

INCOMPACT3D: A POWERFUL TOOL TO TACKLE TURBULENCE PROBLEMS ON SUPERCOMPUTERS

SYLVAIN LAIZET

Turbulence, Mixing and Flow Control Group
Department of Aeronautics
South Kensington Campus, London SW7 2AZ, UK
e-mail: s.laizet@imperial.ac.uk, web page:
<http://www3.imperial.ac.uk/tmfc/people/sylvainlaizet/>

Key words: CFD methodology, Turbulence, High Performance Computing, High order finite-difference schemes, Spectral methods

Abstract. Simulating and understanding turbulent flows remains one of the most challenging problems in classical physics. Significant progress has been made recently using high performance computing (HPC), and computational fluid dynamics (CFD) is now a credible alternative to experiments and theories in order to understand the rich physics of turbulence. Only very few codes dedicated to Direct and Large Eddy Simulations (DNS/LES) are capable of undertaking massive simulations with several billion mesh nodes on thousands of computational cores. Most of them are simulating idealized homogeneous, isotropic turbulence, using spectral methods with at least periodic boundary conditions in two spatial directions. Unfortunately, for engineering problems in slightly more complex geometry, the full spectral approach (the most accurate one) is no yet feasible. In conventional CFD, especially in an industrial context, complex geometries are usually treated using non-structured element meshes, requiring low-order schemes and sophisticated tools for the generation of highly distorted meshes. The resulting accuracy is most of the time clearly incompatible for a detailed analysis of engineering problems. In this paper/talk, I am presenting an innovative numerical tool which can conciliate accuracy, efficiency, versatility and scalability using a Cartesian grid. This tool allows accurate simulations of a wide range of engineering configurations. Various examples will be shown in this paper/talk such as fractal-generated turbulence, gravity currents, impinging jets on a heated plate and a microjet device to control a turbulent jet.

1 CONTEXT

With recent impressive developments in computer technology, High Performance Computing (HPC) has entered in 2008 the reality of petascale computing with far-reaching consequences for scientific research. HPC in Computational Fluid Dynamics is expected to open the doors to solving highly complex turbulence problems that were until very recently beyond our imagination. Characterized by complex, disorderly motions over a wide range of scales in time and space, turbulence is a grand challenge question that

cuts across numerous disciplinary boundaries, from the science of atmospheric phenomena, to the physics of combustion as an energy source for cars or/and jet engines. Many generations of scientists have struggled to understand both the physical essence and the mathematical structure of turbulence. Richard Feynman (1918-1988) said about turbulence that *“It is the most important unsolved problem of classical physics”*.

In the last 50 years, some progress have been made in the understanding of the turbulence problem, thanks to increasingly more complex experiments using advanced and sophisticated techniques, and, of course, thanks to the introduction and widespread use of numerical simulations. The recent unprecedented developments in computer technology had and will have a strong impact on the turbulence research, especially on three aspects: Direct Numerical Simulation (DNS) of idealized turbulence, increasingly sophisticated engineering models of turbulence, and the extraordinary enhancement in the quality and quantity of experimental data achieved thanks to computer storage and computer post-treatment.

From a fundamental point of view, DNS of idealized isotropic, homogeneous turbulence has been revolutionary in its impact on turbulence research because of the possibility to simulate and display the full 3-D velocity field without any modelling. One of the most challenging aspects of the turbulence is that the velocity fluctuates over a large range of coupled spatial and temporal scales. If we want to understand the turbulence problem without introducing any bias through the numerical methods, it is important to use the most accurate computational approach: Direct Numerical Simulation (DNS). Unfortunately, the computational cost of DNS, even for idealized turbulent flows, is very high, especially when increasing the Reynolds number, as a result of the non-linearity and the non-locality of the Navier-Stokes equations, equations describing the fluid motions. Furthermore, for flows with relatively complex geometries at relevant Reynolds number (i.e. representative of real situations), the computational resources required by DNS often drastically exceed the capacity of the most powerful massive parallel platforms. In this context, the few DNS codes capable of undertaking massive simulations with several billion mesh nodes are simulating either idealized homogeneous, isotropic turbulence, turbulent channel flows and/or turbulent boundary layers using spectral methods in at least two spatial directions. For such academic configurations, in terms of accuracy and computational efficiency, the most spectacular gain is obtained using spectral methods. Unfortunately, for fundamental problems in slightly more complex geometry, the full spectral approach is no longer feasible, even if the spectral element method seems to be very promising.

One solution to simulate relatively complex turbulent flows with a very good accuracy using supercomputers is to use the in-house code **Incompact3d**. This in-house code can combined the versatility of industrial codes with the accuracy of the best academic codes based on spectral methods (the most accurate ones) and can be applied to complex turbulent flows. **Incompact3d** is numerical code originating from the University of Poitiers (France) and developed there as well as, more recently, in the Turbulence, Mixing and Flow Control Group at Imperial College London. It is dedicated to Direct and Large

Eddy Simulations (DNS/LES) using High Performance Computing to simulate turbulent flows at the interface between academic research and upstream industrial R&D.

2 NUMERICAL STRATEGY

In conventional Computational Fluid Dynamics (CFD), especially in an industrial context, complex geometries are usually treated using complex numerical methods, mainly based on low-order (low-accuracy) schemes and sophisticated tools for the generation of highly distorted meshes. The resulting accuracy is very often incompatible with the requirements for a detailed analysis of any complex fluid-flow problems. **Incompact3D**, is one of the very few codes in Europe which can combined the versatility of industrial codes with the accuracy of the best academic codes based on spectral methods (the most accurate ones) and that can be used on the most powerful supercomputers in the world. It is an efficient tool to address rigorously high-quality simulations of turbulence and heat transfer. This code is based on a Cartesian mesh. The use of such a simplified mesh offers the opportunity to implement high-order compact schemes (sixth-order schemes, with a quasi-spectral accuracy) for the spatial discretisation whilst an customized Immersed Boundary Method (IBM) allows the implementation of any solid wall/bluff body geometry inside the flow. The main originality of this code is that the Poisson equation (to ensure the incompressibility of the flow) is fully solved in the spectral space via the modified wave number formalism, no matter what the boundary conditions are (periodic, free-slip, no-slip, inflow/outflow). More information about the numerical methods of the code can be found in [1]. Thank to various successful projects in the UK with NAG (National Algorithm Group), **Incompact3d** can be used efficiently on hundreds of thousands computational cores to investigate turbulence and heat transfer problems. This high level of parallelisation is achieved thanks to the development of a highly scalable 2D domain decomposition library and a distributed Fast Fourier Transform (FFT) interface [2].

3 EXAMPLES

Four applications, showing the versatility of the code, will be presented in the talk:

- **Fractal-generated turbulence:** For this research topic, the idea is to create a virtual wind tunnel to reproduce numerically experiments of grid-generated turbulence. The originality of this work is the fractal patterns used to design the grids which shape the nature of the resulting turbulent flow over a broad range of scales. The main goal of the simulations is to gain fundamental understanding of the spatial structure of these flows. A turbulent flow obtained behind a fractal square grid can be seen in figure 1.
- **Gravity currents:** For these simulations we are interested in the prediction of a mono-disperse dilute suspension particle-laden flow in the typical lock-exchange configuration under the Boussinesq approximation. The main originality of this work is that the deposition of particles at the bottom of the computational domain is taken into account for velocities comparable to laboratory experiments. An illus-

tration of the flow can be seen in figure 2 at different times representative of the spatio-temporal evolution of the current.

- **Impinging jet on a heated plate:** Impinging jets are widely used in industrial applications as efficient tools to enhance heat transfer between a uid and an impinged solid target. DNS/LES of an axisymmetric jet impinging a flat wall were carried out with **Incompact3d** at relatively high Reynolds numbers. The impingement wall is heated and both dynamical and thermal features of the ow were simulated. An instantaneous visualisation of the flow can be seen in figure 3.
- **Microjet device to control a turbulent jet:** A large number of studies in the literature have focused on the control of a jet, mainly with two objectives: the control of the mixing properties of the jet and for noise reduction purposes. The objective of the simulations performed with **Incompact3d** is to help in our understanding of the aeroacoustic mechanisms in the context of fluidic control. More precisely, the idea is to propose control solutions of the acoustic sources of a jet. The main originality of this numerical work is that the microjet device is included inside the computational domain as shown in figure 4, using a customized Immersed Boundary Method.

Acknowledgements

The author is grateful to Dr. Ning Li and Prof. Eric Lamballais for helping with the parallel version of **Incompact3d**, and to Prof J. Christos Vassilicos for the constant support for the code and for discussions about the fractal-generated turbulence simulations. The author would like to thank Thibault Dairay, Rémi Gauthier, Jorge Silvestrini, Léandro Pinto and Luis Felipe for providing some of the data for the impinging jets, gravity currents and microjet device control of a turbulent jet simulations. The author also acknowledges support from EPSRC Research grants EP/E00847X/1 and EP/F051468/1.

REFERENCES

- [1] S. Laizet and E. Lamballais. High-order compact schemes for incompressible flows: a simple and efficient method with the quasi-spectral accuracy. *J. Comp. Phys.*, **228(16)**:5989–6015, 2009.
- [2] S. Laizet and N. Li. Incompact3d, a powerful tool to tackle turbulence problems with up to $o(10^5)$ computational cores. *Int. J. Numer. Methods Fluids*, **67(11)**:1735–1757, 2011.

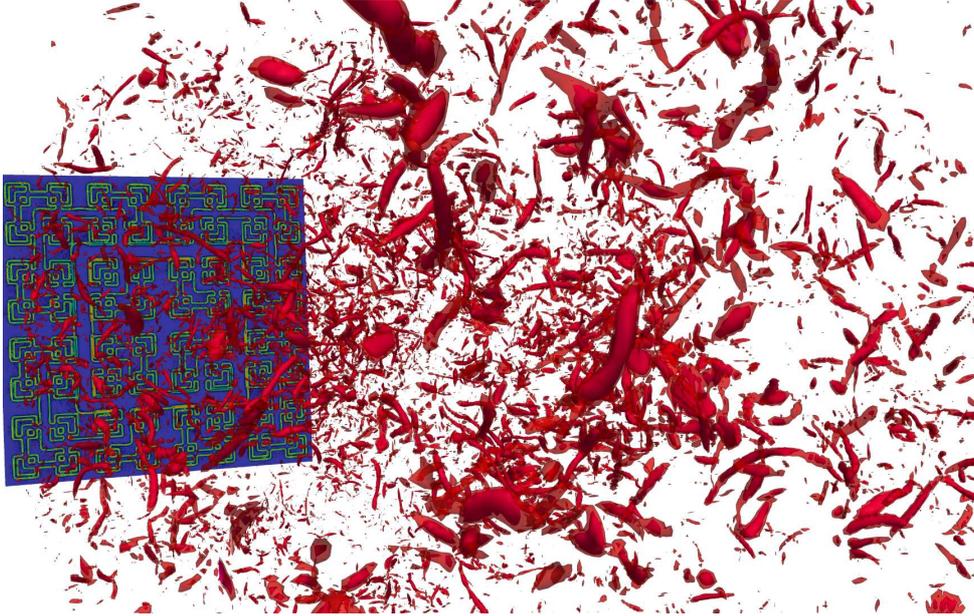


Figure 1: Direct Numerical Simulation of fractal-generated turbulence.

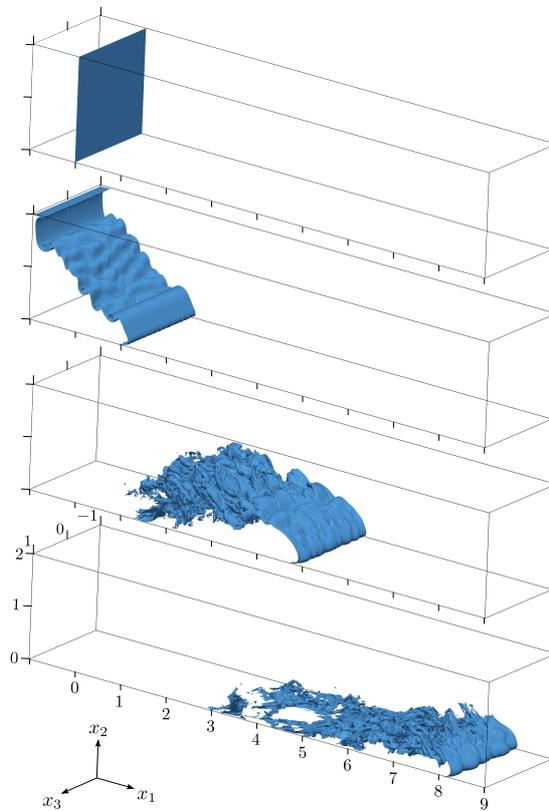


Figure 2: Spatio-temporal evolution of a gravity current in the lock-exchange configuration.

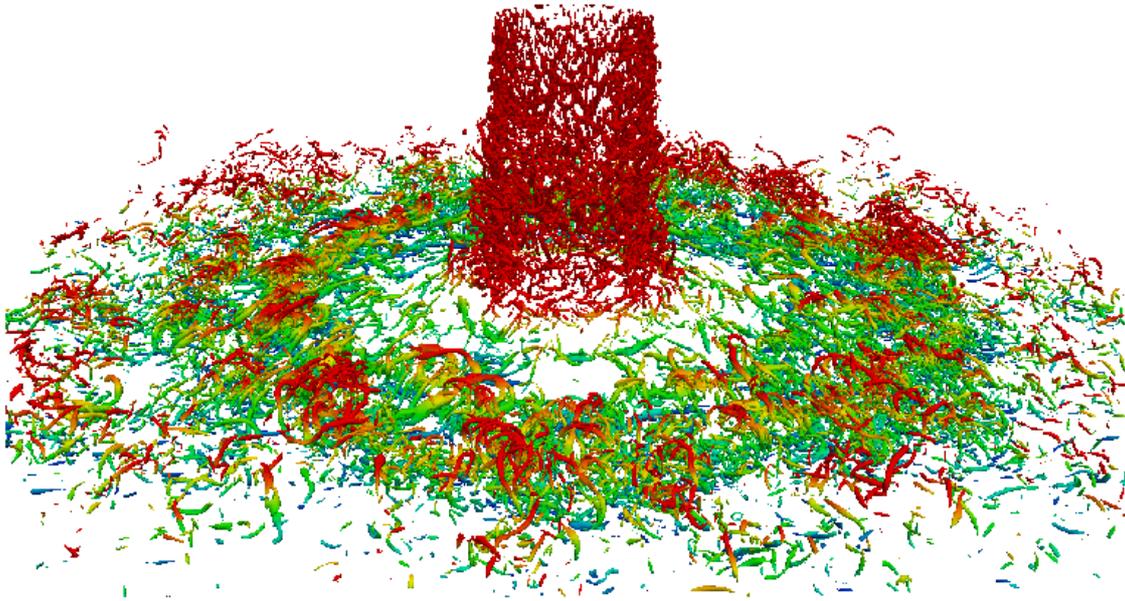


Figure 3: Direct Numerical Simulation of an impinging jet on a heated plate.

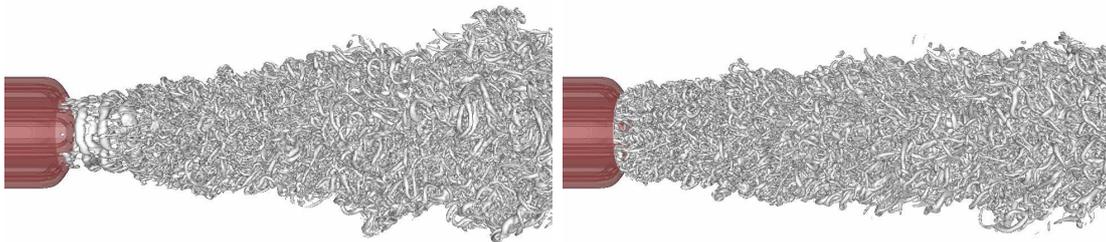


Figure 4: Direct Numerical Simulations of a turbulent jet with no control at the exit of the nozzle (left) and with microjets control at the exit of the nozzle (right).