Numerical simulation of turbulence at lower costs: regularization modeling

F.X. Trias*, A. Gorobets*,+, M. Soria*, A. Oliva*,
Corresponding author: cttc@cttc.upc.edu

* Heat and Mass Transfer Technological Center, Technical University of Catalonia
ETSEIAT, C/Colom 11, 08222 Terrassa, Spain.
+ Keldysh Institute of Applied Mathematics of RAS, Russia.

Abstract: This work is devoted to the development of efficient methods for the numerical simulation of incompressible flows on modern supercomputers. Direct simulation of the Navier-Stokes equations is nowadays an essential tool to provide new insights into the physics of turbulence and indispensable data for the development of better turbulence models. However, since DNS simulations at high Reynolds numbers are not feasible because the convective term produces far too many scales of motion, a dynamically less complex mathematical formulation is sought. In the quest for such a formulation, we consider regularizations of the convective term that preserve symmetry and conservation properties exactly. This yields a novel class of regularizations that restrain the convective production of small scales of motion in an unconditionally stable manner. In this way, the new set of equations is dynamically less complex than the original Navier-Stokes equations, and therefore more amenable to be numerically solved. The only additional ingredient is a self-adjoint linear filter whose local filter length is determined from the requirement that vortex-stretching must be stopped at the scale set by the grid. Here, the performance of the method is tested by means of direct comparison with several DNS reference simulations.

Keywords: DNS, Regularization modeling, Symmetry-preserving, Turbulence

1 Introduction

The incompressible Navier-Stokes (NS) equations form an excellent mathematical model for turbulent flows. In primitive variables the equations are

$$\partial_t u + C(u, u) = D u - \nabla p; \quad \nabla \cdot u = 0,$$

where $u$ denotes the velocity field and $p$ represents the pressure. The convective and diffusive terms are respectively defined by $C(u, v) = (u \cdot \nabla) v$ and $D u = Re^{-1} \Delta u$ where $Re$ is the Reynolds number. Unfortunately, attempts at performing Direct Numerical Simulations (DNS) with the available computational resources and numerical methods are limited to relatively low-Reynolds numbers. Regarding the numerical algorithms, cost reductions can be achieved by one or more of the following issues: (i) decreasing the number of grid points using high-order schemes, (ii) reducing the computational cost per iteration or
(iii) using larger time-steps; all without affecting the quality of the numerical solution. In the companion work [1], we have focused on the improvement of the parallel Poisson solver by means of a hybrid parallelization strategy. The previous version [2] was conceived for single-core (also dual-core) processors and therefore, the distributed memory model with message-passing interface (MPI) was used. The irruption of multi-core architectures motivated the use of a two-level hybrid MPI+OpenMP parallelization with the shared memory model on the second level. Advantages and implementation details for the additional OpenMP parallelization are presented and discussed in the companion paper [1]. Numerical experiments show that, within its range of efficient scalability, the previous MPI-only parallelization is slightly outperformed by the MPI+OpenMP approach. But more importantly, the hybrid parallelization has allowed to significantly improve the range of efficient scalability. The solver has been successfully tested up to 12800 CPUs for meshes with up to $10^9$ grid points (see Figure 1) on the Lomonosov supercomputer. However, estimations based on the presented results show that this range can be potentially stretched up until more than 100000 cores (see the companion paper [1], for details).

![Figure 1: Speedups on the Lomonosov supercomputer for two meshes: a 327.7 million (256 × 800 × 1600) grid points (Mesh2) and a 1003.5 million (256 × 1400 × 2800) grid points (Mesh3). Left: speedups for the Poisson solver and the overall algorithm when varying the number of MPI processes in the $x$-periodic direction, $P_x$, from 1 to 8 and keeping $P_{yz} = 200$ (number of MPI processes in the $\{y,z\}$-plane) and $P_t = 8$ (number of threads) constant. Right: Speedups of the Poisson solver for MPI+OpenMP (varying $P_{yz}$ from 200 to 800 and keeping $P_x = 1$ and $P_t = 8$ fixed). For details, the reader is referred to the companion paper [1]](image)

2 Direct numerical simulation

In the present work, two new DNS simulations are presented. Namely, a turbulent plane impinging jet and a differentially heated cavity (see Figures 2 and 3). They have been carried out on the MareNostrum supercomputer and illustrate some potential applications of the Poisson solver presented in [1]. In both cases, the spatial discretization is fourth-order accurate and preserves the (skew)symmetries of the underlying continuous differential operators [3]. For details about the numerical algorithms, boundary conditions and the verification of the DNS code the reader is referred to [4]. The DNS simulation of the turbulent plane impinging jet corresponds to well-known configuration: Reynolds number 20000 (based on the inflow velocity and the nozzle width, $B$) and aspect ratio 4 (nozzle-to-impingement surface distance/nozzle width). This test-case has been studied both numerical and experimentally (see [5]...
and references therein). However, the availability of DNS reference results is quite scarce and restricted only to lower Reynolds numbers [6]. More importantly, time-averaged DNS results (see Figure 2, left) have revealed that main recirculating flow cannot be captured well unless the outflow is placed at least at $40B$ from the jet centreline approximately. With regard to this point, it must be noted that most of the previous numerical [7] and experimental studies [8, 9] placed the outflow at $x/B = \pm 10 \sim 15$, and therefore cannot capture adequately the recirculating flow, one of the main features of this configuration. This suggests that previous experimental data may not be adequate to study the flow configuration far from the jet. This simulation has been carried out using 300 CPUs ($P_x = 2$, $P_y = 20$ and $P_z = 8$) on a 104 million grid points mesh ($168 \times 1680 \times 368$). An illustrative instantaneous map is displayed in Figure 2 (right).

The second simulation corresponds to a buoyancy-driven turbulent flow in an air-filled ($Pr = 0.71$) differentially heated cavity (DHC) of height aspect ratio 5 and Rayleigh number (based on the cavity height) $4.5 \times 10^{10}$. This resembles the pioneering experimental set-up performed by Cheesewright et al. [10] in the mid-80s. Since then, their results have been widely used for benchmarking purposes to validate turbulence models (see [11], for instance); therefore, the availability of accurate numerical results is of extreme importance. To that end, a new complete direct numerical simulation (DNS) has
been performed. We have used a 128 × 318 × 862 staggered grid to cover the computational domain. Results obtained suggest that the transition of the vertical boundary layer occurs at more downstream positions that those observed in the experiments (see local Nusselt number distribution in Figure 3, right). This feature has already been observed for a turbulent DHC of aspect ratio 4 [4]. Illustrative instantaneous isotherms are displayed in Figure 3 (left). Detailed results obtained of these two new DNS simulations will be presented and analyzed in the final paper and in the conference.

3 Turbulence modeling: $C_4$ regularization

However, direct simulations at high Reynolds (or Rayleigh) numbers are not feasible because the convective term produces far too many relevant scales of motion. Therefore, a dynamically less complex mathematical formulation is needed. In the quest for such a formulation, here we consider regularizations [12, 13] (smooth approximations) of the NS equations. The first outstanding approach in this direction goes back to Leray [14]. The NS-α model also forms an example of regularization modeling [13]. Basically they alter the convective terms to reduce the production of small scales of motion. In doing so, we proposed to preserve exactly the symmetry and conservation properties of the convective terms. Doing so, certain fundamental properties such as the inviscid invariants - kinetic energy, enstrophy (in 2D) and helicity (in 3D) - are exactly preserved in a discrete sense. This requirement yielded a family of symmetry-preserving regularization models [15]. Here, we restrict ourselves to the $C_4$ approximation: the convective term in the NS equations (1) is then replaced by the following $O(\varepsilon^4)$-accurate smooth approximation $C_4(u,v)$ given by

$$C_4(u,v) = C(u,v) + C(u',v) + C(u,v'),$$

where the prime indicates the residual of the filter, e.g. $u' = u - \bar{u}$, which can be explicitly evaluated, and $(\cdot)$ represents a symmetric linear filter with filter length $\varepsilon$. Therefore, the governing equations result to

$$\partial_t u + C_4(u,u) = Du - \nabla p; \quad \nabla \cdot u = 0.$$ 

(3)

Note that the $C_4$ approximation is also a skew-symmetric operator like the original convective operator. Hence, the same inviscid invariants than the original NS equations are preserved for the new set of partial differential equations (3). The $C_4$ method has already been successfully tested for a turbulent channel flow [15] and a DHC of aspect ratio 4 at $Ra \leq 10^{11}$ [16]. In the present work, we propose to test the performance of the $C_4$-regularization method in conjunction with the new family of discrete filters recently proposed in [17] by means of direct comparison with the above-described DNS simulations.

Acknowledgments

This work has been financially supported by the Ministerio de Ciencia e Innovación, Spain (ENE2010-17801), the Russian Federation President grant MK-7559.2010.1, and a Juan de la Cierva postdoctoral contract (JCI-2009-04910) by the Ministerio de Ciencia e Innovación. Calculations have been performed on the Lomonosov supercomputer at the Moscow State University and on the IBM MareNostrum supercomputer at the Barcelona Supercomputing Center. The authors thankfully acknowledge these institutions.

References


