Journal of Computational Physics 272 (2014) 108–126

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp

A new high-order method for the simulation of incompressible wall-bounded turbulent flows

Peter Lenaers *, Philipp Schlatter, Geert Brethouwer, Arne V. Johansson

Linné FLOW Centre, KTH Mechanics, Osquars Backe 18, SE-100 44 Stockholm, Sweden

ARTICLE INFO

Article history:
Received 20 March 2013
Received in revised form 12 April 2014
Accepted 17 April 2014
Available online 24 April 2014

Keywords:
Incompressible wall-bounded turbulent flows
High-order
Compact finite differences
Collocated grid
Influence matrix method

ABSTRACT

A new high-order method for the accurate simulation of incompressible wall-bounded flows is presented. In the stream- and spanwise directions the discretisation is performed by standard Fourier series, while in the wall-normal direction the method combines high-order collocated compact finite differences with the influence matrix method to calculate the pressure boundary conditions that render the velocity field exactly divergence-free. The main advantage over Chebyshev collocation is that in wall-normal direction, the grid can be chosen freely and thus excessive clustering near the wall is avoided. This can be done while maintaining the high-order approximation as offered by compact finite differences. The discrete Poisson equation is solved in a novel way that avoids any full matrices and thus improves numerical efficiency. Both explicit and implicit discretisations of the viscous terms are described, with the implicit method being more complex, but also having a wider range of applications. The method is validated by simulating two-dimensional Tollmien–Schlichting waves, forced transition in turbulent channel flow, and fully turbulent channel flow at friction Reynolds number $Re_\tau = 395$, and comparing our data with analytical and existing numerical results. In all cases, the results show excellent agreement showing that the method simulates all physical processes correctly.

1. Introduction

In the last few decades, Direct Numerical Simulations (DNS) where all scales of motion are resolved, have proven very useful to investigate the features and properties of incompressible wall-bounded turbulent flows. The simulation of these flows is a challenging area with high demands on the accuracy and efficiency of the code, which are amplified by the ever-continuing need for simulating flows at higher Reynolds numbers [1]. A major issue when solving the governing three-dimensional incompressible Navier–Stokes equations is the lack of an evolution equation for the pressure. Instead, the pressure is present in the momentum equations and instantaneously corrects the velocities such that the continuity equation is satisfied, i.e. the divergence of the velocity field is equal to zero. Different methodologies have been developed to deal with this issue.

Probably the most popular approach is the pressure correction or fractional step method [2]. Here, the integration over one time step is split into three parts. In the predictor step, an intermediate velocity field is calculated without taking into account the pressure. Secondly, a Poisson equation for the pressure is solved. Lastly, in the corrector step, the intermediate velocity is projected by the pressure onto a divergence-free field. However, even though the method is widespread, there

* Corresponding author. Tel.: +46 8 790 7195.
E-mail address: Lenaers@mech.kth.se (P. Lenaers).

http://dx.doi.org/10.1016/j.jcp.2014.04.034

are still unresolved issues regarding the choice of boundary conditions for the pressure when solving the Poisson equation. Usually, boundary conditions are derived by extrapolating the velocities and pressure gradients from previous time steps [3,4]. This introduces splitting errors in the integration scheme reducing the maximum allowable time step that ensures numerical stability and thus decreasing the efficiency of the method. The method is mainly applied on a staggered grid [5,6] to avoid checkerboard patterns in the solutions stemming from odd–even decoupling. Although a staggered grid has proven very useful in some cases, it also has its disadvantages, namely that it requires frequent interpolation, which can introduce unwanted filtering and complicate implementation.

One way to avoid this problem is to write the Navier–Stokes equations in vorticity–velocity (VV) form [7–10] by taking the curl of the momentum equations. This eliminates the pressure and gives a set of evolution equations for the vorticity and a set of elliptic equations relating the vorticity to the velocities. Boundary conditions for the vorticity are required but not known, such that similar issues arise as in the fractional step method where boundary conditions for the pressure are unknown.

A third method is the influence matrix (IM) method [11–13]. In this case as well, a Poisson equation for the pressure is derived that replaces the continuity equation in the interior of the flow domain. For problems with non-periodic boundaries in one dimension, this results in a sequence of one-dimensional scalar Helmholtz equations, which is solved to calculate the pressure boundary conditions that after applying a correction step render the entire velocity field divergence-free. The advantage of this method is that continuity is fulfilled exactly in the discretised equations. It can also be applied on a collocated grid, thus avoiding interpolation that can cause unwanted filtering effects. Reuter and Rempfer [14] use this method to simulate turbulent pipe flow (although they make no mention of the correction step), while Tuckerman [15] derives a generalised and more formal method for other geometries. When comparing the IM method with the VV method, we see that both have the issue of missing boundary conditions, for the pressure in the former, for the vorticity in the latter method. The commonly used method to calculate the missing boundary conditions is similar for the two methods. An advantage of the IM method is that the pressure is one of the unknowns and thus follows immediately from the calculation, while in the VV method an extra equation needs to be solved to obtain the pressure. Furthermore, the IM method is more straightforward to extend to other geometries. The VV method has been found to give stability issues when using cylindrical coordinates to simulate turbulent pipe flow [16].

All of the examples mentioned implement the IM method with Chebyshev polynomials in the wall-normal direction. Although the use of Chebyshev collocation is widespread in the simulation of wall-bounded flows, it also has its restrictions. With simulations of wall-bounded flows being performed at ever rising Reynolds number ($Re_\tau = 10000$ should be reached in the foreseeable future [17]), these restrictions are being exposed and we feel there is a need for a new code that is not subject to these restrictions. The restriction is that the prescribed grid when using Chebyshev collocation is the Gauss–Lobatto–Chebyshev grid which follows a cosine distribution. The high resolution required to resolve all scales at high Reynolds number simulations causes an extremely clustered grid near the wall for this grid. For example, 3841 Gauss–Lobatto–Chebyshev points are required for a simulation at $Re_\tau = 10000$ if a maximum spacing of 8 viscous units at the centre of the channel is prescribed. This grid has 56 points within the first 10 viscous units near the wall and a minimum spacing of $\Delta y^+_{min} = 0.0034$. As a result the maximum allowable time step becomes so small that it is not feasible any more to run a simulation for a sufficient amount of time. Because of the numerical issues caused by extreme clustering of gridpoints, there exists a desire to have more freedom in the allocation of the grid points in the wall-normal direction. An alternative is to use compact finite differences on a staggered grid, but this requires frequent interpolation that might cause unwanted filtering effects. Therefore, we use compact finite differences on a collocated grid. Compact finite difference schemes are summarised by Lele [18] and show good resolution characteristics over a large range of wavenumbers while maintaining the freedom to choose the grid points and boundary conditions. Because this grid can be freely chosen, extreme clustering can be prevented and thus the time step restriction is not as severe.

The viscous terms can be treated explicitly or implicitly. The equations are simpler when they are treated explicitly, but this does impose more severe restrictions on the maximum allowable time step in certain flow cases. Kleiser and Schumann [12] use an implicit discretisation for the viscous terms to avoid this severe time step limitation, while Simens et al. [19] suggest that in a turbulent boundary layer the viscous time step limit is only critical in the wall-normal direction, so that is the only direction they treat implicitly. Thus, they treat the viscous terms in the stream- and spanwise directions explicitly. Akselvoll and Moin [20] give a clear dissection of the terms that can be treated explicitly and the terms that need to be treated implicitly in different parts of the domain (near the axis and near the wall) when simulating turbulent pipe flow. We describe a general method of which the explicit scheme is a special case. The results and numerical efficiency of both methods are shown and discussed in separate sections.

In this paper, we present a new method to simulate wall-bounded flows. The aim of this method is to simulate these flows at high Reynolds number in an efficient way. We extend the IM method to allow the use of compact finite differences, which gives the user the freedom to choose the location of the grid points and thus providing more flexibility. To validate the method we perform low-Reynolds number simulations and compare with analytical and existing results.