LES of the turbulent channel flow at Re_τ=550

contact: koen.hillewaert@cenaero.be, corentin.cartondewiart@nasa.gov

Introduction

This test case concerns the LES of the channel flow atRe_τ=550, which is slightly different from the corresponding test case at the 4th edition of the workshop, where Re_τ=590. This well-known benchmark was chosen since detailed DNS reference results are publicly available [1,2], providing various quantities to assess the accuracy of the LES approach. This includes the mean velocity and velocity fluctuations profiles as well as kinetic energy spectra. An instantaneous flow field is presented (at Re_τ=590) on Figure 1.

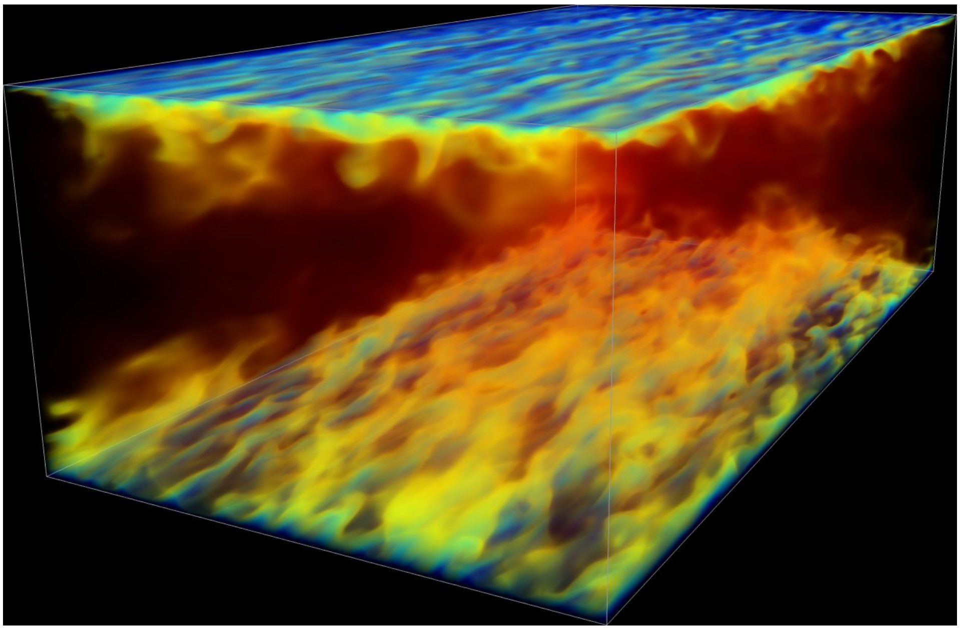

Figure 1. Volume rendering of the velocity.

The benchmark allows to compare the accuracy and the efficiency of the LES approach for near wall turbulence. The case is very sensitive to the dissipation of the model/discretisation and the correct velocity distribution can be tricky to obtain if the correct amount of dissipation is not applied on the right range of turbulent scales.

Required simulations/data

Two computational campaigns are proposed: one (mandatory) on structured grids and one (optional) on unstructured grids. The participants will provide the following statistics:

- Temporal and spatial (wall-parallel directions) averaged profiles in wall units
  - Time-averaged velocity
  - Velocity variances (<u’2>, <v’2>, <w’2>)
  - Velocity component temporal cross-correlations (<u’v’>,<v’w’>,<w’u’>)
- Temporal evolution of
  - the integrated shear stress at top and bottom walls separately
  - the mass flux through the inlet plane
- Optional : spectra

Mandatory campaign - Structured grids

Participants must provide grid convergence on structured grids. Only the number of *dof* in the wall-normal direction will be kept constant. Grids will be provided to the participants to ensure that they use the same mesh distribution in the wall normal direction.

Optional campaign - Unstructured grids

For the solvers that can handle unstructured grids, participants are greatly encouraged to perform studies on unstructured grid. A similar campaign than that on the structured will be performed: three mesh resolutions will be provided to the participants with a fixed number of *dof* in the wall-normal direction.

Detailed description

Governing Equations, flow conditions and models

Compressible or incompressible Navier-Stokes equations can be used for this benchmark. For compressible code, the Mach number, based on the bulk velocity u_b is set to M_b=0.1. The flow is assumed periodic in the streamwise and the spanwise directions. The friction Reynolds number is imposed through a constant forcing in the x-momentum equation. This forcing is given by a pressure gradient, linked to the friction Reynolds number by the following relations:

$$Re_\tau = \frac{\delta u_\tau}{\nu}, \quad u_\tau = \sqrt{\frac{\tau_w}{\rho}}, \quad \frac{dp}{dx} = \frac{\tau_w}{\delta}$$

with δ the half height of the channel. Finally the Prandtl number is fixed to Pr=0.71.

Geometry and grids

The computational domain is 2πδ * 2δ * πδ. A set of structured and unstructured grids are provided on this computational domain. Structured grids are mandatory and the participants are greatly encouraged to run their codes on the unstructured grids.
Two GMSH files are provided to generate the meshes:

- channel_structured.geo
- channel_unstructured.geo

These two files can generate (linear) meshes suitable for high-order solution polynomial. To do so, users must provide the order N=p+1 of the solution together with the mesh resolution (coarse, baseline or fine, see below).

The following mesh resolutions are provided:

Type	Resolution	ID	Number of *dof*	$\Delta x^+ \times \Delta y^+ \times \Delta z^+$
Structured	Fine	MS1	256 * 96 * 128	14.5 × ~0.7 × 14.5
	Baseline	MS2	192 * 96 * 96	19.3 × ~0.7 × 19.3
	Coarse	MS3	128 * 96 * 64	29.0 × ~0.7 × 29.0
Unstructured	Fine	MU1	-	~14.5 × ~0.7 × ~14.5
	Baseline	MU2	-	~19.3 × ~0.7 × ~19.3
	Coarse	MU3	-	~29.0 × ~0.7 × ~29.0

Meshes are designed to give a y+=h0/N<1 at the wall, with h0 the first element height and N=p+1, the solution order.
For each mesh resolution/type, a set of hexahedral meshes compatible with different orders of accuracy (following the p+1 rule) will be provided the participants. For the unstructured meshes, the number of dof are close to those of the structured meshes and the distribution in the y-direction identical.

Initial condition and simulation time

The choice of the initiation solution is crucial to avoid long transient times and to ensure the thermodynamics properties (for the compressible solvers) to remain at a correct level. To this end, we propose to use an initial condition based on the Reichardt function:

$$u^+ = \frac{1}{\kappa} ln\left(1+\kappa y^+\right) + \left(C - \frac{1}{\kappa} ln\left(\kappa\right)\right)\left(1 - e^{-\frac{y^+}{11}} - \frac{y^+}{11} e^{-\frac{y^+}{3}}\right)$$

This initial solution can be perturbed using sine and cosine functions of various wave lenghts.

To get rid of the transient region, at least 20t+ must be computed before the accumulation of the statistics. At least 20t+ additional time must be computed to obtain statistical convergence. To summarise, **the computation should run for at least 40t+!**

Output format

For the averaged profiles, the format should be the following:
y+ *u_mean+* *u_rms+* *v_rms+* *w_rms+* *-uv+*
with the following definitions:

$$y^+ = \frac{\Delta y}{y_\tau} = \frac{\rho u_\tau y}{\mu} \qquad \overline{u^+} = \frac{\overline{u}}{u_\tau}$$

$$(u')^+_{RMS} = \frac{\sqrt{\overline{u'_i u'_i}}}{u_\tau} \qquad -\overline{(u'v')^+} = \frac{\overline{u'v'}}{u_\tau^2}$$

For the monitor files, the format is:
t+ tau_w' m_flux'
using non-dimensional values.

Bibliography

[1] Reference data are available at http://torroja.dmt.upm.es/channels/data/
[2] Sergio Hoyas and Javier Jimenez, "Reynolds number effects on the Reynolds-stress budgets in turbulent channels", 2008, Phys. Fluids, Vol. 20, 101511.
[3] J Jiménez, S Hoyas, MP Simens, Y Mizuno, "Turbulent boundary layers and channels at moderate Reynolds numbers", 2010, Journal of Fluid Mechanics Vol. 657, pp335-360