SPANWISE PERTURBATION OF A TURBULENT CHANNEL FLOW

Flageul Cédric[1], Benhamadouche Sofiane [1], Lamballais Éric [2], Laurence Dominique [1]
[1]*EDF R&D, 6 Quai Watier, 78401 Chatou, France*
[2]*Laboratoire d'Études Aerodynamiques - Universite de Poitiers, 86022 Poitiers, France*

Abstract
During an Emergency Core Cooling (ECC) in a Pressurized Water Reactor (PWR), cold water is injected in the cold leg of the Reactor Pressure Vessel (RPV) which may lead to large stresses due to thermal loads. The overall objective of the present work is to evaluate the accuracy of numerical predictions of the heat flux into a RPV wall. As the wall heat transfer is highly sensitive to the 3D turbulent structures in the boundary layers, modelling it by wall-functions in Reynolds Averaged Navier-Stokes (RANS) or in Large Eddy Simulation (LES) is disputable. These approaches are based on correlations derived from steady-state mean-flow equations which do not take into account meandering cascades and coherent structures. In order to study the fine 2D coherent structures, a simplified test-case is manufactured, yet keeping the mean flow-features of the industrial configuration. Direct Numerical Simulations (DNS) and Large Eddy Simulations (LES) of a channel flow under external forcing depending on the span-wise direction will be performed.

TEST-CASE

A twin wall-jet immersed in a channel flow is generated by a modulated forcing term, in addition to the conventional pressure gradient used in periodic inflow-outflow DNS. The flow is driven by an external forcing $\overrightarrow{f}$ which appears in Navier-Stokes equations (1). One assumes this forcing to be in the x direction and to depends only on the span-wise coordinate z. When the forcing is independent of z ($f_k = 0$ for $k \neq 0$), the Reynolds number based on wall friction velocity is $Re_\tau = 180$.

$$D_t \overrightarrow{u} = -\frac{\nabla p}{\rho} + \nu \Delta \overrightarrow{u} + f(z)\overrightarrow{e_x} \text{ with } f(z) = \sum_{k=0}^{+\infty} f_k cos\left(\frac{2\pi k z}{L_z}\right) \tag{1}$$

NUMERICAL SIMULATIONS

LES is performed with EDF in-house open source CFD tool *Code Saturne*, Archambeau et al. [4]. No-slip boundary conditions on the walls and periodicity in the stream-wise and the span-wise directions are applied. A standard Smagorinsky model with $C_S = 0.065$ combined to a Van-Driest damping function is used. The initial condition is a standard fully developed velocity profile combined with the synthetic eddy method of Jarrin et al. [3]. The computational domain size is $(12.8 \times 2 \times 12.8)$ and the grid is cartesian, stretched in the y-direction and discretized with $(128 \times 64 \times 128)$ cells. The averaged non-dimensional cell sizes in the stream-wise (x) and span-wise (z) directions are equal to 18 wall units, respectively, whereas it goes from 1.13 to 8.48 in the wall normal direction. The maximum value of the instantaneous non-dimensional distance to the wall reached during the simulation is 2.2, which is reasonable. The high-resolution DNS will be performed using *Incompact3d* code, Laizet et al. [1] in which the pressure equation is solved in the spectral space while the momentum equations are solved in the physical space using finite differences (sixth order compact scheme).

PRELIMINARY RESULTS

Figue 1 shows the forcing imposed to the stream-wise component. Locations F_{min}, F_{max} and $F_{1/2}$, which correspond to $z = 3.2$, 6.4 and 4.8 are located at the minimum, the maximum and the unit amplitudes, respectively. All the mean quantities $< . >$ are averaged in time and in the stream-wise direction x. Only preliminary LES results are shown herein. Figure 2 shows the velocity vectors in (y,z) plane based on $< V_y >$ and $< V_z >$ mean velocity components. Although the statistics are either not fully converged or the stream-wise length of the channel is not long enough (tests are still performed concerning this parameter), one can observe opposite rolls (2 per period) with a maximum velocity magnitude of the order of 0.5 % of the bulk velocity. The fluid in a low-forcing area tends to move from the center of the channel to the wall. In the near wall region, it goes back to the high-forcing area, staying relatively close to the wall then goes to the middle of the channel where the fluid drifts back to the low-forcing area. In-depth investigations of the statistical properties of the flow, including an analysis of the balance equations, are planned in order to explain this mean behavior.
Figure 3 shows the mean streamwise velocity at 3 different z locations. The results are compared to DNS ones from Kim et al. [2] corresponding to uniform forcing (without the harmonic part). Component $< V_x >$ is slightly depending on z : the higher the forcing, the higher the velocity. The difference between the maximum axial mean velocities corresponding to F_{min} and F_{max} is of the order of harmonic amplitude (around 15 %). The flow dependency on z leads to a variable

Reynolds number based on the wall-friction velocity (Re_τ), oscillating between 172 and 188.

Figures 4 to 6 show the diagonal components of the Reynolds stress tensor. The results are qualitatively similar to DNS ones. The dependendy on z of these components is similar to that of $< V_x >$ component. In addition, the peak of $< w'w' >$ is around $y^+ = 30$ for the present simulation while it is rather around 40 in the reference DNS. Similarly, the peak of $< v'v' >$ is slightly shifted, but in this case, to the right (farther from the wall).

In addition to what has been introduced in the present abstract, the final paper will include DNS results using *Incompact3d*. An in-depth analysis will be carried out concerning the flow structure and in particular the structures of the eddies in the near-wall region. Indeed, these latters are a key point in understanding the heat flux penetration in the wall as they rule the temperature patterns which one may observe.

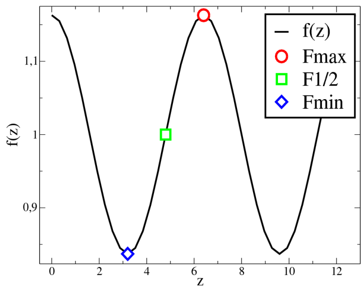

Figure 1 : External forcing

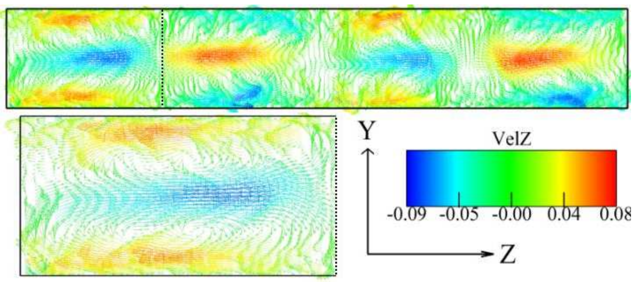

Figure 2 : Mean tangential velocity vector in a cross-section colored by $< Vz >$

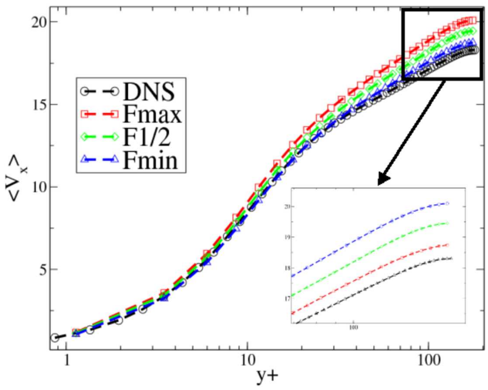

Figure 3 : Mean axial velocity at different z locations.

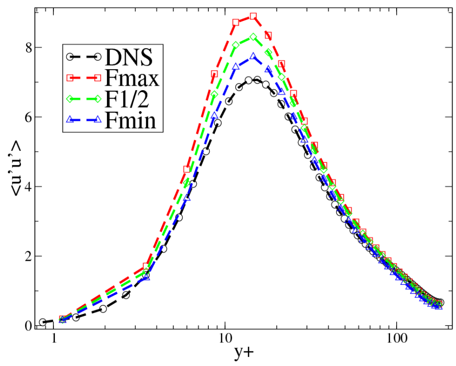

Figure 4 : $< u'u' >$ at different z locations

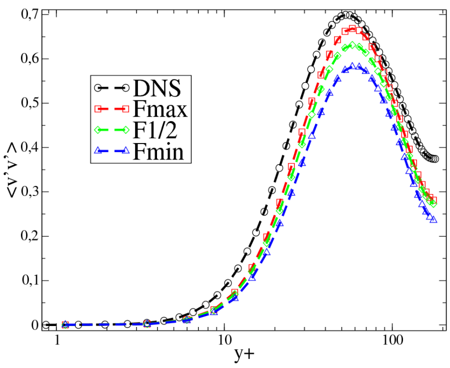

Figure 5 : $< v'v' >$ at different z locations

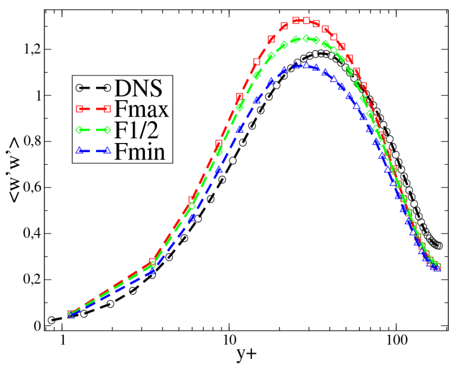

Figure 6 : $< w'w' >$ at different z locations

References

[1] S. Laizet & E. Lamballais. High-order compact schemes for incompressible flows: a simple and efficient method with the quasi-spectral accuracy. *Journal of Computational Physics*, 2009.

[2] J. Kim, P. Moin, R. Moser. Turbulence statistics in fully developed channel flow at low reynolds number. *Journal of Fluid Mechanics*, **177**:133–166, 1987.

[3] N. Jarrin, S. Benhamadouche, D. Laurence, R. Prosser. A synthetic-eddy-method for generating inflow conditions for les. *International Journal of Heat and Fluid Flow*, **27**:585–593, 2006.

[4] Archambeau, N. Mehitoua, M. Sakiz. Code saturne: A finite volume code for the computation of turbulent incompressible flows. *International Journal on Finite Volume*, 2004.