REDUCTION OF TURBULENT WALL FRICTION BY SPINNING DISCS

Pierre Ricco[1] & Stanislav Hahn[2]
[1]*Department of Mechanical Engineering, The University of Sheffield, S1 3JD Sheffield, UK*
[2]*Honeywell Turbo Technologies, Turanka 100/1236, Brno, Czech Republic*

Abstract The turbulent drag reduction technique proposed by Keefe[4, 5] is studied for the first time by direct numerical simulation in the turbulent channel flow geometry at a Reynolds number of R_τ=180, based on the friction velocity of the stationary-wall case and the half channel height. The actuator consists of arrays of flush-mounted discs which rotate at constant angular velocity. For a fixed maximum disc tip velocity, drag reduction can be achieved when the disc diameter is larger than a threshold, while below this threshold the drag increases. We find a maximum drag reduction of 23%. The net power saved, computed by taking into account the power spent to activate the discs against the fluid viscous resistance, is found to be positive and reach 10%. We estimate the disc-flow parameters required for flows of technological interest and discuss possible future implementations based on micro-electromagnetic motors and micro-air turbines.

INTRODUCTION

The reduction of turbulent wall friction is one of the major challenges in fluid mechanics research and represents an opportunity to achieve immense energy savings. The need for lower fuel consumption and improved environmental sustainability has spurred widespread efforts in the academic and industrial worlds to study new drag-reduction techniques. Amongst the active methods, the techniques of spanwise wall oscillations, first studied by [3], and of streamwise-traveling waves of spanwise wall velocity, first investigated by [7], have attracted interest because of the large drag reductions and net energy savings (when the power used to move the walls is taken into account). However, the realization of these methods on systems of technological importance, such as flows over aircraft wings, appears elusive. One of the main reasons lies in the estimated optimal oscillation frequency being very large, i.e. about 15 kHz over the wing of a commercial aircraft at a cruise speed of 225 m/s at 10 km above sea level. The novel device proposed by [4, 5], based on discs which are flush-mounted on a flat surface and rotate at constant angular velocity, may instead offer interesting opportunities for industrial applications. To the best of our knowledge, neither experimental nor numerical studies exist on this type of flow. Our objective is to investigate the effects of the disc diameter and rotational frequency on the near-wall turbulence by means of direct numerical simulations in the channel flow geometry. The focus is on the turbulent friction drag and on the net power saved [8].

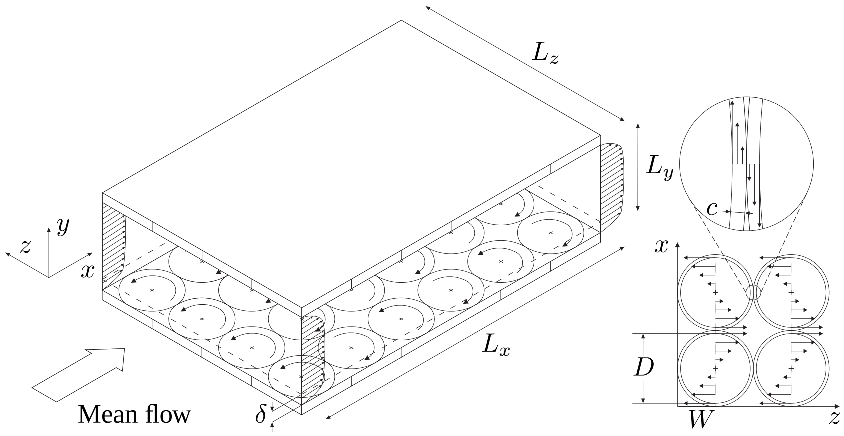

Figure 1. Schematic of the turbulent channel flow with rotating discs. The graph on the right shows the z-component of the wall velocity along lines parallel to x and passing through the disc centres. Lengths are scaled by h^*, the half channel height, and velocities are scaled by U_p^*, the maximum centreline velocity of the laminar Poiseuille flow at the same mass flow rate.

RESULTS

A pressure-driven turbulent flow between infinite parallel flat plates at a Reynolds number of R_τ=180, based on the friction velocity of the stationary-wall case and the half channel height, has been studied by direct numerical simulations. The open-source numerical code available on the Internet [2] has been modified to impose the rotation of the discs on the walls. The code solves the incompressible Navier-Stokes equations in the channel flow geometry using Fourier series expansions along the streamwise (x) and spanwise (z) homogeneous directions, and Chebyshev polynomials along the wall-normal direction (y). The discs are located next to one another, have a diameter D and rotate at a constant angular

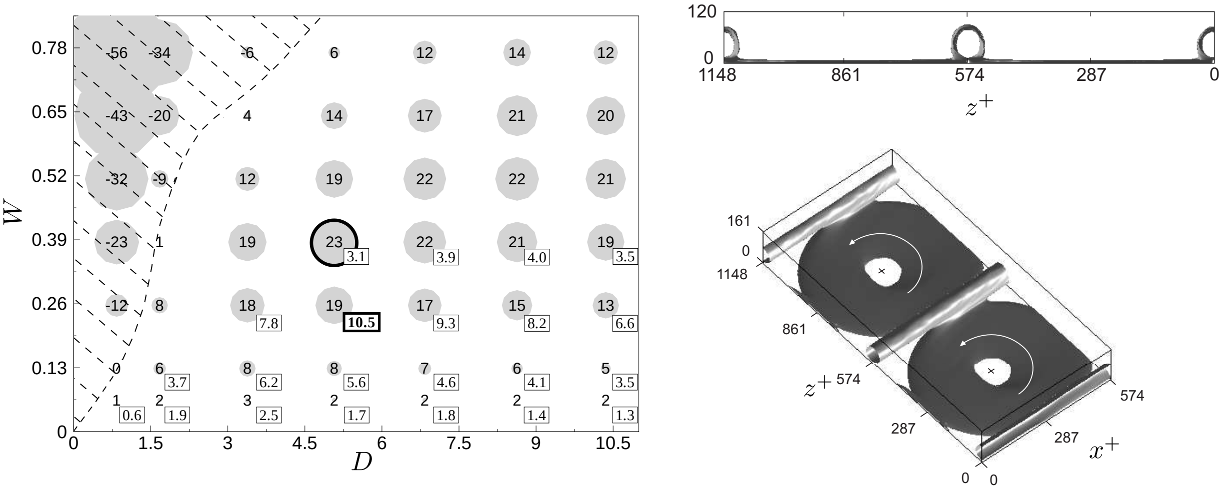

Figure 2. Left: Map of $\mathcal{R}(D, W)(\%)$. The size of the grey circles is proportional to the absolute value of $\mathcal{R}$. The shaded areas highlight the drag-increase cases and the zero-$\mathcal{R}$ lines are found by linear data interpolation. The maximum $\mathcal{R}$=22.9% is circled and the boxed values report the positive net power saved $\mathcal{P}_{net}$(%) (the thick box denotes the maximum $\mathcal{P}_{net}$=10.5%). Right: Isosurfaces of the time-averaged disc flow. The symbol $+$ denotes scaling by the viscous inner units of the flow, i.e. by the kinematic viscosity ν^* and the friction velocity $u_\tau^* = \sqrt{\tau_w^*/\rho^*}$, where τ_w^* is the space- and time-averaged wall-shear stress and ρ^* is the density.

velocity Ω with tip velocity W=$\Omega D/2$. Discs neighbouring along x have opposite direction of rotation, while the direction of rotation along rows in the z direction is unchanged.

Figure 2 (left) shows the three-dimensional map of the drag reduction $\mathcal{R}(D, W)(\%)$. For fixed W, drag reduction occurs when D is larger than a threshold, while the drag increases for smaller D (the shaded areas denote drag-increase cases). An overall maximum $\mathcal{R}$=22.9% is computed for D=5.07 and W=0.39 (D^+=801 and W^+=10.2). Another quantity of interest is the net power saved $\mathcal{P}_{net}$, defined as the difference between the power saved thanks to the wall motion (=$\mathcal{R}$ at constant mass flow rate) and the power spent $\mathcal{P}_{sp,t}$, i.e. $\mathcal{P}_{net}(\%) \equiv \mathcal{R}(\%) - \mathcal{P}_{sp,t}(\%)$. The map in figure 2 (left) shows that a positive $\mathcal{P}_{net}$ may occur for $W \leq 0.39$ and a maximum $\mathcal{P}_{net}$=10.5% is computed for D=5.07 and W=0.26 (D^+=820, W^+=6.7, $\mathcal{R}$=19.2%). Isosurfaces of $q^+ \equiv \sqrt{u_d^{+2} + w_d^{+2}}$=2.3 (where u_d and w_d are the time-averaged streamwise and spanwise velocity components, the time- and space-averaged streamwise flow has been subtracted), shown in figure 2 (right, bottom), distinctly visualize the disc flow as near-wall circular patterns of thickness of about $10\nu^*/u_\tau^*$. Well-defined, streamwise-elongated structures appear over sections of stationary wall, where the shear brought about by the tangential disc flow is largest because discs neighbouring along z have opposite sense of rotation. Figure 2 (right, top) shows that these structures have a round shape when observed from the $y-z$ plane at x^+=0, are centred at about $y^+ \approx 40$, and are higher than the ring-shaped patterns as they extend to about $y^+ \approx 80$.

We estimate the optimal diameter and rotational frequency if the discs were implemented over a commercial aircraft wing, over hulls of large-scale ships and over high-speed trains ($R_\tau \approx 5000$); the diameters are 6.7 mm, 6.5 mm, and 8.1 mm, respectively, and the rotational frequencies are 3700 Hz, 170 Hz, and 1140 Hz. Rotation rates of $\mathcal{O}(10^3)$ Hz may be obtained by electromagnetic motors ($D^* \approx 2$ mm [6]) and by the micromachined air turbines ($D^* \approx 4$ mm [1]). Future work should be focussed on assessing the effect of Reynolds number and to verify whether these estimates are valid at higher R_τ. As foreseen by [5], the possibilities of improvement, in terms of reduction of drag, net power saved, separation control, lift enhancement, may be broad and interesting.

References

[1] L.G. Frechette et al. *J. Microel. Sys.*, **14**(1):141–152, 2005.

[2] J. F. Gibson. Channelflow: a spectral Navier-Stokes simulator in C++. *http://www.channelflow.org/*, 2006.

[3] W. J. Jung, N. Mangiavacchi, and R. Akhavan. *Phys. Fluids A*, **4**(8):1605–1607, 1992.

[4] L. Keefe. *AIAA Paper*, **97-0547**, 1997.

[5] L. Keefe. *United States Patent*, **5,803,409**, 1998.

[6] D. Kuang-Chen Liu, J. Friend, and L. Yeo. *Acoust. Sci. & Tech.*, **31**:115 – 123, 2010.

[7] M. Quadrio, P. Ricco, and C. Viotti. *J. Fluid Mech.*, **627**:161–178, 2009.

[8] P. Ricco and S. Hahn. Submitted (second revision) to J. Fluid Mech., 2013.