International Journal of Heat and Fluid Flow 56 (2015) 137–151

Contents lists available at ScienceDirect

International Journal of Heat and Fluid Flow

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

Large Eddy Simulations of convergent–divergent channel flows at moderate Reynolds numbers

Luiz A.C.A. Schiavo [a], Antonio B. Jesus [b], João L.F. Azevedo [c,*], William R. Wolf [a]

[a] Universidade Estadual de Campinas, 13083-970 Campinas, SP, Brazil
[b] Instituto Tecnológico de Aeronáutica, 12228-900 São José dos Campos, SP, Brazil
[c] Instituto de Aeronáutica e Espaço, 12228-903 São José dos Campos, SP, Brazil

ARTICLE INFO

Article history:
Received 22 January 2015
Received in revised form 15 May 2015
Accepted 13 July 2015
Available online 1 August 2015

Keywords:
LES
Turbulent flow
Adverse pressure gradient
Streak instability
Channel flow

ABSTRACT

The paper is focused on the study of fully turbulent channel flows, using Large Eddy Simulations (LES), in order to address the effects of adverse pressure gradient regions. Analyses of the effects of streak instabilities, which have been shown to be relevant in such regions, are extended to moderate Reynolds numbers. The work considers two different channel geometries in order to further separate influences from wall curvature, flow separation and adverse pressure gradients. Turbulent kinetic energy and Reynolds stress budgets are investigated at separation and re-attachment points. The numerical approach used in the present work is based on the incompressible Navier–Stokes equations, which are solved by a pseudo-spectral methodology for structured grids. Wall-resolved LES calculations are performed using the WALE subgrid scale model. The study shows that the streak instability mechanism persists at higher Reynolds numbers with and without wall curvature in the adverse pressure gradient regions. Moreover, the observed effects are also present regardless of the existence of flow separation regions. Finally, the study of turbulent kinetic energy budgets indicates that, independently of the flow condition, there are well-defined patterns for such turbulent properties at separation and re-attachment points.

1. Introduction

Turbulence is observed in several flows present in nature and in industrial applications. It is characterized by a large disparity of spatial and temporal scales. In order to accurately resolve these scales, direct numerical simulation (DNS) and Large Eddy Simulation (LES) have been used to study the physical phenomena associated with transition and turbulence. DNS resolves all ranges of scales in a turbulent flow. In LES, the larger scales, which are mostly affected by the topology of the flow, are effectively resolved. The high wavenumber turbulent scales are modeled by a subgrid model, since these smaller scales are less energetic and their statistics have a more universal character. One should expect lower requirements in terms of mesh resolution and time step restrictions in LES calculations, once the small turbulent scales are modeled. Therefore, LES has lower computational costs, when compared to DNS, although it is still able to capture the main unsteady features in turbulent flows.

In industrial applications, the accurate prediction of separation and re-attachment points in turbulent flows is an important factor in the design of aircraft and turbo-machinery. Adverse pressure gradient (APG) regions change the shear stress distributions and this can often impact the dynamics of the flow, leading to separation. Currently, the application of Computational Fluid Dynamics (CFD) tools, which solve the Reynolds-averaged Navier–Stokes (RANS) equations, is common practice in industry. However, RANS turbulence models are still not capable of adequately predicting the flow features in adverse pressure gradient (APG) regions in spite of continuous improvements to such models.

The study presented by Jesus et al. (2014) indicates that two-equation eddy-viscosity models tend to either under-predict or over-predict flow separation. Reynolds stress transport models show improved results, especially with regard to the extension of the separated flow region. Nevertheless, no RANS model is fully accurate in skin friction calculations (Jesus et al., 2014; Jeyapaul et al., 2013) for APG regions. The issues that prevent RANS models from accurately predicting the flow physics along APG regions have been associated with the inability of such models in correctly capturing flow separation and re-attachment locations (Jeyapaul et al., 2013; Menter et al., 2003). However, the present results suggest that the behavior of the skin friction coefficient in APG regions

* Corresponding author.
E-mail addresses: augustoschiavo@gmail.com (L.A.C.A. Schiavo), antoniobjota@gmail.com (A.B. Jesus), joaoluiz.azevedo@gmail.com (J.L.F. Azevedo), wolf@fem.unicamp.br (W.R. Wolf).

http://dx.doi.org/10.1016/j.ijheatfluidflow.2015.07.006
0142-727X/

of smooth bumps does not depend on flow separation. In other words, the behavior of attached and mildly separated flows is very similar, indicating that the prevailing physical mechanism is mostly associated to the adverse pressure gradient condition.

In the present work, wall-resolved LES of incompressible turbulent flows are presented for convergent–divergent channels with adverse pressure gradient regions. Simulations are performed for Reynolds numbers $Re_\tau = 617$ and 950. The current LES results are compared to DNS performed by Laval et al. (2012) and Marquillie et al. (2011) for the lower Reynolds number case here investigated, $Re_\tau = 617$, and excellent agreement is observed. The work considers two different channel geometries in order to separate influences from wall curvature, flow separation and adverse pressure gradients on the physical mechanisms present at such channel flows. The effects of APG regions are evaluated through the analysis of friction coefficient distributions and turbulent kinetic energy (TKE) budgets, including the role of production, transport and dissipation of turbulence. Furthermore, a discussion of the streak bursting instability phenomenon, originally observed by Marquillie et al. (2008), is also presented. Analyses of the effects of streak instabilities are extended to moderate Reynolds numbers. The study shows that the streak instability mechanism persists at higher Reynolds numbers and regardless of the existence of flow separation regions. Finally, turbulent kinetic energy and Reynolds stress budgets are investigated at separation and re-attachment points. Such study indicates that, independently of the flow condition, there are well-defined patterns for these turbulent properties at separation and re-attachment points.

2. Numerical formulation

LES computations are performed using the MFLOPS3D code (Marquillie and Ehrenstein, 2003), developed at Laboratoire de Mécanique de Lille, in Lille, France. This is a semi-spectral code developed for the study of boundary layers and channel flows with non-regular cross sections. The code has been used for computations of laminar flow instabilities (Marquillie and Ehrenstein, 2003) and DNS of turbulent channel flows around two-dimensional bumps (Marquillie et al., 2008, 2011). The LES formulation implemented in the code solves the dimensionless filtered incompressible Navier–Stokes equations, which can be written as

$$\begin{aligned} &\text{Mass conservation}: \ \nabla \cdot \vec{\overline{u}} = 0, \\ &\text{Momentum}: \ \frac{\partial \vec{\overline{u}}}{\partial t} + \left(\vec{\overline{u}} \cdot \nabla\right)\vec{\overline{u}} = -\nabla \overline{p} + \frac{1}{Re}\nabla^2 \vec{\overline{u}} - \nabla \cdot \tau^{sgs}, \\ &\text{Poisson for pressure}: \ \nabla^2 \overline{p} = -\nabla \cdot \left(\vec{\overline{u}} \cdot \nabla\right)\vec{\overline{u}} - \nabla^2 \tau^{sgs}. \end{aligned} \tag{1}$$

In the set of equations above, $\vec{u}$ represents the velocity vector, p is the pressure and τ^{sgs} is the subgrid scale stress tensor. The bars, $\overline{(\;)}$, indicate filtered variables and the subgrid scale tensor in indicial notation is defined as

$$\tau_{ij}^{sgs} = \overline{u_i u_j} - \bar{u}_i \bar{u}_j. \tag{2}$$

One should mention that throughout the text, the bars will be omitted to simplify the notation. In the present paper, subgrid scale terms are evaluated using the WALE model (Nicoud and Ducros, 1999), which is adequate for wall bounded flow computations as it was conceived to recover the correct eddy-viscosity near-wall scaling without the need for an explicit damping. The dimensionless quantities are given by

$$\vec{x''} = \frac{\underline{\vec{x}}}{\delta}, \quad \vec{u} = \frac{\underline{\vec{u}}}{U_c}, \quad t = \frac{\underline{t} U_c}{\delta}, \quad p = \frac{\underline{p}}{\rho U_c^2}, \quad Re = \frac{\delta U_c}{\underline{\nu}}, \tag{3}$$

where the underbar, $\underline{(\;)}$, indicates a variable with dimension, δ is the half height of the channel and U_c is the mean velocity at the inlet centerline. For the configurations of interest in the present paper, x'' represents the streamwise direction, y'' is the transversal direction and z'' is the spanwise coordinate. It is further convenient, for the purpose of implementing the numerical methodology, to normalize the spatial coordinates as indicated in Appendix A.

The MFLOPS3D code performs a coordinate transformation along the y direction to allow the analysis of geometries with non-regular cross sections. The mapping coordinates are introduced into the formulation by the splitting of the gradient, divergent and Laplacian operators. The new operators are given by

$$\nabla = \nabla_\eta + G_\eta \tag{4}$$

and

$$\nabla^2 = \Delta_\eta + L_\eta. \tag{5}$$

The ∇_η operator represents the Cartesian components of the transformed gradient operator and G_η includes the terms of the gradient operator involving the cross-section profile. Similarly, Δ_η contains the Cartesian terms of the transformed Laplacian operator and L_η groups the terms associated to the cross-section profile and its derivatives. With the previous operators, the Navier–Stokes equations are re-written as

$$\nabla_\eta \cdot \vec{u} + G_\eta \cdot \vec{u} = 0, \tag{6}$$

$$\begin{aligned} \frac{\partial \vec{u}}{\partial t} + (\vec{u} \cdot \nabla_\eta)\vec{u} + (\vec{u} \cdot G_\eta)\vec{u} = -\nabla_\eta p - G_\eta p + \frac{1}{Re}\Delta_\eta \vec{u} + \frac{1}{Re} L_\eta \vec{u} \\ - \nabla_\eta \cdot \tau^{sgs} - G_\eta \cdot \tau^{sgs}, \end{aligned} \tag{7}$$

$$\Delta_\eta p + L_\eta p = J(u, v, w) - \nabla \cdot \left(\nabla_\eta \cdot \tau^{sgs} - G_\eta \cdot \tau^{sgs}\right). \tag{8}$$

In the above equations, $J(u, v, w)$ includes the effects of all nonlinear terms appearing in the transformed equations. All the details concerning the mapping coordinates and transformed operators can be found in Appendix A. Eqs. (6)–(8) are the modified mass conservation, momentum and Poisson equations, which are solved as part of the numerical procedure to be described in the next paragraphs.

The first derivatives in the streamwise direction are discretized using an 8th-order centered finite difference scheme. The second derivatives in the streamwise direction are discretized using a 4th-order centered finite difference scheme. Chebyshev polynomials, collocated in Chebyshev–Lobatto points (Canuto et al., 1988), are employed for all derivatives in the y direction. Fourier transforms are performed in the spanwise direction, which is assumed to be periodic. The conventional 3/2 rule (Canuto et al., 1988) is employed to remove aliasing errors of the discrete Fourier transform in the nonlinear terms.

Time integration is performed using an implicit 2nd-order backward Euler method for the terms containing the Cartesian components of the Laplacian operator. An explicit 2nd-order Adams–Bashforth method is used for all the other terms, including the subgrid scale tensor. Pressure–velocity coupling is achieved by a fractional-step method which performs an iterative process. In this process, the solution of the momentum equations yields an intermediate velocity field, whereas the solution of the pressure Poisson equation determines an intermediate pressure (Karniadakis et al., 1991). Afterwards, iterations based on the continuity equation are used in order to obtain a pressure correction that produces a divergent-free velocity field. Calculations are performed in Fourier space, and each Fourier mode is solved independently using parallel computations. The nonlinear and subgrid tensor terms are computed in physical space. The computational process is parallelized by mesh partition and the message passing