International Journal of Heat and Fluid Flow 31 (2010) 274–283

Contents lists available at ScienceDirect

International Journal of Heat and Fluid Flow

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

Massive separation of turbulent Couette flow in a one-sided expansion channel

George K. El Khoury [a], Helge I. Andersson [b], Mustafa Barri [b], Bjørnar Pettersen [a,*]

[a] Department of Marine Technology, The Norwegian University of Science and Technology, Trondheim NO-7491, Norway
[b] Department of Energy and Process Engineering, The Norwegian University of Science and Technology, Trondheim NO-7491, Norway

ARTICLE INFO

Article history:
Received 1 September 2009
Received in revised form 20 January 2010
Accepted 23 January 2010
Available online 23 February 2010

Keywords:
Couette flow
Separation
DNS
Backward-facing step

ABSTRACT

Direct numerical simulation has been performed to study wall-driven flow over a backward-facing step at Reynolds number $Re = 5200$ based on the step height h and the upper-wall velocity U_w. The flow configuration consisted of a step with height equal to that of the upstream channel yielding an expansion ratio 2:1. Instantaneous enstrophy contours revealed the formation of Kelvin–Helmholtz instabilities downstream of the step. Intense velocity and vorticity fluctuations were generated in the shear-layer formed between the bulk flow and the massive recirculation zone in the lee of the step. Extraordinarily high turbulence levels persisted in the center region even $7.5h$ downstream of the step, i.e. where the separated shear-layer reattached to the wall. A fully redeveloped Couette flow cannot be reached in the downstream part of the channel due to the principle of mass conservation. The local wall pressure coefficient gave evidence of an adverse pressure gradient in the recovery region, where a Couette–Poiseuille flow type prevailed.

1. Introduction

Turbulent flow over a backward-facing step (BFS) is a simplified case of the general family of separated flows with widespread industrial applications. Although its geometry is simple, the flow physics is still complex. Typical prototypes of BFS flows are the boundary layer, the plane channel and the Couette flow cases, see e.g. Eaton and Johnston (1981). A common feature of these flows is the existence of a shear-layer emanating from the step corner and reattaching further downstream leading to the formation of a recirculation bubble. The presence of the internal shear-layer and the massive recirculation zone gives rise to complex flow dynamics which for instance affect the turbulence production and Reynolds stress anisotropy.

The understanding of the flow over a backward-facing step was initially acquired by experiments and two-dimensional numerical simulations. The early studies were performed by Abbot and Kline (1962) and Goldstein et al. (1970). This type of flow is characterized by the channel expansion ratio *ER* and the upstream Reynolds number. Armaly et al. (1983) conducted experiments on air flow over a backward-facing step with an expansion ratio of 2:1 and provided information on the relationship between the Reynolds number and the reattachment length X_R. The authors covered a wide range of Reynolds numbers from about 50 to 6000 and found that X_R tends to increase with the increase of Re in the laminar flow regime and decrease in the transitional one while X_R remains relatively constant in the fully-developed turbulent state. The findings of Kuehn (1980), Durst and Tropea (1981), Ötügen (1991), and Ra and Chang (1990), on the other hand, showed that the reattachment length increases with the expansion ratio.

Owing to the rapid developments in high-performance computing, three-dimensional numerical simulations of turbulent flow over a backward-facing step have been performed since the late 1980s. Friedrich and Arnal (1990) studied high Reynolds number turbulent backward-facing step flow using the large-eddy simulation (LES) technique. This technique was also used by Neto et al. (1993) who performed a numerical investigation of the coherent vortices in turbulence behind a backward-facing step. Later, Le et al. (1997) provided an extensive DNS study of turbulent boundary layer flow over a backward-facing step with an expansion ratio of 6:5 and reported a reattachment length equal to six step heights.

Since the BFS problem is not homogeneous in the streamwise direction, proper inflow conditions are to be employed in order to provide a realistic fully turbulent flow at the input. Several methods of different complexity have been used by researchers in the past years to generate suitable inflow conditions. In the boundary layer backward-facing step flow, Le et al. (1997) imposed a mean velocity profile for a flat plate turbulent boundary at the input (Spalart, 1988), whereas Meri and Wengle (2002) utilized a time-dependent inflow condition from a precursor Poiseuille flow simulation to perform DNS and LES of pressure-driven backward-facing step flow. A recent DNS study on the same flow problem was performed by Barri et al. (2010), where the authors used a cost-effective method to generate realistic dynamic inflow conditions. This technique was proposed by the same authors in an

---

* Corresponding author. Tel.: +47 735 95533.
E-mail address: bjornar.pettersen@ntnu.no (B. Pettersen).

0142-727X/$ - see front matter © 2010 Elsevier Inc. All rights reserved.
doi:10.1016/j.ijheatfluidflow.2010.01.008

earlier study on numerical simulation of plane channel flow (see Barri et al., 2009) and consists of recycling finite-length time series (t_s) of instantaneous velocity planes at the input. These profiles were taken from a precursor simulation and a physical constraint was introduced on t_s to be of order of the large-eddy-turnover-time. The authors kept the inflow time series discontinuous and showed that this discontinuity in the inflow signal vanishes due to the nonlinear interactions of the turbulent flow.

Due to the principle of mass conservation, the Reynolds number in pressure-driven plane channel flow with a sudden one-sided expansion remains the same downstream of the step as in the upstream part of the channel. In a BFS Couette flow, on the other hand, the Reynolds number becomes higher downstream of the step. It is well known that the shear-driven turbulent Couette flows (see e.g. Bech et al., 1995) exhibit a number of characteristic features which make them distinguishingly different from the pressure-driven Poiseuille flow, notably the monotonically increasing mean velocity profile. The only investigation of BFS Couette flow we are aware of is the recent experimental study by Morinishi (2007). He considered a configuration with the step height h equal to half of the upstream channel height, i.e. with an expansion ratio 3:2. The Reynolds number based on the wall-friction velocity at the input was fixed to 300. This configuration (i.e. fixed ER and $Re_{\tau i}$) allowed the author to investigate the effect of the non-dimensional upstream pressure gradient β_o. By varying β_o Morinishi (2007) set up conditions for the following fully-developed upstream flows: the pure Poiseuille flow, the mixed Couette–Poiseuille flow and the pure Couette flow. For the three cases, the reattachment lengths were almost identical and about $6.5h$ and $2h$ for the primary and secondary recirculation regions, respectively.

In the present study we perform direct numerical simulation (DNS) of turbulent Couette flow over a BFS. This will enable us to gather accurate mean flow and turbulence statistics throughout the flow domain, as well as to explore in detail the instantaneous vortex topology in the shear-layer and the recirculation bubble as well as in the re-development zone. We intentionally considered a BFS configuration, where the flow upstream of the step is the same as that studied by Bech et al. (1995), i.e. a fully-developed turbulent Couette flow.

2. Flow configuration and governing equations

Fig. 1 shows a schematic view of the Couette backward-facing step flow which is composed of a step of height h and an upper-wall moving with velocity U_w. Of particular relevance in backward-facing step flows is the expansion ratio ER. This dimensionless parameter is defined as the ratio between the downstream and upstream channel heights, i.e. $ER = H/(H-h)$. In the present study we consider a flow configuration, where the step height is equal to that of the upstream channel, i.e. $H = 2h$. This gives an expansion ratio of 2:1.

The governing equations are the time-dependent, incompressible Navier–Stokes equations for a viscous fluid expressed in non-dimensional form:

$$\nabla \cdot \mathbf{u} = 0, \tag{1}$$

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla)\mathbf{u} = -\nabla p + \frac{1}{Re}\nabla^2 \mathbf{u}. \tag{2}$$

Here, the variables have been non-dimensionalized by h and U_w and the Reynolds number based on the step height and upper-wall velocity, $Re = U_w h/\nu$, is 5200.

3. Numerical approach

The computational domain has a length of $L_x = 39h$ in the streamwise x-direction including an inlet section $L_s = 15h$, $H = 2h$ in the wall-normal y-direction, and $L_z = 9.43h$ in the spanwise z-direction. A total of $(672 \times 384 \times 192)$ grid points are used in x, y and z, respectively. In order to adequately resolve the turbulence scales in the separation region and the vicinity of the walls, a non-uniform mesh distribution is used in the streamwise and wall-normal directions. Thus for a viscous length scale of $l_i = \nu/u_{\tau i}$, based on the wall-friction velocity at the input $u_{\tau i} = 0.032U_w$, the first grid point next to the walls is at $y^+ \approx 0.083$ while the largest grid spacing is about $\Delta y^+ \approx 2.4$ (measured in wall units). For the streamwise direction, the minimum grid spacing $\Delta x^+ \approx 4.8$ is at the step corner and maximum at the beginning and the end of the domain with $\Delta x^+ \approx 14.8$. A uniform mesh is used in the spanwise z-direction with $\Delta z^+ \approx 8.2$. The grid specification in the inlet section is given in Table 1 together with that of Bech et al. (1995).

No-slip boundary conditions are imposed on the solid surfaces in the domain. The flow in the spanwise direction is assumed to be statistically homogeneous and periodic boundary conditions are imposed. A realistic fully turbulent flow is generated at the input by recycling finite-length time series of the instantaneous velocity planes. This technique was first used by Barri et al. (2009) in a numerical simulation of plane channel flow. The length of the time series for the current simulation was $33h/U_w$. This is consistent with the recommendation of Barri et al. (2009). For a plane Poiseuille flow, they demonstrated that the potential periodicity introduced by the recycling of a finite-length time series vanished when the duration of the time series equaled the large-eddy-turnover-time h/u_τ. The imposition of dynamic Dirichlet conditions at the inflow implies that the flow rate is fixed to that of the precursor simulation. The latter was that of plane Couette flow at exactly the same Reynolds number as in the extensive investigation by Bech et al. (1995). As an outflow condition, we solve the convective equation $\partial\mathbf{u}/\partial t + U_c \partial\mathbf{u}/\partial x = 0$ at the exit plane and set the total normal stress acting on it to zero by means of a pressure boundary condition $-p + 2\mu\partial u/\partial x = 0$. The convective boundary condition was used in previous numerical simulations by Lowery and Reynolds (1986) for a mixing-layer and Le et al. (1997) and Barri et al. (2010) for turbulent flow over a backward-facing step and is considered suitable for vortical structures moving out of the domain.

The DNS code used to numerically solve the governing Eqs. (1) and (2) is MGLET (see Manhart, 2004). MGLET is a finite-volume code in which the Navier–Stokes equations are discretized on a

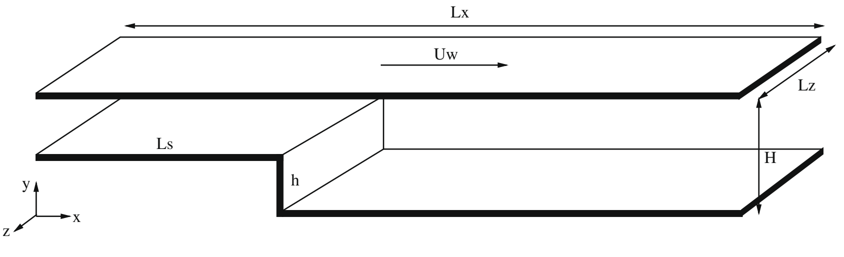

Fig. 1. Flow configuration and coordinate system (not to scale).