HPC EUROPA Transnational Access Meeting, Helsinki, June 15th 17th 2010

Direct Numerical Simulation of fractal generated turbulence at high Reynolds numbers

Sylvain Laizet
Imperial College London

Abstract

Recent experiments conducted at Imperial College London (Hurst & Vassilicos, Seoud & Vassilicos, Coffey et al., 2007) have shown that multiscale-generated flows have unusual properties: unlike all other known turbulent flows, such flows offer the unprecedented possibility to generate, with the same input energy, low or high turbulence intensities just by adapting the object's multiscale parameters such as bar thickness or multiscale dimension. Therefore, it is possible to generate high levels of turbulence with a relatively small amount of power and conversely: multiscale objects (see figure 1) can therefore operate as energy-efficient mixers or as low-noise spoilers/airbrakes/flaps that could be used for noise-reduction technologies.

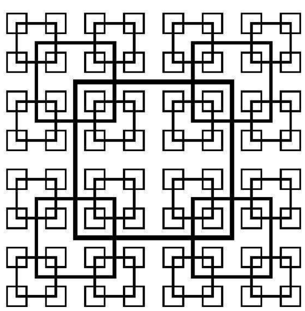

Figure 1: *Scaled diagram of a fractal square grid.*

It is now necessary to build on these new flow configurations and develop a three-dimensional understanding of such flows, especially their mixing and acoustic properties. Such flows pose a formidable challenge to researchers due to complex geometries and the presence of a multitude of length scales. This requires state-of-the art top-end parallel computing for high-resolution simulations to understand the origins of the unique properties of multiscale objects and propose new strategies for industrial applications.

Presentation of the code

The Computational Fluid Dynamics code used in this project is called Incompact3d and is briefly described below. It is based on a compromise between spectral method accuracy and industrial code versatility. The incompressible Navier-Stokes equations and advection-diffusion equations for passive scalars are solved on a Cartesian mesh with 6th-order compact finite difference schemes. The incompressibility condition is ensured by a fractional step method using a Poisson equation. The time advancement can be performed with an Adams-Basforth scheme (2nd order and 3rd order) or a Runge-Kutta scheme (3rd and 4th order). The specificity of this code is that this Poisson equation is solved in the framework of the modified spectral formalism. This one of the only few codes where this equation is solved in spectral space using 3D Fast Fourier Transforms (FFT3D) despite the use of inflow/outflow boundary conditions for the computational domain. This non-iterative solution approach is obviously possible for periodic and free-slip boundary conditions, but also when Dirichlet conditions for the velocity are combined with homogeneous Neuman conditions for the pressure (see for instance Laizet & Lamballais, 2009 for the basic principles of the spectral solution approach to a Poisson equation using a cosine expansion).To simulate a solid body in the flow, this code can be combined with an immersed boundary method. The basic idea behind this method is to mimic the effect of a solid surface on the fluid by a forcing applied in the body region. This forcing is implemented as an additional term in the Navier-Stokes equation. For more details about the code see Laizet et al. (2010).

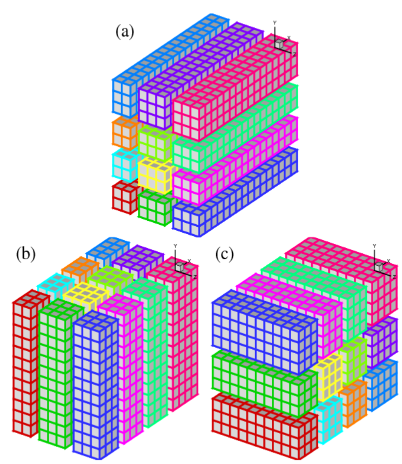

Figure 2: *2D domain decomposition strategy in Incompact3d.*

Incompact3d is a FORTRAN 90 code that was initially designed for serial processors, then converted to vector processors and more recently converted for parallel platforms, parallelised with MPI implementation for running using thousands of cores. The code's 2D domain decomposition (see figure 2, inspired from fully spectral codes, and based on pencil subdomains) strategy offers two major advantages: parallelisation is possible without reducing the order of our schemes, and scalability is excellent because, even though our schemes are implicit in space, there is no data communication (overlapping) at the boundaries of each subdomain. It is very expensive in terms of communication as it requires many global transpose operations (performed with the MPI command ALLTOALL to swap from state a, b and c). Note that the 2D domain decomposition strategy has been developed thanks to the Numerical Algorithms Group (NAG). A dedicated NAG expert in High performance Computing is working in our team in order to maintain the good performance of the code on the current and next generation of supercomputers. It is possible to undertake simulations with up to 100,000 computational cores with Incompact3d (see Laizet and Li, 2010 for more details about the parallel strategy).

Flow configuration

In this study, we are only working with a fractal square grid with three fractal iterations. The computational domain $Lx \times Ly \times Lz = 460.8t_{min} \times 115.2t_{min} \times 115.2t_{min}$ (where t_{min} is the lateral thickness of the smallest squares) is discretized on a Cartesian mesh of $n_x \times n_y \times n_z = 2305 \times 576 \times 576$ mesh nodes. It is split up in 288 computational cores. Two simulations were performed to investigate the influence of the Reynolds number. One with a relatively small Reynolds number equal to 300 and one with a Reynolds number equal to 1500, corresponding to a velocity of 10m/s in a wind tunnel.

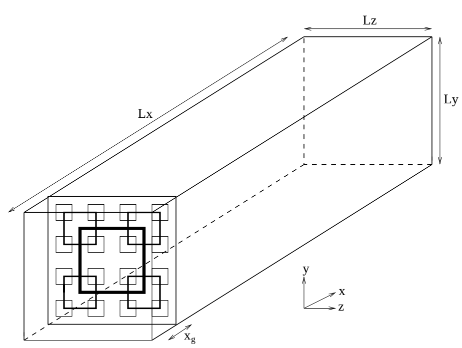

Figure 3: *Schematic view of the flow configuration for the fractal square grid.*

Results

It was found that, unlike a regular object (where the turbulence is generated by only one scale), a slight modification of one of the object's parameters can deeply modify the turbulence generated by the fluid's impact on the object. Furthermore, the experiments exhibit that there is no influence of the input energy when the flow is generated by a multiscale object, unlike classical grids.

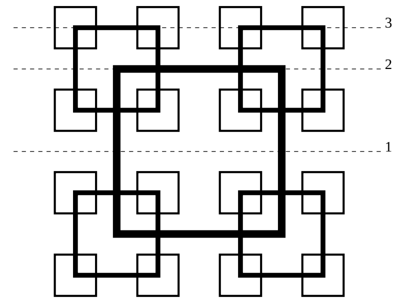

Figure 4: *Locations of the instantaneous visualizations in the $(x-y)$ plane of the flow.*

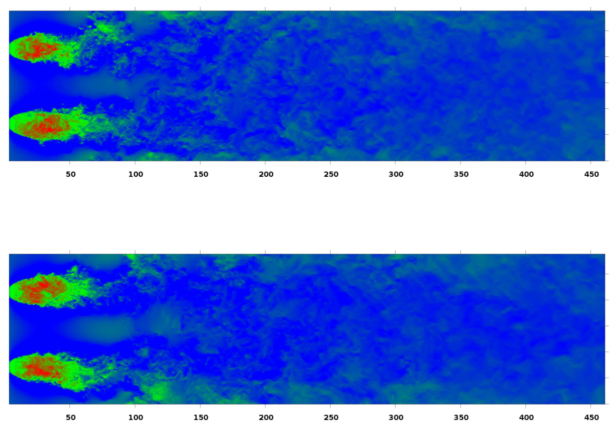

Figure 5: *Instantaneous visualizations of the streamwise component of the velocity at the location 1. Top: $Re = 1500$, bottom: $Re = 300$.*

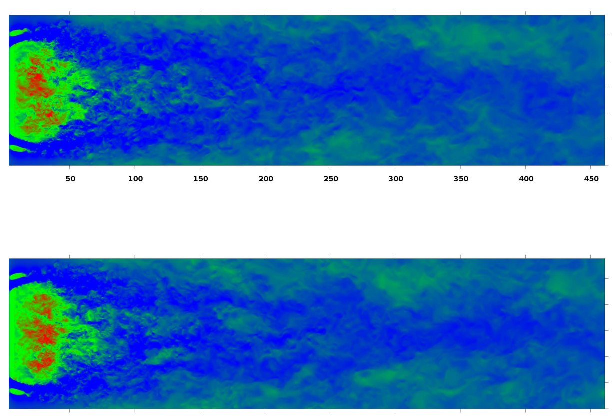

Figure 6: *Instantaneous visualizations of the streamwise component of the velocity at the location 2. Top: $Re = 1500$, bottom: $Re = 300$.*

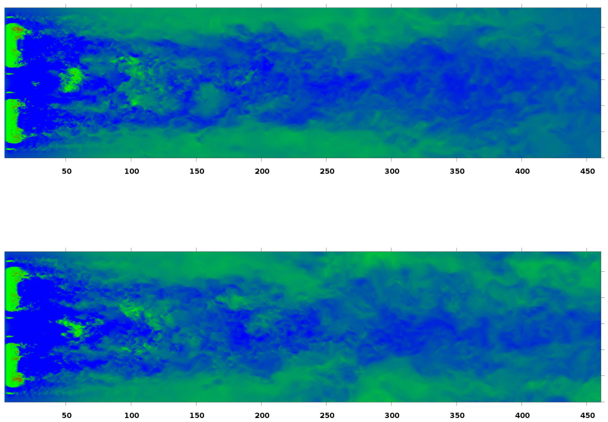

Figure 7: *Instantaneous visualizations of the streamwise component of the velocity at the location 3. Top: $Re =$ 1500, bottom: $Re = 300$.*

In these two simulations, we are investigating the effect of the input energy on the flow. In agreement with the wind tunnel measurements, the preliminary results based on 2D instantaneous flow visualizations (see figures 5 to 7) seem to prove that the turbulent flow generated by a fractal grid is independent with the energy input. The flow obtained with the small Re is almost the same as the one obtained with the high Re. Usually, with a classical grid, when you increase the input energy, you increase the level of turbulence of the flow downstream of the grid. Furthermore, and in good agreement with the experiments, the location of the peak of turbulence on the centreline of the grid, as shown in figure 8, is independent of the input energy. It should be notice as well that the level of turbulence at the end of the computational domain is almost the same for the two simulations. It is also important to point out that the level of turbulence here is at least four times bigger than the one generated by a classical grid with the same Reynolds numbers.

Further investigations are currently in progress in order to examine more in details the three-dimensional structure of multiscale-generated turbulence. Statistical analysis are required but the post-processing is quite a long task in order to obtain well converged data in time. The idea is to obtain insights into the unusual properties of the flow dynamics and the mixing/acoustic properties of multiscale-generated flows.

The pioneering of research on fractal-generated turbulent flows is particularly important because the properties of some of them in their homogeneous isotropic flow region are so different from the usual properties of homogeneous isotropic turbulence that one is offered with a unique opportunity to exploit these differences for an unprecedented attempt at understanding turbulence dynamics.

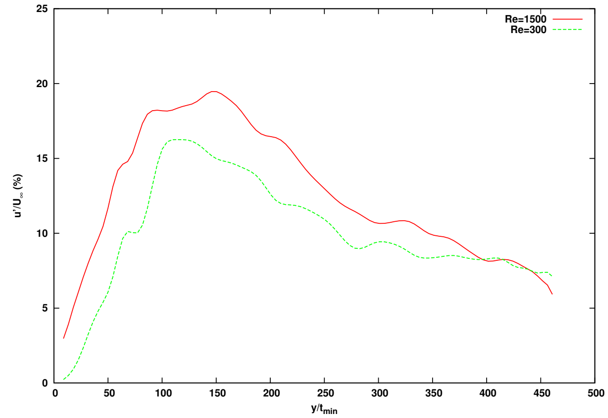

Figure 8: *Streamwise evolution of the turbulence intensity along the centreline of the fractal grid.*

Acknowledgements

Sylvain Laizet acknowledges support from the UK EPSRC grant EP/E029515/1. He also thanks Ning Li, Eric Boyer, Prof. Eric Lamballais and Prof. Christos Vassilicos for very useful discussions.

The work has been performed under the HPC-EUROPA2 project (project number: 228398) with the support of the European Commission - Capacities Area - Research Infrastructures)

References

Coffey C.J., Hunt G.R., Seoud R.E. and Vassilicos J.C., (2007), Mixing effectiveness of fractal grids for inline static mixers, Proof of Concept report for the attention of Imperial Innovations.

Hurst D. and Vassilicos J.C., (2007), Scaling and decay of fractal-generated turbulence, Phys. Fluids, Vol. 19, 035103.

Laizet S., Lamballais E. and Vassilicos J.C. (2009) A numerical strategy to combine high-order schemes, complex geometry and parallel computing for high-resolution DNS of fractal generated turbulence, Computers & fluids, 39(3), 471–484

Laizet S. and Li N. (2010) Incompact3d: a powerful tool to tackle turbulence problems with up to $O(10^5)$ computational cores, submitted to Int. J. Numer. Methods Fluids

Laizet S. and Lamballais E. (2009) High-order compact schemes for incompressible flows: a simple and efficient method with the quasi-spectral accuracy, J. Comp. Phys., 228(15), 5989-6015

Seoud R.E. and Vassilicos J.C., (2007), Dissipation and decay of fractal-generated turbulence, Phys. Fluids, Vol. 19, 105108.