DNS OF TURBULENT CHANNEL AND PIPE FLOW WITH HIGH REYNOLDS NUMBER

Shin-ichi Satake
Department of Applied Electronics Engineering,
Tokyo University of Science
2641 Yamazaki, Noda, Chiba, 278-8510, Japan
satake@te.noda.tus.ac.jp

Kunugi Tomoaki
Department of Nuclear Engineering,
Kyoto University
Yoshida, Sakyo, Kyoto, 606-8501, Japan
kunugi@nucleng.kyoto-u.ac.jp

ABSTRACT

A direct numerical simulation (DNS) of turbulent channel flow has been carried out to understand the effects of Reynolds number. In this study, the Reynolds number for channel flow based on a friction velocity and channel half width was set to be constant; Re_τ = 1100. The number of computational grids used in this study was 1024 × 1024 × 768 in the x-, y- and z -directions, respectively. The turbulent quantities such as the mean flow, turbulent stresses and the turbulent statistics were obtained via present DNS.

INTRODUCTION

A fully developed channel and pipe flow are one of turbulent wall-bounded flow, which have been investigated in many researchers. In the first stage, although the researches were only experiments, in recent years, direct numerical simulation (DNS) is performed by development of a computer and detailed database is offered. The pioneering DNS of turbulent channel flow (Kim et al., 1987) and the turbulent pipe flow (Eggeles et al., 1994) were performed from Re_τ=180. In recently, the channel turbulent flow for Re_τ=650(Iwamoto et al., 1999), Re_τ=800(Miyamoto et al., 2002) were performed. Furthermore, the high Reynolds number (Re_τ=1050) equivalent to the experiment value of Laufer.(1954) was carried out by the authors group (Satake et al., 2000). The aim of this study is to investigate the Reynolds number dependence in turbulent channel flow and pipe flow. Specifically, in order to consider the Reynolds number effect of turbulent channel and pipe flow about the case beyond Re_τ=1000, the turbulent channel flow of Re_τ=1100 is performed, and authors' existing pipe data of Re_τ=1050 (Satake et al., 2000) is compared.

NUMERICAL PROCEDURE

We have developed that the DNS code with cylindrical coordinates can numerically solve the continuity and momentum equations using the radial momentum flux formulation (Satake and Kunugi, 1998). In present study, however the spatial descritization in the homogeneous directions is changed from a second-order finite volume scheme to spectral method. The coordinate system is also changed from cylindrical to Cartesian grid system. In this code, the spectral method is used to compute to the special derivative in the stream and spanwise direction. The continuity and incompressible Navier-Stokes equations are

$$\frac{\partial u_i}{\partial x_i} = 0,$$

(1)

$$\frac{\partial u_i}{\partial t} + \frac{\partial u_i u_j}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \nu\frac{\partial^2 u_i}{\partial x_i^2}.$$

(2)

In Fourier space, the spatial derivative operator for continue equation is computed:

$$\widehat{\frac{\partial u_i}{\partial x_i}} = ik_x\widehat{u} + \frac{d\widehat{v}}{dy} + ik_z\widehat{w},$$

(3)

where the k_x, k_z are wave number in the stream and spanwise directions, respectively. The derivative in wall normal direction is computed by the second order finite difference scheme at staggered grid arrangement in Fig. 1. The stream and spanwise components located at the pressure point. The wall normal component only shifted to the half mesh size. The second-order finite difference method in y-direction on a staggered mesh system is applied to the stretched grid by hyperbolic tangent function. For the second derivative, the same manner can be applied and the term is expressed as:

$$\widehat{\frac{\partial^2 u}{\partial x^2}} = -k_x^2\widehat{u}.$$

(4)

In Fourier space, the nonlinear terms in the streamwise direction are

$$\widehat{\frac{\partial u_i u_j}{\partial x_j}} = ik_x\widehat{uu} + \frac{d\widehat{uv}}{dy} + ik_z\widehat{uw}$$

(5)

However, $\widehat{uu}$,$\widehat{uv}$,$\widehat{uw}$ must be calculated in physical space. Because the grid point used to compute the nonlinear terms had 1.5 times finer resolution in the directions to remove aliasing errors.

These equations in time are integrated by using the fractional-step method (Dukowicz and Dvinsky, 1992). The second-order Crank-Nicholson scheme is applied to the viscous terms treated implicitly and a modified third-order Runge-Kutta scheme (Spalart et al., 1991) is used for other terms explicitly. The Helmholtz equation for viscous terms and pressure Poisson equations are only solved by using tridiagonal matrix technique in Fourier space.

COMPUTATIONAL CONDITION

The number of grid points, the Reynolds number and grid resolutions summarized in Table 1. To perform a DNS with high Reynolds number in this study, a grids size of 1024 × 1024 × 768 is adopted for 224GB main memory as 32 PEs on a vector-parallel computer Fujitsu VPP 5000 at JAERI. The Reynolds number based on the friction velocity, viscosity and the channel half width (δ) is assumed to be 1100. The periodic boundary conditions are applied to the streamwise (x) and the spanwise (z) directions. As for the wall normal direction (y), non-uniform mesh spacing specified by a hyperbolic tangent function is employed. The number of grid points is 1024 × 1024 × 768 in the $x-$, $y-$ and $z-$directions, respectively. The all velocity components imposed the non-slip condition at the wall.

RESULTS

Mean velocity profiles are shown in Fig. 3. Iwamoto et al. (2002) found that the effect of Reynolds number for channel flow is at Re_τ =110,150,300,400,650. The logarithmic profile at Re_τ =650 appears clearly. The profile is coincident with the $y^+ = 2.5lny^+ + 5.0$. The present result also attends the equation. However, the result in pipe flow (Re_τ =1050) by Satake et al. (2000) is not coincident with the equation. In the logarithmic region, the result of pipe flow located at between $y^+ = 2.5lny^+ + 5.0$ and $y^+ = 2.5lny^+ + 5.5$. Figure 4 shows the distributions of velocity fluctuations. It is interesting that the streamwise component at the peak location for channel at Re_τ =1100 is coincident with that of pipe DNS. For the distributions of wall normal velocity fluctuations in Fig. 5 , the values at peak location for channel at Re_τ =1100 and pipe at Re_τ =1050 are larger than that of channel at Re_τ =650. The similar tendency also appears in the spanwise component in Fig. 6. Thus, the turbulence behavior in the DNS with High Reynolds number tends to be more isotropic than that low Reynolds number, the streamwise component near wall region saturated, and the behavior of the wall normal and spanwise velocity fluctuating components near wall region is more enhance near wall region.

SUMMARY

The DNS for the channel flow with High Reynolds number was carried out. The result is compared with the channel flow and previous DNS data for pipe flow. Although further computations will be necessary to confirm the current turbulent statistics, the effect of Reynolds number can be predicted via this DNS. The streamwise component near wall region is saturated. The other components are more enhanced. The DNSs for channel and pipe have been continued. More detailed results, completed convergence for statistical data and discussion will be presented in the poster session.

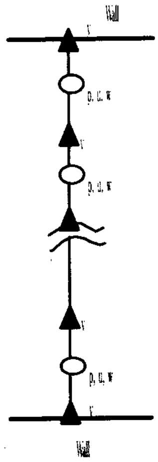

Figure 1: Staggered grid system

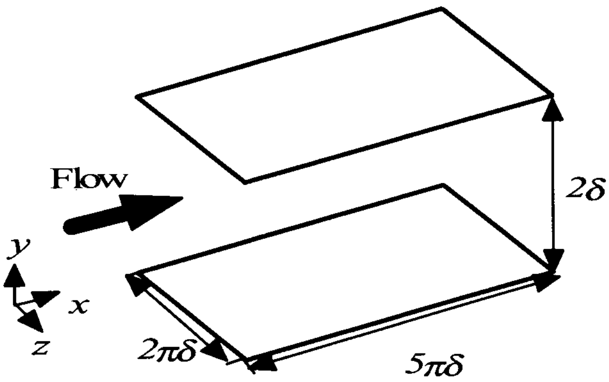

Figure 2: Computational domain

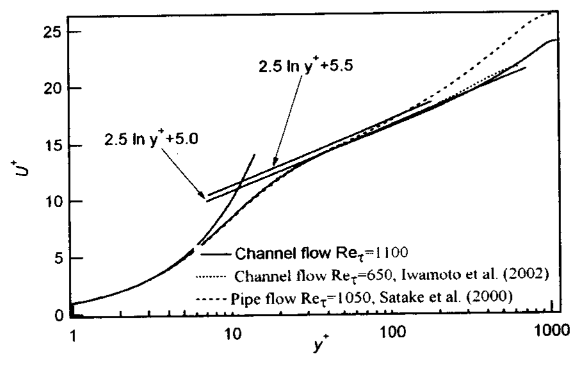

Figure 3: Mean velocity profile

REFERENCES

Dukowicz, J. K. and Dvinsky, A. S., 1992, J. Comp.,Phys., Vol.102, No.2, pp. 336-347.

Eggels, J.G.M., Unger, F., Weiss, M.H., Westerweel, J. Adrian, R.J., Friedrich, R., and Nieuwstadt, F.T.M.,1994,

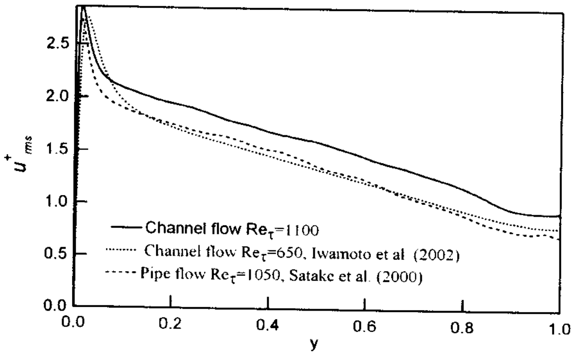

Figure 4: Streamwise velocity fluctuation profile

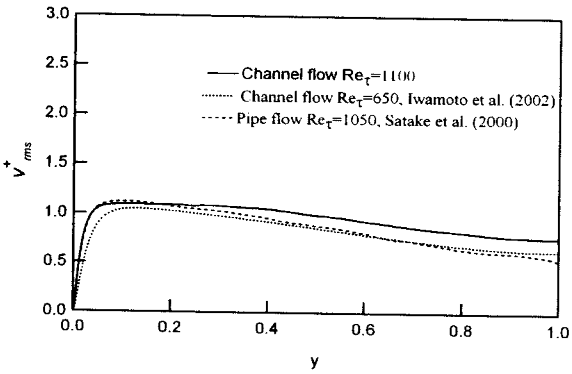

Figure 5: Wall normal velocity fluctuation profile

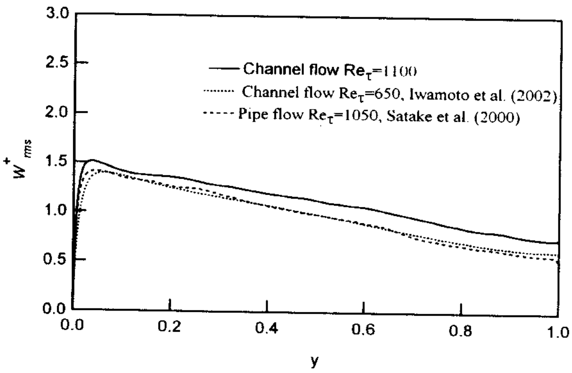

Figure 6: Spanwise velocity fluctuation profile

Table 1: Calculation parameter

Re_τ	region	Grid point (x,y,z)
1050	$5\pi\delta \times 2\delta \times 2\pi\delta$	1024×1024 × 768

"Fully developed turbulent pipe flow: comparison between direct simulation and experiment, J. Fluid Mech., Vol. 268, pp. 175-209.

Kim, J., Moin, P., and Moser, R.D.,1987, " Turbulence statistics in fully developed channel flow at low Reynolds number, J. Fluid Mech., Vol. 177, pp. 133.

Laufer, J. , "The structure of turbulence in fully developed pipe flow, " NACA report 1174,1954.

Miyamoto, T., Tanahashi, M. and Miyauchi, T. 2002," Scaling law of fine scale eddies in high Reynolds number turbulent channel flow,"Proc. of 16 th Computational Fluid Dynamics Conf., in Tokyo, E12-2.

Spalart, P.R.., Moser, R. D. and Rogers, M.,M., 1991, J. Comp. Phys., 96, pp. 297-324.

Satake, S. and Kunugi, T., 1998,"Direct numerical simulation of turbulent pipe flow," Bulletin JSME Vol.64, pp. 65-70.

Satake, S., Kunugi, T. and Himeno, R., 2000, " High Reynolds Number Computation for Turbulent Heat Transfer in a pipe flow," Lecture Notes in Computer Science 1940, High Performance Computing, M. Valero et al. (Eds.), Springer-Verlag Berlin Heidelberg, pp. 514-523.

Iwamoto, K., Suzuki, Y., and Kasagi, N., 2002,"Database of fully developed channel flow,"THTLAB Internal Report, N0. ILR-0201,see http://www.thtlab.t.u-tokyo.ac.jp.