International Journal of Heat and Fluid Flow 72 (2018) 317–328

Contents lists available at ScienceDirect

International Journal of Heat and Fluid Flow

journal homepage: www.elsevier.com/locate/ijhff

Turbulent rectangular ducts with minimum secondary flow

A. Vidal[*,a], R. Vinuesa[b], P. Schlatter[b], H.M. Nagib[a]

[a] Department of MMAE, Illinois Institute of Technology, Chicago, IL 60616, Sweden
[b] Linné FLOW Centre, KTH Mechanics SE-100 44, Stockholm, Sweden Swedish e-Science Research Centre (SeRC), Stockholm, Sweden

ARTICLE INFO

Keywords:
Wall-bounded turbulence
Turbulent duct flow
Side-wall geometry
Secondary flow
Direct numerical simulation

ABSTRACT

In the present study we perform direct numerical simulations (DNSs) of fully-developed turbulent rectangular ducts with semi-cylindrical side-walls at $Re_{\tau,\,c} \simeq 180$ with width-to-height ratios of 3 and 5. The friction Reynolds number $Re_{\tau,\,c}$ is based on the centerplane friction velocity and the half-height of the duct. The results are compared with the corresponding duct cases with straight side-walls (Vinuesa et al., 2014), and also with spanwise-periodic channel and pipe flows. We focus on the influence of the semi-cylindrical side-walls on the mean cross-stream secondary flow and on further characterizing the mechanisms that produce it. The role of the secondary and primary Reynolds-shear stresses in the production of the secondary flow is analyzed by means of quadrant analysis and conditional averaging. Unexpectedly, the ducts with semi-cylindrical side-walls exhibit higher cross-flow rates and their secondary vortices relocate near the transition point between the straight and curved walls. This behavior is associated to the statistically preferential arrangement of sweeping events entering through the curved wall and ejections arising from the adjacent straight wall. Therefore, the configuration with minimum secondary flow corresponds to the duct with straight side-walls and sharp corners. Consequences on experimental facilities and comparisons between experiments and various numerical and theoretical models are discussed revealing the uniqueness of pipe flow.

1. Introduction

One of the most noticeable characteristics of fully-developed turbulent flows through straight ducts with rectangular cross-section is the presence of secondary flow near the corners. The secondary flow refers to the non-zero mean cross-stream vertical V and spanwise W velocity components. These secondary motions are known as Prandtl's secondary motions of the second kind (Prandtl, 1952) and are entirely due to turbulence. The magnitude of the cross-flow is relatively weak compared with the streamwise velocity U but strong enough to significantly affect the turbulent flow statistics (Bradshaw, 1987).

In square ducts, the cross-flow consists of two counter-rotating vortices per duct quadrant which drive the cross-flow into the corner through the bisector (i.e., the duct diagonal), and out of it through the buffer-layer region in the wall-tangent direction. This region goes from $y^+ \simeq 5$ up to the beginning of overlap region. Therefore, the mean in-plane streamfuction Ψ is anti-symmetrically distributed with respect to the corner bisector. The underlying physical mechanisms that generate Prandtl's secondary flow of the second kind in square ducts have been addressed by many authors. According to Perkins (1970), the term associated to the secondary normal-stress difference in Eq. (2) acts as a source term in the transport equation of the mean streamwise vorticity Ω_x (1); note that the second term in Eq. (2) acts as a transport term. In these equations, v' and w' are the wall-normal and spanwise fluctuating velocity components and their equivalents in capital letters refer to their mean quantities. The wall-tangent velocity fluctuations become more constrained as the corner is approached, a fact that increases their spatial gradient, which in turn leads to a larger source term in Eq. (1). Moinuddin et al. (2004) performed experiments over external corners and analyzed the secondary Reynolds shear-stress $\overline{v'w'}$ and the anisotropy of the cross-stream deviatoric Reynolds-stress $\overline{v'^2} - \overline{w'^2}$. Krasnov et al. (2012) found that the magnitude of the mean cross-flow decreases if the cross-stream fluctuations are suppressed by a magnetic field. Near the corner, Huser and Biringen (1993) associated the Reynolds shear-stress components $\overline{u'v'}$ and $\overline{u'w'}$ with the transfer of momentum between the streamwise and the cross-stream normal stresses due to the inhomogeneous interaction between bursting events originated on the horizontal and vertical walls. The same authors related the nonzero secondary shear stress $\overline{v'w'}$ to the transport of momentum from the spanwise velocity fluctuations along the horizontal wall to the vertical fluctuations along the vertical wall and vice versa. Similar observations were made by Marin et al. (2016) in turbulent hexagonal ducts. Later, Pinelli et al. (2010) showed that high-speed streaks have a statistically preferential location in the near-corner region, thereby

* Corresponding author.
E-mail address: avidalto@hawk.iit.edu (A. Vidal).

https://doi.org/10.1016/j.ijheatfluidflow.2018.06.014
Received 3 November 2017; Received in revised form 5 June 2018; Accepted 27 June 2018
0142-727X/

transferring momentum from the streamwise to the cross-stream fluctuations in this region. The authors also found that a low-speed streak is preferentially located at each flank of the high-speed streak on each of the perpendicular walls. They related the mean streamwise vorticity distribution to the preferred buffer-layer location of the quasi-streamwise vortices associated with the in distributed streaks. Therefore, this mean quantity scales in viscous units. However, their study showed that the mean in-plane stream function scales in outer units. This behavior reflects the non-local nature of the Poisson Eq. (3), which relates both quantities, and highlights the multiscale character of the secondary flow. Other simulations of turbulent flow through square ducts have been performed by Gavrilakis (1992) at low Reynolds numbers and by Zhang et al. (2015) at friction Reynolds numbers Re_τ up to 1200, based on the total height of the duct.

$$V\frac{\partial\Omega_x}{\partial y} + W\frac{\partial\Omega_x}{\partial z} = S(y, z) + Re^{-1}\left(\frac{\partial^2\Omega_x}{\partial y^2} + \frac{\partial^2\Omega_x}{\partial z^2}\right), \tag{1}$$

$$S(y, z) = \frac{\partial^2}{\partial y\partial z}(\overline{v'^2} - \overline{w'^2}) - \left(\frac{\partial^2}{\partial y^2} - \frac{\partial^2}{\partial z^2}\right)\overline{v'w'}, \tag{2}$$

$$\left(\frac{\partial^2\Psi}{\partial y^2} + \frac{\partial^2\Psi}{\partial z^2}\right) = -\Omega_x. \tag{3}$$

Prandtl's secondary motions of the second kind were studied by Vinuesa et al. (2014, 2015) in rectangular ducts with varying aspect ratio AR (defined as the duct total width divided by its total height) using a similar numerical approach to the one presented in this study. The same authors performed direct numerical simulations (DNSs) at $Re_{\tau,\,c} \simeq 180$, which is the friction Reynolds number based on the friction velocity on the vertical centerplane $u_{\tau,\,c}$ and the half-height of the duct h, for AR between 1 and 14.4, and at $Re_{\tau,\,c} \simeq 360$ for $AR = 1$ and 3. They compared their results with the spanwise-periodic channel by Jiménez et al. (2004), where no mean cross-flow is present, and concluded that aspect ratios larger than 10 are required to obtain channel-like conditions in the centerplane region (Vinuesa et al., 2018). These conclusions provided valuable insight on how to reproduce two-dimensional channel flow in an experimental facility despite the necessary presence of side walls. These vertical walls introduce two three-dimensional effects: the secondary flow and boundary-layer growth in the spanwise direction. Using a similar numerical approach, Straub et al. (2017) compared the skin-friction drag-reduction effects by means of spanwise wall oscillations in rectangular ducts with $AR = 3$ and spanwise-periodic channels. The authors concluded that the presence of the side-walls significantly decreases the drag reduction effects of this flow control approach. Finally, Cuvier et al. (2017) documented the impact of the three-dimensional effects on a decelerating boundary layer in a wind tunnel with straight side-walls, using particle image velocimetry (PIV) measurements.

In the present work, we use a similar numerical approach to analyze the influence of the side-wall geometry on the secondary flow in an effort to provide further insight on this topic and on the mechanisms that produce the cross-flow. The preliminary results corresponding to this study were discussed by Vidal et al. (2017a). We substitute the straight side walls of the rectangular ducts by two semicircles and compare our results with the corresponding 90°-corner cases by Vinuesa et al. (2014, 2015). Since the secondary flow of Prandtl's second kind is absent in turbulent pipe flow (El Khoury et al., 2013), this geometry could potentially minimize the interaction between bursting events and homogenize the distribution of velocity streaks along the wall. However, the results discussed below show that the rounded side walls lead to a stronger mean cross-flow. Note that an increased secondary flow in square ducts with rounded corners was also previously reported by Vidal et al. (2017b).

This article is organized as follows: the numerical method used to perform the present simulations is described in Section 2. In Section 3 we compare the cross-flow distribution in ducts with rounded and straight side-walls, whereas in Section 4 we compare the turbulence statistics from both cases. In Section 5 we study the primary and secondary shear-stress components of the rotated Reynolds-stress tensor to gain more insight into the mechanisms that produce the secondary flow. Finally, the conclusions of the present work are summarized in Section 6.

Table 1
Summary of the turbulent duct cases under study. The case nomenclature consists of the aspect ratio followed by the rounding radius.

Case	AR	r	Re_b	$Re_{\tau,\,c}$	$(t_a U_b/h)$	$(t_a u_{\tau,\,c}/h)$	Grid points	$(\Delta t U_b/h)$
AR3r0	3	0	2581	179	5664	393	62×10^6	1.8×10^{-3}
AR3r1	3	1	2800	191	1350	92	57×10^6	1.0×10^{-3}
AR5r0	5	0	2592	177	4193	287	96×10^6	2.0×10^{-3}
AR5r1	5	1	2600	178	1400	96	99×10^6	1.0×10^{-3}

2. Description of the simulations

DNSs of the turbulent duct cases, which are summarized in Table 1, have been performed using the numerical code Nek5000, developed by Fischer et al. (2008) at Argonne National Laboratory. The code utilizes the spectral-element method (SEM), originally proposed by Patera (1984), to spatially discretize the incompressible Navier–Stokes equations subject to the corresponding boundary conditions. The cases under consideration have been computed using periodic boundary conditions in the homogeneous streamwise direction and no-slip boundary conditions at the walls. The SEM provides the geometrical flexibility we need to discretize the round corners using finite elements with rounded edges, while preserving the high-order accuracy of spectral methods, which is required to properly resolve the scale disparity of turbulent flows. Therefore, the mesh has been designed to satisfy the standard resolution criteria for DNS with at least three and fourteen grid-points located below $y^+ = 1$ and 10, respectively. The superscript '+' denotes inner scaling in terms of the friction velocity $u_\tau = \sqrt{\tau_w/\rho}$ (where τ_w is the wall-shear stress and ρ the fluid density). Similarly, the maximum and minimum spacing in viscous units between the spectral nodes of the largest elements is $\Delta_x^+ \simeq (2, 10)$ in the streamwise direction and $\Delta_{y,z}^+ \simeq (1, 5)$ in the vertical/spanwise directions. Note that within each element the velocity grid points follow the Gauss–Lobatto–Legendre distribution and Largrange interpolants of order N are used as basis functions. All of the cases presented in Table 1 have been computed using a polynomial order of $N = 11$ and a streamwise length of $L_x = 25h$, which according to Chin et al. (2010) is sufficiently long to obtain converged turbulence statistics in the Reynolds-number range under consideration in the present study.

The computational meshes used to simulate the ducts with semi-cylindrical side-walls are shown in Fig. 1. The top panel in this figure shows a contour plot of the instantaneous streamwise velocity distribution in the inlet plane $x = 0$, the horizontal plane $y = -0.95h$ and the centerplane $z = 0$ of the AR3r1 case. The contour plot reveals the characteristic pattern of high-speed and low-speed near-wall streaks and their length in the streamwise direction, which is indicated by the arrow in the figure. The instantaneous streamwise vortices associated with the streaks can be observed in the vector plot of the instantaneous cross-flow on middle and bottom panels, which correspond to the AR3r1 and AR5r1 cases, respectively. These panels also show a two-dimensional in-plane view of the meshes and a contour plot of the instantaneous streamwise velocity in the inlet plane. The time-step was chosen to keep the maximum Courant number below 0.35. The higher irregularity of the mesh near the semi-cylindrical side-walls required a shorter time-step to meet this condition compared to the straight side-wall cases, as shown in Table 1.

The averaging periods used to calculate the streamwise- and time-