NUMERICAL STUDY OF WALL-BOUNDED TURBULENCE OVER D-TYPE ROUGHNESS

Yves DUBIEF, Pierre COMTE and Marcel LESIEUR

Laboratoire des Ecoulements Géophysiques et Industriels
BP53, 38041 Grenoble-Cedex, France

October 1, 1998

d-type roughness is commonly used to described smooth surfaces with series of spanwise small cavities. As defined by Perry *et al.* (1969), d refers to the outer lengthscale which turns out to be the most significant lengthscale for d-type rough wall. The possibility of drag reduction has been suggested by Choi & Fujisawa (1993). An experiment on a turbulent boundary layer passing over a single groove, carried out by Pearson *et al.* (1997), showed that downstream of the groove, the skin friction is fairly high at the stagnation point created by the edge. This overshoot in the drag compared to the smooth wall case is followed by a sudden undershoot and a slow relaxation towards the smooth wall value. Similar features have recently been observed by Ching & Parsons (1998) for sparse d-type roughness. They investigated different configurations of cavity spacing s to cavity width w, in the range $s/w = 10$ to 40. The distribution of the skin-friction along the streamwise direction is periodic and resembles to the data collected by Pearson *et al.*,at least for the largest cavity spacing. Their work reports a slight increase of the drag over the rough wall compared to the smooth case. In spite of the sharp rise of the wall-shear stress at the downstream edge of each groove, it is remarkable that the overall drag remains close to the drag over a smooth plate. One of the most interesting features of this type of flow is the communication between the recirculating flow within the groove and the boundary layer flow. Flow visualisations performed by Pearson *et al.* (1997) allowed to isolate intermittent events such as ejections of fluid inwards or outwards of the cavity. The latter authors suggested to relate these events to quasi-streamwise vortices passing over the groove. In the light of these experimental results, d-type roughness proves to be an interesting device to study wall manipulation and near wall structures.

At the previous conference, preliminary results of a numerical simulation of a boundary layer passing over a groove were presented (Dubief & Comte, 1997). The aim of this paper is to further discuss the effect of a groove or a series of grooves on near wall turbulence. The first simulation performed in 1997 concerned a cavity embedded in a flat plate. A fully turbulent boundary layer is simulated (using Lund *et al.*, 1996's method) upstream of the groove whose dimensions are of the order of the boundary layer thickness δ_0. A second calculation is currently underway to investigate the effect of a much smaller groove on a periodic channel flow. The groove width w to channel half-width h ratio is approximately 0.3. Thanks to the periodicity assumption, the flow on the lower wall of the channel develops over a sparse d-type rough wall whose cavity spacing is $20w$. Each simulation are carried out using a compressible code which is 2nd order accurate in time and 4th order in space. The code is similar to that used by Ducros *et al.* (1996) in a finite-volume formulation. Both simulations are performed at low Reynolds number, $Re_{\delta_0} = 5100$ for the boundary layer and $Re_h = 3750$ for the channel. The first calculation is a large-eddy simulation using the filtered-structure function model (Ducros *et al.* 1996). The channel flow calculation is a direct numerical simulation.

The manipulation of a turbulent boundary layer by a fairly large cavity ($w/\delta_0 \simeq 1$) is found to significantly affect the distribution of the skin-friction coefficient as well as near wall turbulent structures. The longitudinal profile of the wall shear stress (fig. 1) exhibits similar behaviour as the one observed by Pearson *et al.* (1997) for a much smaller cavity ($w/\delta_0 \simeq 0.17$ in Pearson *et al.*'s experiment) and described in the introduction of this abstract. However the magnitude of the drag reduction is lower than that found by Pearson *et al.*. This discrepancy might be explained by the cavity width used in the present simulation. Turbulence statistics (not shown here) are affected by the groove up to $y^+ \simeq 100$. The streaky structure of the flow (see figure 2) is intensified by the wall singularity. Streak dimensions are reduced in the spanwise and streamwise direction. The flow is less anisotropic downstream of the groove compared to canonical turbulent boundary layers (not shown here). Streamwise vortical structures in the near-wall region are tilted upwards. The communication between the recirculating flow inside the cavity and the boundary layer is obviously significant.

The latter feature of the first simulation is investigated in a sligthly different perspective using a smaller cavity. The configuration of the flow consists of a periodic channel flow. The dimensions of the channel in the streamwise, wall-normal and spanwise directions are 1000, 320 and 500 wall units, respectively. A single cavity whose width is $0.3125h$, is embbeded in the lower wall. This simulation uses a cavity width which is close to those chosen in the experiments of Pearson *et al.* (1997) and Ching & Parsons (1998). Statistical convergence has not been achieved yet but the trend clearly indicates a good agreement with experimental data (not shown here). Preliminary results show vertical motions of fluid inwards and outwards of the groove. Figure 3 plots the intensity of instantaneous vertical velocity at the top of the groove. Ejections occur over the full width of the cavity, whereas injections seems to be located near the downstream edge. Figure 4 shows vorticity contours in the streamwise-vertical plane. This plane corresponds to the valley of v intensity on the left of figure 3. These plots seem to correlate injection of fluid inside the cavity with intense vortical structures passing over the groove. It should be noted that those features are in agreement with flow visualisations carried out by Pearson *et al.* (1997) and Djenidi *et al.* (1996). A full investigation of the interaction between near-wall structures and the recirculating flow will be presented at the conference. The guideline of our work will be to track quasi-streamwise vortices above the groove and try to understand the mechanism of drag reduction immediatly downstream of the groove.

REFERENCES

CHOI, K. S. and FUJISAWA, N., "Possibility of drag reduction using d-type roughness", In K. K. Prasad (ed.) *Further Developments in Turbulence Management*, Kluwer Academic Publishers, 315-324, 1993.

CHING, C.Y. and PARSONS, B.L., "Drag Characteristics of a turbulent boundary layer over a flat plate with transverse square grooves", *To be published in Expts. in Fluids*, 1998.

DJENIDI, L., ANSELMET, F. and ANTONIA, R.A., "LDA measurements in a turbulent boundary layer over a *d*-type rough wall", *Expts. in Fluids* **16** 323-329, 1994.

DUBIEF, Y. and COMTE, P., "Large-eddy simulation of a boundary layer passing over a groove", *Proceedings of Turbulent Shear Flows 11*, Grenoble, 1.1-1.6, 1997.

DUCROS, F., COMTE, P. and LESIEUR, M., "Large-eddy simulation of transition to turbulence in a boundary layer developing spatially over a flate plate", *J. Fluid Mech.* **326** 1-36, 1996.

LUND, T.S., WU, X. and SQUIRES, K.D., "On the generation of turbulent inflow conditions for boundary layer simulations" *Ann. Briefs of the Center For Turbulent Research* 287-295, 1996.

PEARSON, B.R., ELAVARASAN, R. and ANTONIA, R.A., "The Response of a Turbulent Boundary Layer to a Square Groove" *To be published in J. Fluids. Eng.*, 1997.

PERRY, A. E., SCHOFIELD, W. H. and JOUBERT, P. N. "Rough wall turbulent boundary layers" *J. Fluid Mech.* 383-413, 1969.

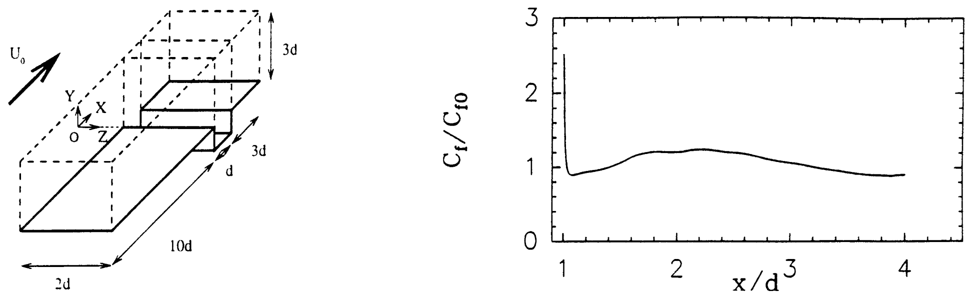

Figure 1: Left: flow configuration of the first simulation. Right: longitudinal evolution of the skin friction coefficient normalised by its smooth wall value.

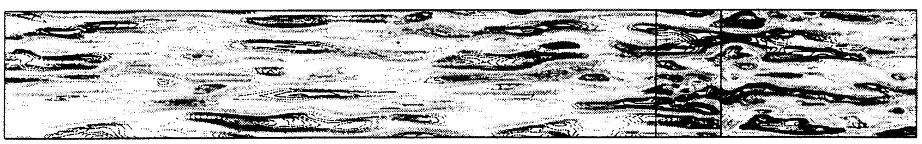

Figure 2: Contours of instantaneous streamwise velocity fluctuations in the (x, y) plane located at $y^+ = 8$.

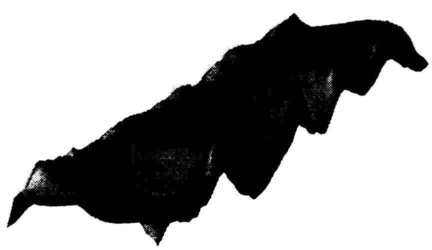

Figure 3: 3D plot of the intensity of the instantaneous vertical velocity at the top of the cavity, for the channel flow simulation.

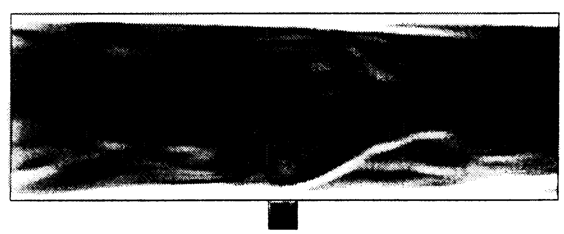

Figure 4: Vorticity contours in the streamwise-vertical plane. The flow goes from left to right. High vorticity region are represented in white