathbf{u} = \frac{\gamma}{\mathrm{RePr}}\nabla^2 P \tag{2}$$

$$\frac{\partial \mathbf{u}}{\partial t} + \nabla\cdot(\mathbf{uu}) = -\nabla P + \frac{1}{\mathrm{Re}}\nabla^2\mathbf{u} + F_i \tag{3}$$

where P is the kinematic pressure, $\mathbf{u}$ is the velocity and F_i represents an external forcing term, $\gamma, \mathrm{Re}, \mathrm{Pr}$ are the heat capacity ratio, Reynolds number and Prandtl number respectively. The Mach number is defined as $\mathrm{Ma} = u_c/C_s$, where C_s is the artificial sound speed and u_c is the characteristic velocity in the flow field. γ/Pr is regarded as a numerical parameter. Eqs.2 and 3 converge to the INS equations with an error of $O(\mathrm{Ma})$. The governing equations (eq.2-3) are discretized on a non-uniform staggered grid where pressure is stored at the center of control volume and velocities are stored at the interface by using finite-volume approach. Spatial derivatives for velocities and pressure are approximated by second-order central difference scheme. The temporal integration is performed using the 3rd order TVD Runge-Kutta scheme [4].

In the present work, a GPU implementation is built on a hybrid of message passing interface (MPI) and CUDA based on the C++ programming language, in which CPUs are only responsible for the program flow control and file IO. The flow variables are organized by structure of arrays (SoA) format in the GPU memory.

In the present work, the turbulent flow within a cubic cavity of Reynolds number $\mathrm{Re} = u_{lid}L/\nu = 3200$, based on lid velocity and cavity height ($L = 1$), is explored. The grid density adopted is 192^3, which is symmetry clustered towards the wall in the x, y, and z-directions. The distributions of mean velocity and the Reynolds stress along the wall bisector $(x, 0.5, 0.5)$ and $(0.5, 0.5, z)$ at the symmetry plane are presented in Fig. 1. Here the experimental data of Prasad et al. [5] and results of incompressible Navier-Stokes (INS) solver are included for comparison. The results calculated by the GPE method agree well with both the experiments and INS simulations.

Turbulent flows along a square duct are characterized by the existence of mean secondary flows of the Prandtl's second kind which appreciably alter the transfer of momentum and scalar quantities near the walls albeit with its relevantly small magnitude (only 1% to 3% of the streamwise bulk velocity). In the present work, the Reynolds number has been kept being 360 based on the friction velocity u_τ and the height of the duct H, corresponding to a Reynolds number of 5180 based on bulk velocity u_b. A corresponding grid number is $384 \times 192 \times 192$ where a nonuniform grid is used.

The contours of the mean streamwise velocity superimpose on the vector field is shown in Fig. 2(a). The bulging of the streamwise velocity contours, induced by the secondary velocities, towards the corners is evident. For comparison, an instantaneous flow field at $x = \pi$ plane is shown Fig, 2(b).The mean streamwise velocity distribution normalized with the local friction velocity along the lower wall bisector is given in Fig. **??** and compared with the DNS data of channel flow of $\mathrm{Re}_\tau = 180$. The present result is in good agreement with the simulation of Moser et al. [6]. Distributions of the mean streamwise velocity and V component of the secondary velocity normalized by the maximum mean streamwise velocity are shown in Fig. 3. Good correspondence with results of Gavrilakis [7] is obtained, indicating the accurate predictions of the secondary flows.

1. Conclusion

In the present study, the velocity and pressure coupling is achieved by adopting the general pressure equation proposed by Toutant [2]. The method is fully explicit, and the method does not require either solving the pressure Poisson equation nor executing sub-iteration for incompressible flow simulation. Here, the general pressure equation-based method is used to perform the direct numerical simulation of turbulent lid-driven cavity flow at $\mathrm{Re} = 3200$ and fully developed turbulent flow through a square duct at $\mathrm{Re}_\tau = 360$. Predicted turbulence statistics are contrasted with existing numerical and experimental data, providing an excellent quantitative agreement. The intricate flow patterns such as the Taylor-Görtler-Like vortices in LDC flow and the mean secondary flow at the cross-section in the square duct are captured, showing both qualitative and quantitative agreements with measurements. Results from the present study indicate the capability of the GPE method for accurate incompressible turbulent flow calculation.

References

[1] A.J. Chorin, Bull. Amer. Math. Soc. **73**(6), 928 (1967)

[2] A. Toutant, Physics Letters A **381**(44), 3739 (2017)

[3] A. Toutant, Journal of Computational Physics **374**, 822 (2018)

[4] J. Williamson, Journal of Computational Physics **35**(1), 48 (1980)

[5] A.K. Prasad, J.R. Koseff, Physics of Fluids A: Fluid Dynamics **1**(2), 208 (1989)

[6] R.D. Moser, J. Kim, N.N. Mansour, Physics of Fluids **11**, 943 (1999)

[7] S. Gavrilakis, Journal of Fluid Mechanics **244**, 101 (1992)

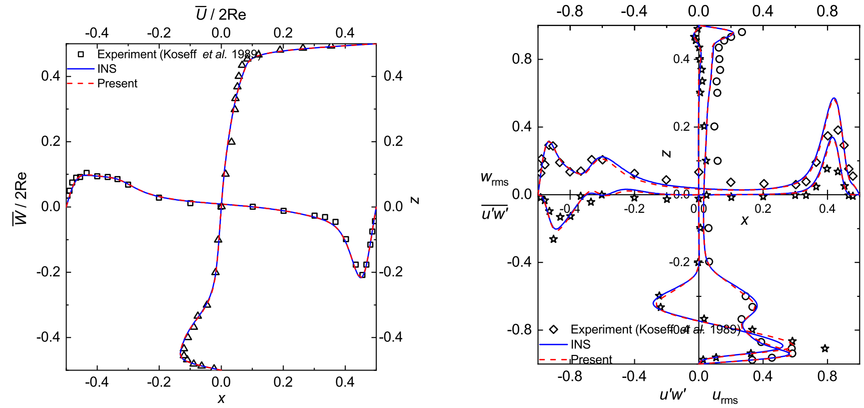

Figure 1: (a) Mean and (b) Reynolds stress profile along the horizontal and vertical centerlines in the symmetry plane. The symbols are experimental results of Prasad et al. [5].

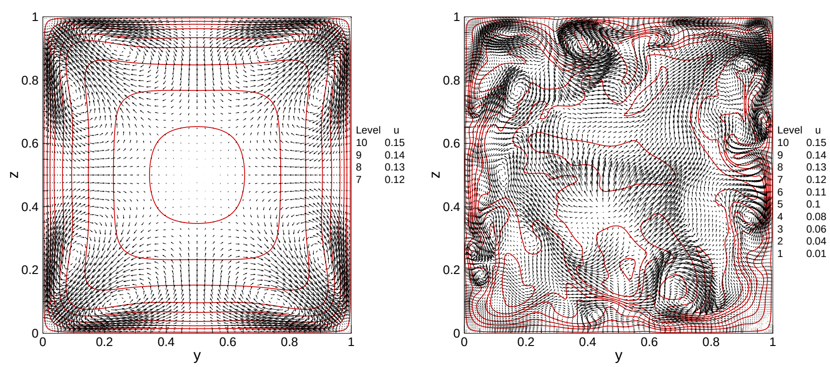

Figure 2: (a) Contours of mean streamwise velocity and velocity vectors of mean secondary flow. (b) Instantaneous streamwise velocity contours and secondary velocity vector at $x = \pi$.

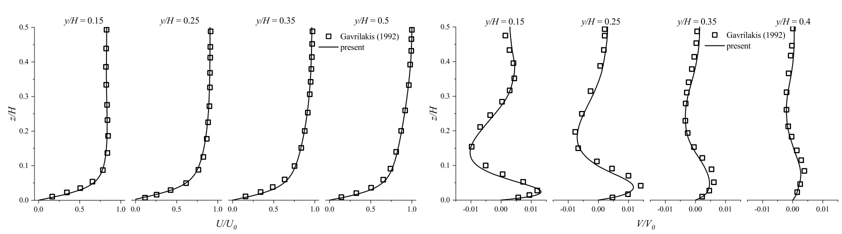

Figure 3: Comparison of the calculated mean (a) streamwise and (b) spanwise velocity profile with the simulations of Gavrilakis [7].