1 Introduction

Reduction of flow drag in turbulent flow has a great impact on the global environment. For instance, higher energy efficiency in the major transportation system will lead to reduce emission of greenhouse gases and to prevent the exhaustion of fossil fuels. The drag of the fluid flow can be classified into two groups: the pressure drag by flow separation and the skin-friction drag by the viscosity of the fluid. It is well-known that the shape optimization, such as a streamlined form, reduces the pressure drag. Such a passive control has already been applied for aircrafts or bullet train. On the other hand, there have been great efforts devoted to the reduction of the skin-friction drag. Despite the extensive studies have been carried out, a reduction control for external flow is still far from practice due to its difficulty in their mechanism or size; thus, it is still under investigation. However, the skin-friction drag accounts for a large portion in the total fluid drag. For instance, it reaches about 50% around an airfoil and about 100% in the pipe flow (Gad-el Hak, 1996). Therefore, the realization of higher energy efficiency derived from the skin-friction drag reduction will greatly contribute to the engineering development.

Among the variety of ideas for skin-friction drag reduction, uniform blowing (UB) from the wall attracts considerable attentions. Kametani and Fukagata (2011) conducted a direct numerical simulation (DNS) of a spatially developing turbulent boundary layer at $Re_T = 160$ with UB. By UB, the skin-friction drag is reduced while turbulence is enhanced. The control efficiency of UB is estimated to be higher than that of other control methods. However, the Reynolds number assumed in that study is quite low as compared with turbulent flow in engineering applications. It is said that the friction Reynolds number reaches the order of $10^4$ to $10^5$ in the flow around the wing of aircrafts. To consider the practical application of UB, the investigation of the control effect in high Reynolds number flows is necessary.

Although computing performance has dramatically been raised in late years, enormous computational cost needed for a high Reynolds number wall-bounded flow is still an obstacle for practical use of DNS. The total number of grid for three-dimensional DNS is scaled as $Re^{3/4}$. Thus, in order to simulate practical engineering flows, turbulence modeling is necessary to study turbulent high Reynolds number flow. The steady or unsteady Reynolds-averaged Navier-Stokes (RANS) simulation is widely used for computation of high Reynolds number turbulent flow. Although RANS simulation gives a good prediction for flows around relatively plain shape, selecting a suitable model for the geometry is necessary. This means that RANS has disadvantage to address geometry-specific flow. On the other hand, the large eddy simulation (LES) has become increasingly popular for three-dimensional unsteady flow or flow with massive separations. However, to capture small turbulence structure near the wall, LES requires severe fine grid resolution. Indeed, the grid spacings in wall unit of LES needs $\Delta x^+ \approx 50$, $\Delta y^+ \approx 1$, $\Delta z^+ \approx 15$ (Sagaut and Deck, 2009). LES of a high Reynolds number wall bounded flow is still impossible due to that high computational resources required.

To overcome these modeling issues and to simulate high Reynolds number wall-bounded flows accurately as well as much less costly, hybrid RANS/LES methods have been developed. In hybrid RANS/LES models, RANS is used near the wall to predict the attached boundary layers, and in the region away from the wall, LES is carried out to resolve three dimensional large eddies accurately. A detached eddy simulation (DES) proposed by Spalart et al. (1997) is one of hybrid RANS/LES models. DES is based on the Spalart-Allmaras RANS model (Spalart and Allmaras, 1992) and originally developed to treat massively separated high Reynolds number flows such as flow around an airfoil. DES has been applied to various geometries. Travin et al. (1999) performed DES of
a circular cylinder with turbulent separation. Forsythe et al. (2006) compute the motion of F-15E entering a spin by using DES. DES has been used not only for turbulent flows with massive separation, but also for wall-bounded flows. Nikitin et al. (2000) conducted DES of turbulent channel flow at a high friction Reynolds number, 80,000. Prediction using DES is good agreement with experimental data or other simulations even at that high Reynolds number. However, the skin-reduction control for DES has not been carried out yet. Thus, it is necessary to investigate whether DES is applicable to the reduction control such as UB.

In the present study, DES is performed for a high Reynolds number spatially developing turbulent boundary layer with UB. The mechanism of drag reduction is investigated, and the Reynolds number dependency of drag reduction rate is discussed.

2 Detached eddy simulation

Governing equations

The governing equations of the present DES are the incompressible continuity, spatially-filtered Navier-Stokes equation and transport equation for the eddy viscosity, \( \chi \), i.e.,

\[
\nabla \cdot \mathbf{u} = 0
\]

\[
\frac{D\mathbf{u}}{Dt} = -\nabla p + \frac{1}{Re} \nabla \cdot ((1 + \nu_t)(\nabla \mathbf{u} + (\nabla \mathbf{u})^T))
\]

\[
\frac{D\chi}{Dt} = C_{b1} \left( S + \frac{1}{Re \kappa^2 d^2} f_{v2} \right) \chi
+ \frac{1}{Re\sigma} \left[ \nabla \cdot ((1 + \chi)\nabla \chi) + C_{b2}(\nabla \chi)^2 \right]
- \frac{1}{Re} C_{w1} f_w \left( \frac{\chi}{d} \right)^2
\]

Here, \( \mathbf{u} \) and \( p \) denote the velocity and the pressure, respectively. Superscripts (\()^*\) denote dimensional values, and all variables are non-dimensionalized by the fluid density, \( \rho^* \), the freestream velocity \( U_{\infty}^* \), and the displacement thickness at the inlet of the computational domain \( \delta_{00}^* \). The Reynolds number is defined as \( Re = U_{\infty}^* \delta_{00}^* / \nu^* \), where \( \nu^* \) is the kinematic viscosity. In the transport equation of \( \chi \), \( C \) and \( \sigma \) are constants, \( S \) denotes the magnitude of the vorticity, and \( f \) is the function of \( \chi \).
No control

\[
\nu_t = \chi f_{v1} = \chi \frac{\chi^3}{\chi^3 + C_{v1}^3}
\]  \hspace{1cm} (4)

In the DES model, the interface between the RANS mode and the LES mode is determined by the length scale \(\tilde{d}\), defined as

\[
\tilde{d} \equiv \min(d, C_{\text{DES}} \Delta),
\]  \hspace{1cm} (5)

where \(d\) is the distance to the wall and \(\Delta\) is the maximum grid spacing: \(\Delta = \max(\Delta x, \Delta y, \Delta z)\). \(\Delta x\) and \(\Delta z\) is independent of wall-normal direction respectively. When the distance \(d\) becomes smaller than \(C_{\text{DES}} \Delta\) near the wall, DES works as Spalart-Allmaras mode and the LES mode is determined by the length \(d\). In a region away from the wall, where \(d = C_{\text{DES}} \Delta\), the mode switches into the LES mode.

Other characteristic functions are defined below.

\[
f_{v1} = \frac{\chi^3}{\chi^3 + C_{v1}^3},
\]  \hspace{1cm} (6)

\[
f_{v2} = 1 - \frac{\chi}{1 + \chi f_{v1}},
\]  \hspace{1cm} (7)

\[
r = \frac{Re_\ell \overline{S^2 d^2}}{\chi f_{v1}} + f_{v2},
\]  \hspace{1cm} (8)

\[
g = r + C_{w2}(r^6 - r),
\]  \hspace{1cm} (9)

\[
f_w = g \left(1 + \frac{C_{w3} \overline{S^2}}{g^6 + C_{w3}^6} \right)^{\frac{1}{2}}.
\]  \hspace{1cm} (10)

Here, \(S\) is the magnitude of the vorticity, constants are \(C_{\delta_{11}} = 0.1355\), \(\sigma = 2/3\), \(C_{\delta_{22}} = 0.622\), \(\kappa = 0.41\), \(C_{v1} = C_{\delta_{11}}/\sigma^2 + (1 + C_{\delta_{22}})/\sigma\), \(C_{w2} = 0.3\), \(C_{w3} = 2\), \(C_{v1} = 7.1\), \(C_{\text{DES}} = 0.65\).

**Numerical procedure**

The present DES code is based on a spatially turbulent boundary layer DNS code developed by Kametani and Fukagata (2011). The spatial discretization uses the energy-conservative second-order finite difference scheme. The time integration uses the low-storage third-order Runge-Kutta/Crank-Nicolson scheme.

The computational domain is composed of two regions: a driver region and a main region, as shown in Figure 1. The streamwise, the wall-normal, and the spanwise lengths of the driver and main regions are \((L_x^D, L_y^D, L_z^D) = (3\pi \delta_{\text{ref}}, 3\delta_{\text{ref}}, \pi \delta_{\text{ref}})\) and \((L_x, L_y, L_z) = (9\pi \delta_{\text{ref}}, 3\delta_{\text{ref}}, \pi \delta_{\text{ref}})\), where the superscript \(D\) denotes the driver region. The numbers of grid points are \((N_x^D, N_y^D, N_z^D) = (128, 96, 128)\) and \((N_x, N_y, N_z) = (512, 96, 128)\).

In order to generate an inflow condition, the recycle method of Jewkes et al. (2011) is used in the driver region. The recycle station is set at \(x_D = 2\pi \delta_{\text{ref}}\).

The periodic boundary condition is applied to the spanwise direction in both domains. The upper boundary conditions for the streamwise velocity \(u\), the wall-normal velocity \(v\), the spanwise velocity \(w\), the eddy viscosity-like variable \(\chi\), the kinematic eddy viscosity \(\nu_t\) are \(\partial u / \partial y = \partial v / \partial y = \partial \chi / \partial y = \partial \nu_t / \partial y = 0\) and \(w = 0\) respectively. On the wall, no-slip condition is applied in the driver region and uniform blowing velocity \(v = V_{\text{ctr}}\) is added on the wall in the main region. The convective boundary condition is applied at the outlet of each domain. For the pressure at the inlet and the outlet of computational domain, the Navier-Stokes characteristic boundary condition (NSCBC) of Miyachi et al. (1996) is used.

In the present study, the friction Reynolds number \((Re_f = \overline{u_*^2 \delta_{\text{ref}}}/\nu^*}) is set to be 160 and 2000 at the inlet of the main region, where \(u_*^*\) is local friction veloc-
ity. Hereafter, the quantities are non-dimensionalized by the local friction velocity of the uncontrolled flow $u_f$.

At $Re_x = 160$, the grid spacings in wall unit in the streamwise and the spanwise directions are $\Delta x^+ = 8.83$ and $\Delta z^+ = 3.93$, respectively. In the wall-normal direction, the minimum grid spacing is $\Delta y^+_{\text{min}} = 0.47$ and the maximum grid spacing is $\Delta y^+_{\text{max}} = 6.67$. At $Re_x = 2000$, $\Delta x^+ = 40.3$ and $\Delta z^+ = 108$. In the wall-normal direction, the minimum grid spacing is $\Delta y^+_{\text{min}} = 1.05$ and the maximum grid spacing is $\Delta y^+_{\text{max}} = 102$.

The magnitude of UB is set to be 0% (uncontrolled), 0.1% or 0.5% of the freestream velocity. A transition zone is placed at $0 \leq x \leq \pi$, in which the control amplitude is gradually increased as shown in Figure 2.

3 Results and discussion

As a representative case, the results at $Re_x = 2000$ is shown from Figure 3 to Figure 7. In all figures, the chained line in the figures represents the boundary between the RANS region and the LES region.

Figure 3 shows a contour plot of an instantaneous streamwise velocity field of the uncontrolled case and UB case. The streamwise velocity in UB case becomes slower than that of the uncontrolled flow near the wall after transition zone at $x^+ \approx 6000$. The mean streamwise velocity profile at $x = 6\pi$ is shown in Figure 4. The effect of UB depends on its amplitude. It is found that the velocity profiles are shifted away from the wall by UB. This push-up effect of UB makes the streamwise velocity at the same height near the wall slower than that in the uncontrolled flow.

In the uncontrolled case, the momentum thickness is found to develop downstream as shown in Figure 5. As shown in Figure 5, UB is found to drastically increase the boundary layer thickness.

As observed in Figure 6, UB enhances the Reynolds shear stress in the region away from the wall. On the other hand, UB reduces the viscous shear stress in the region near the wall. As a result, the local skin friction coefficient, $c_f$, shown in Figure 7 is obviously reduced by UB in the controlled region. This is basically due to the modification of the mean streamwise velocity profile as shown in Figure 4. These modifications are in accordance with the DNS result at $Re_x = 160$ of Kametani and Fukagata (2011). Figure 8 shows the Reynolds dependency of the drag reduction rate. Here, the global skin friction drag coefficients is defined as

$$C_f = \frac{1}{L_{ctr}} \int_0^{L_{ctr}} c_f dx$$ \hspace{1cm} (11)

where $L_{ctr}$ is the streamwise length of the main region. The drag reduction rate is expressed by using the global friction coefficients as

$$R = \frac{C_{f, nc} - C_{f, ctr}}{C_{f, nc}}$$ \hspace{1cm} (12)

where $C_{f, nc}$ and $C_{f, ctr}$ denote the friction coefficients in the uncontrolled and the UB cases. At any Reynolds numbers, UB is found to reduce the skin-friction drag. A higher drag reduction rate is obtained at a stronger blowing; the drag reduction of 29.0% in 0.1% UB case and 81.1% in 0.5% UB case is obtained at $Re_x = 2000$. It is also found that regardless of the Reynolds number the drag reduction rate is roughly scaled by the UB velocity expressed in wall units, $V^+$.

There is a difference between the present DES and the DNS result in 0.1% UB case at $Re_x = 160$. A similar amount of error may be included in the drag reduction rate at $Re_x = 2000$. This is due to underestimation of $c_f$ in uncontrolled flow as shown in Figure 7.

The underprediction of $c_f$ is found in other DES studies. Nikitin et al. (2000) carried out DES of turbulent channel flow and reported that $c_f$ is underestimated about 15% at several Reynolds numbers. This is considered due to so-called log-layer mismatch: the mean streamwise velocity takes an outward shift from
log-law profile after RANS/LES interface. In fact, this issue is observed in the present result of uncontrolled flow as shown in Figure 4. Not only in DES, but in other hybrid RANS/LES models, the log-layer mismatch is also reported and it is said that the log-layer mismatch is common issue for hybrid turbulence models.

According to Spalart et al. (2006), the log-layer mismatch is caused by an insufficient supply of the Reynolds shear stress. Detached eddy simulation is originally developed to treat the whole boundary layer in the RANS mode and elsewhere, such as separated regions, in the LES mode. However, it is problematic when the interface is set inside the boundary layer. Turbulence is almost modeled in the RANS region; the resolved Reynolds stress becomes large and the modeled Reynolds stress gradually decreases in the LES region. The total Reynolds stress, however, cannot develop to the required level because the grid is not fine enough in the LES region to provide the resolved velocity fluctuations in the boundary layer. As a result, the LES mode reduces the eddy viscosity to result in an insufficient modeled Reynolds stress without sufficient resolved Reynolds stress to cover the balance. This is referred to as a modeled-stress depletion (Spalart et al. 2006). The present base flow at $Re = 2000$ in Figure 9 actually shows the lack of Reynolds shear stress around the RANS/LES interface. To keep the momentum conservation, the velocity gradient is steepened to compensate for the low supply of shear stresses. Due to this log-layer mismatch, $c_f$ is underestimated.

As an attempt to overcome this problem, Spalart et al. (2006) proposed a delayed detached eddy simulation (DDES), and Deck (2012) developed a zonal detached eddy simulation (ZDES), which clarify individual RANS and LES regions. By using these modification of DES, the prediction of drag reduction rate can be improved. For more accurate prediction of the effect of UB, further investigation using these improved DESs will be necessary.

4 Conclusions

Detached eddy simulation of a spatially developing turbulent boundary layer at a high Reynolds number
with uniform blowing (UB) was carried out aiming at skin-friction drag reduction. It is confirmed that, even at $Re_f = 2000$, the velocity profiles are shifted away from the wall, the Reynolds shear stress is enhanced, and the viscous shear stress is reduced. The trends in modifications are in accordance with the DNS results of Kametani and Fukagata (2011) at $Re_f = 160$. It is also found that the drag reduction rate is roughly scaled by the blowing velocity expressed in the wall units.

Acknowledgments

The authors are grateful to Drs. Shinnosuke Obi and Keita Ando (Keio University) for fruitful discussion. This work was partly supported through Grant-in-Aid for Scientific Research (C) (No. 25420129) and Grant-in-Aid for JSPS Fellows (No. 24-3450) by Japan Society for the Promotion of Science (JSPS).

References


