Journal of Computational Physics 300 (2015) 710–731

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp

A parallelized multidomain compact solver for incompressible turbulent flows in cylindrical geometries

Romain Oguic [a], Stéphane Viazzo [a,*], Sébastien Poncet [a,b]

[a] Aix-Marseille Université, CNRS, École Centrale Marseille, Laboratoire M2P2 UMR 7340, 38 rue Frédéric Joliot-Curie, Technopôle Château-Gombert, 13451 Marseille, France

[b] Département de génie mécanique, Faculté de Génie, Université de Sherbrooke, 2500 Boulevard de l'Université, Sherbrooke J1K 2R1, Québec, Canada

ARTICLE INFO

Article history:
Received 26 February 2015
Received in revised form 26 June 2015
Accepted 1 August 2015
Available online 6 August 2015

Keywords:
Fourth-order compact finite differences
Axis singularity
Multidomain decomposition solver
Hybrid parallelization
Turbulent pipe flow
Rotating flows

ABSTRACT

We present an efficient parallelized multidomain algorithm for solving the 3D Navier–Stokes equations in cylindrical geometries. The numerical method is based on fourth-order compact schemes in the two non-homogeneous directions and Fourier series expansion in the azimuthal direction. The temporal scheme is a second-order semi-implicit projection scheme leading to the solution of five Helmholtz/Poisson equations. To handle the singularity appearing at the axis in cylindrical coordinates, while being able to have a thinner or conversely a coarser mesh in this zone, parity conditions are imposed at $r = 0$ for each flow variable and azimuthal Fourier mode. To simulate flows in irregularly shaped cylindrical geometries and benefit from a hybrid OpenMP/MPI parallelization, an accurate perfectly free-divergence multidomain method based on the influence matrix technique is proposed. First, the accuracy of the present solver is checked by comparison with analytical solutions and the scalability is then evaluated. Simulations using the present code are then compared to reliable experimental and numerical results of the literature showing good quantitative agreements in the cases of the axisymmetric and 3D unsteady vortex breakdowns in a cylinder and turbulent pipe flow. Finally to show the capability of the algorithm to deal with more complex flows relevant of turbomachineries, the turbulent flow inside a simplified stage of High-Pressure compressor is considered.

1. Introduction

Turbulent confined flows in cylindrical geometries may be found in a large variety of fundamental and industrial systems, concerning meteorological and geophysical problems, pipe heat exchangers or rotating machineries. Accurate numerical methods like spectral or high-order compact finite-difference methods have demonstrated their capability to investigate instability mechanisms and weakly turbulent flows. From a numerical point of view, these flows remain very challenging and dedicated numerical techniques must be developed to overcome the main following difficulties:

- to use high-order methods able to accurately predict the coexistence of laminar, transitional and highly turbulent flow regions,

* Corresponding author. Tel.: +33 4 91 11 85 50; fax: +33 4 91 11 85 02.
E-mail address: stephane.viazzo@univ-amu.fr (S. Viazzo).

http://dx.doi.org/10.1016/j.jcp.2015.08.003
0021-9991/

- to handle appropriately the axis singularity in the continuity and Navier–Stokes equations due to the terms multiplied by $1/r$ or $1/r^2$ (r the radial distance),
- to simulate flows in "complex geometries", especially for turbomachinery applications,
- to simulate high Reynolds number flows, which require very high spatial resolutions and so a huge amount of computational resources.

In order to capture all space and time scales of turbulent flows, the use of highly accurate schemes is required. Among them, the most popular ones are spectral and compact finite difference methods. Spectral schemes provide highly accurate results but all in with considerable computational efforts. They suffer also from a lack of flexibility such that even semi-complex geometries are difficult to consider. On the other hand, finite difference schemes (explicit or implicit) may overcome this last drawback while keeping a high-order accuracy. Implicit finite-difference schemes, also called compact schemes, exhibit a certain benefit compared to explicit schemes: the order of accuracy is higher for a same stencil size and the dissipation is smaller [1], while retaining the easy-to-use of finite differences.

The choice of cylindrical coordinates to simulate pipe or rotating flows seems the most natural one and allows a spatial discretization in the periodic direction with the Fourier expansions that can be especially beneficial with the use of Fast Fourier Transform (FFT). Nevertheless, in purely cylindrical geometries, it introduces a singularity at the axis. Some terms in the governing equations of fluid motion are indeed multiplied by $1/r$ or $1/r^2$, whereas the flow field does not exhibit any singular behavior on the axis ($r = 0$). The singularity treatment is not a novel problem and many techniques to avoid it have been developed. The choice of the appropriate technique is strongly dependent on the numerical method (spectral, finite-volume, finite-difference methods, etc.) and the first point to address is to know if the flow dynamics requires a coarse or very thin mesh distribution around $r = 0$. One of the first methods was developed by Griffin et al. [2]. It was based on central interpolation without any consideration of the radial momentum equation to solve the radial velocity component. Constantinescu and Lele [3] reinterpreted the regularity conditions developed in the context of pseudospectral methods to develop a more general method for treating the singularity by using series expansions around the axis. Mohseni and Colonius [4] proposed to shift the grid points in the radial direction by half the radial mesh spacing to avoid the resolution of twice the same mesh points. Serre and Pulicani [5] introduced a variable change by multiplying all velocity components and pressure by a factor r. Recently, Peres et al. [6] handled the axis singularity by discretizing the whole diameter with an even number of radial Gauss–Lobatto collocation points and hence skipping the axis. An angular shift is also introduced in the Fourier transform to avoid pole and parity conditions usually required. Though the method keeps the spectral accuracy, their method is well suited only for specific configurations, where the flow is mainly laminar around the axis as the mesh grid is relatively coarse in this zone. The use of a collocative mesh appears to be an efficient way to avoid the singularity but this kind of methods can rapidly lead to huge computational efforts. For example, the use of any existing FFT algorithm is banned in the method developed by Peres et al. [6] and has been then replaced by simple matrix products, which become much more expensive depending on the number of Fourier modes in the tangential direction. Mercader et al. [7] combined a collocative mesh with regularity conditions. Their approach cannot be directly applied to the present numerical solver due to the use of fully staggered mesh or it would require significant and drastic numerical developments. Based on the former work of Lewis and Bellan [8], Sandberg [9] employed parity conditions for each flow variable at the axis. His method is particularly easy to implement and has demonstrated its accuracy. It has been then selected for the present solver. The reader can refer to the monograph of Boyd [10] for a complete overview of the different options of radial basis sets and radial grids for a finite cylindrical domain in the context of spectral methods.

Immersed Boundary Methods (IBM) and penalization methods are now commonly used to simulate flows in complex geometries. Laizet and Lamballais [11] developed a massively parallelized code combining high-order compact schemes and IBM to simulate the multi-scale turbulent flows generated by a fractal grid. Unfortunately, the use of IBM involves a decrease of the spatial accuracy: the theoretical maximum order obtained with this method is only second-order [12]. Moreover, in the context of turbomachinery, where T-shape geometries are often encountered, these methods could induce a considerable waste of computational resources. Spectral element methods overcome this drawback and have proven very successfully their efficiency since they can accommodate irregular domains [13,14]. As an alternative, patching multidomain decomposition technique is also attractive. It may maintain the spatial and temporal accuracy and appeared to some authors necessary to consider more complex geometries [15], atypical boundary conditions [16] or higher Reynolds numbers [17] when coupled with a parallelization strategy to overcome the very thin meshes required for turbulent flow regimes. This technique consists of splitting the fluid domain into a set of subdomains with appropriate transmission conditions. The patching multidomain methods can be divided into two classes:

- iterative methods with or without overlapping between subdomains in which the continuity is ensured by successive internal iterations,
- direct methods in which the continuity is enforced at each iteration (continuity influence matrix method).

For the first class, Morchoisne [18] solved the Navier–Stokes equations in the vorticity-stream function formulation. Louchart and Randriamampianina [19] solved the 2D Navier–Stokes equations written in the velocity–pressure formulation using a spectral method and a projection algorithm for the temporal discretization. They introduced a relaxation parameter whose optimal value has to be tuned. More recently, Sengupta et al. [20] proposed a new class of compact schemes applied