TURBULENT SEPARATION IN LOWER CURVED WALL CHANNELS

J.-P. Mollicone[1], F. Battista[1] & C.M. Casciola[1]
[1]*Department of Mechanical and Aerospace Engineering, University La Sapienza, via Eudossiana 18, 00184, Rome, Italy*

Abstract Turbulent boundary layer separation in channels with a lower curved wall is studied using direct numerical simulations (DNS). Turbulence dynamics are studied through classical statistical tools such as the turbulent kinetic energy budget for varying lower curved wall dimensions. The geometry features are expected to have a significant effect on the fluid flow structures and the characteristic scales of separation. The separation bubble behind the bump is studied in terms of its size, turbulent kinetic energy production mechanisms and transfer and scale-by-scale energy budget. New innovative data-analysis techniques will be used based on the generalisation of the Kolmogorov equation to anisotropic and spatially non-homogeneous flow configurations.

INTRODUCTION

Separation in turbulent flows has been significantly studied [8] as a phenomenon directly relevant for many engineering fields such as vehicle aerodynamics, fluid-structure interactions, mixing, transport of particles or pollutants and many more small to large-scale applications. From a more fundamental point of view, experiments and numerical simulations [5, 4] have been carried out to study turbulent boundary layers often focusing on how to control boundary layer separation [6, 3]. Turbulence dynamics have been thoroughly addressed for idealised flow conditions, i.e. homogeneous isotropic turbulence, shear turbulent flow and plain channel flow. Our study, using direct numerical simulations (DNS), is intended to contribute to the gap between idealised conditions and actual flow geometries where the effects of wall curvature and the presence of bluff bodies immediately generates flow separation, originates recirculating regions in the bulk of the flow and produces turbulent wakes behind the bluff body.

SIMULATIONS

Figure 1 is a contour plot of velocity magnitude which shows the main simulation setup for the DNS. The domain has dimensions $(L_x \times L_y \times L_z) = (24 \times 2 \times 10) \times h$ where L_i is the length in the i^{th} direction and $h = 1$ is half the channel height. Flow is from left to right with periodic boundary conditions in both stream-wise and span-wise directions with no slip boundary conditions at the top and bottom walls.

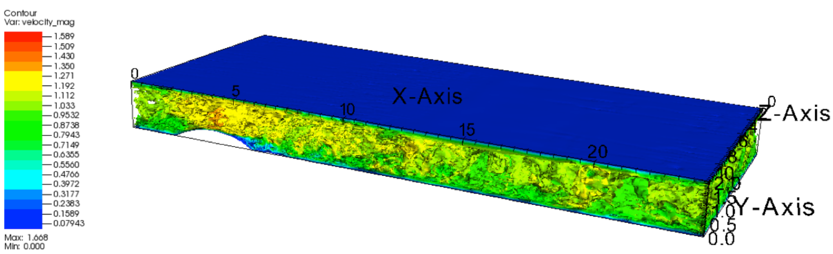

Figure 1. Three dimensional velocity magnitude contour plot of all the computational domain.

The CFD solver used is Nek5000 [7] which is based on the spectral element method (SEM) [9]. The simulations require well-refined grids to capture the turbulence dynamics at all physically relevant length-scales especially in the near wall region therefore entailing large memory and computational power requirements. The domain consists of over 600 million grid points with the Reynolds number based on half the channel height, $Re_h = 10000$, and a corresponding $Re_\tau = 430$.

RESULTS AND CURRENT WORK

The domain in Figure 1 was simulated for varying lower curved wall (or bumps) dimensions. By changing the bump's aspect ratio or height, three different geometries where used, namely the Thinner, Default and Higher cases as shown in Figure 2. The figure shows the turbulent kinetic energy (TKE), $k = 1/2\langle u_i' u_i' \rangle$ (with u_i' being the ith component of velocity fluctuation), contour plots in the left panel together with the respective line plots over a vertical line extended from the end of the bump to the top wall for the three cases in the right panel.

The contour plots show a clear flow separation behind the bump. Fluid flow is at the onset of separation at the top wall due to the bump restricting the flow but this is mitigated by the incoming turbulent flow. Even though the plots for the

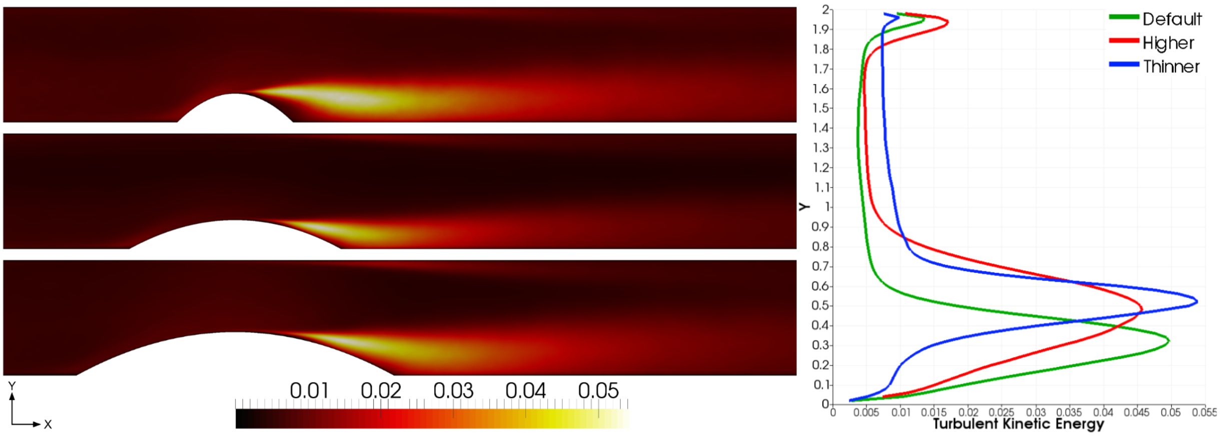

Figure 2. Left panel: Detail of geometry variations with averaged turbulent kinetic energy contour plots. Top: Thinner geometry, Middle: Default geometry, Bottom: Higher geometry. Right panel: Averaged turbulent kinetic energy plots over a vertical line extended from the end of the bump to the top wall for each geometry. Thinner in blue, Default in green and Higher in red.

three cases are similar, the contour plot for the Thinner case shows that a larger recirculation bubble is present behind the bump. This is also seen in the upward shift of the line plot of TKE for the Thinner case where the TKE starts increasing at a higher y-value with respect to the other two cases. The TKE in the Higher and Default line plot cases drastically drops at around 0.75 and 0.5, which are the heights of the bump for each case respectively. The latter two cases tend to show similar behaviour which scales with the height of the bump when keeping the aspect ratio constant.

The current work is based on extending a generalised form of the Karman-Howarth equation [1] to our anisotropic and non-homogeneous conditions. The Karman-Howarth equation is a powerful tool which follows directly from the equations of motions and characterises all the dynamical effects occurring at each scale of the turbulent flow [2]. The budget successfully accounts for non-homogeneous effects, i.e. spatial energy fluxes which arise due to the presence of the bump. Such tool allows to characterise the scale-by-scale mechanisms of turbulent kinetic energy production, the energy transfer across the scales and the spatial energy flux in different flow regions.

References

[1] Cimarelli A., E. De Angelis, and Casciola C. M. Paths of energy in turbulent channel flows. *Journal of Fluid Mechanics*, **715**, 2013.

[2] Cambon C. and Gracia B-J. The role of directionality on the structure and dynamics of strongly anisotropic turbulent flows. *Journal of Turbulence*, 2013.

[3] Bai H. L. et al. Active control of a turbulent boundary layer based on local surface perturbation. *Journal of Fluid Mechanics*, 2014.

[4] Harun Z. et al. Pressure gradient effects on the large-scale structure of turbulent boundary layers. *Journal of Fluid Mechanics*, 2013.

[5] Schlatter P. et al. Turbulent boundary layers up to re= 2500 studied through simulation and experiment. *Physics of fluids*, **21**(051702), 2009.

[6] Marusic I., Talluru K.M., and Hutchins N. Controlling the large- scale motions in a turbulent boundary layer. *Fluid-Structure- Sound Interactions and Control*, 2014.

[7] Fischer P., Lottes J.W., and Kerkemeier S.G. Nek5000 - open source spectral element cfd solver - http://nek5000.mcs.anl.gov, 2008.

[8] Simpson R.L. Turbulent boundary-layer separation. *Annual Review of Fluid Mechanics*, 1989.

[9] Patera A. T. A spectral element method for fluid dynamics. *Journal of Computational Physics*, **54**:468–488, 1984.