Testing Auxiliary Pointers to Stabilize the Numerical Simulations

A. Jirk and J. Brechler
Charles University Prague, Faculty of Mathematics and Physics, Prague, Czech Republic.

Abstract. In this contribution we describe simulations of laminar and turbulent incompressible flow by the created model which is tested for several significant influences, such as stability coefficients (CFL), different computational areas, staggered grids etc. To solve the equation system on limited area the finite control volume method is employed. The initial value problems and boundary conditions on control volumes are described and discussed in some numerical examples, e.g. lid-driven cavity flow and three-dimensional Taylor-Green vortex (TGV) for the low and high Reynolds numbers. The used model consists of the Navier–Stokes equations for the laminar flow and the continuity equation in the form for incompressible fluid flow. The conservative high order methods are used for solving the system of the equations describing the conservation laws. The advection terms are reconstructed using the 5th order weighted essentially non oscillatory (WENO) scheme and the time evolution is solved by the application of the 3rd order explicit TVD Runge-Kutta scheme. The implicit large eddy simulation (ILES) was employed as a turbulent model which is implicitly added to the system by WENO scheme.

Introduction

Any atmospheric flow within the atmospheric boundary layer is turbulent. This fluid flow is described by the equations fulfilling the conservation laws. The exact solution is unknown so the numerical methods have to be employed. Many numerical schemes can be applied for solving the system, but more sophisticated method should be used to resolve correctly the turbulence. In the contribution, we focus on the problem of numerical simulations of fluid flow in 2D and 3D.

Hence the higher-order accuracy methods are required in order to capture both the large- and small-scale structures. In 2D the simpler methods are used, e.g. for resolving viscous term and the model of turbulence is not necessary. With the transition to three dimensions a lot of non trivial upgrades must be made to catch better stability and convergence of the solution.

To prevent the occurrence of undesired spurious oscillations in our numerical modeling, we employed the finite volume approach with higher-order (fifth-order) WENO reconstruction. For temporal discretization, we employed the explicit TVD (Total-Variation diminishing) Runge–Kutta (R–K) scheme. For splitting computational domain the finite volume method is used. As the turbulent model the ILES method is employed that goes hand in hand with WENO schemes. It is a form of Large Eddy Simulations (LES) in which the large energy containing structures are resolved, whereas the smaller, more isotropic, structures are filtered out and, therefore, their effects need to be modeled.

For testing stability coefficient and convergence of the solution we used cases with well-known results from literatures. 2D lid-driven cavity flow [Ghia et al., 1982] and flow past an obstacle [Brauer, 2000] are computed, and there are found some problems with boundary condition, especially solid walls and free outflow conditions. The upgrades of conditions in 2D are easier to change.

In 3D there are employed cases the flow past an obstacle and the fundamental case The Taylor-Green Vortex (TGV) that has been traditionally used as prototype for vortex stretching and the consequent production of small-scale eddies, to investigate the basic dynamics of transition to turbulence [Drikakis et al., 2006; Don et al., 2002]. For transition to higher dimension the Crank Nicholson method is employed which leads to better stability. There are improved boundary and input conditions.

Governing system

The equations representing the conservation laws of various quantities are used to describe a fluid motion. They are the Navier-Stokes equations (1) for momentum conservation. And these three equations are completed with the continuity relation (2). All these equations (1–2) are evolved in Jirk [2008] and they are written in a non-dimensional form.
\[ \frac{\partial v}{\partial t} + \frac{\partial v}{\partial x} = -\frac{\partial p}{\partial x} + \frac{1}{Re} \Delta v \tag{1} \]
\[ \frac{\partial v}{\partial x} = 0 \tag{2} \]

**Numerical method**

For the spatial discretization of the governing equations (1–2) the finite volume method is used [see, for example, Ferziger and Perić, 1997; McDonough, 2003]. The advection terms in (1) are reconstructed by WENO scheme [Liu et al., 1994] that was extended from ENO scheme [Harten et al., 1987; Shu and Osher, 1988]. A key idea in WENO scheme is a linear combination of lower order fluxes or reconstruction to obtain a higher order approximation. Both ENO and WENO schemes use the idea of adaptive stencils to automatically achieve high order accuracy and non-oscillatory property near discontinuities. In this work the WENO scheme of the fifth order accuracy has been used [Titiarev and Toro, 2004]. For the computation of the temporal partial derivation in (1) there has been used the explicit TVD Runge-Kutta scheme of the third order accuracy [Gottlieb and Shu, 1998]. This scheme has CFL=1 (It is stability condition between time step and length interval, \( \Delta t \leq CFL \Delta x \)). For computing of viscous terms in (1) there is applied explicitly 2nd derivation in 2D and in 3D the Crank-Nicholson method [Kim et al., 2001]. The fractional-step method [Brown et al., 2001] has been used for solution of the Navier-Stokes equations (1) and continuity relation (2). This approach un-groups the solution of equations into several steps. In this work the three step method has been applied. For the simulation of the obstacle the second order accuracy immersed boundary method [Kim et al., 2001] is implemented.

**Results, discussion**

2D and 3D cases are used for the investigation of good stability and right convergence of the system. There is also discussed the usage of boundary and input conditions and different numerical methods.

**2D Lid-driven cavity flow**

This case was computed on the grid \(161^2\) and for the Reynolds number 1000. The Dirichlet boundary conditions are displayed in Figure 1. In Figure 2 there are shown two solid wall conditions applied to control volumes (CV). They are shifted by half of CV against each other. In 2D (this case) the left method in Figure 2 is used, second approach is implemented in 3D (next cases). There are no problems with the stability but in higher dimension there had to be used the second approach because of higher accuracy. The interpolation and derivation in CV are derived from Taylor expansion.

In the left hand side of Figure 3 the stabilized situation is shown. This result is set after a long computation, except primary vortex, the secondary and tertiary vortices are pronounced thanks to the influence of walls. Additional vortices are formed close the borders with the increase of Reynolds number.

The positions of vortices are measured and compared with the literature. The results are in a good agreement in the interval of error \(\pm 0.006\) (Table in Figure 3).

**Flow past an obstacle**

In this case the flow past a cube is modelled. Computational area has proportions in dimensionless form: length \(L=50\), widths \(H=12\). The obstacle is located in the middle of computed area in distance 12 from the inflow. As entry condition there is used the Dirichlet one \(v=w=0, u=1\), at the exit of flow the Neumann...
condition \( \frac{\partial u}{\partial x} = \frac{\partial v}{\partial x} = 0 \) and in the sides of computed area the periodic boundary conditions. The spatial step is 0.125. Reynolds number is chosen as 1000. In this case there is a problem how to describe the outflow condition because of the reflexes from the rear border. In Figure 4 there are 2 approaches how to handle with the derivation at the outflow boundary. The second one suppresses these reflexes.

The von Karman vortex street is pronounced behind the obstacle how it is shown in Figure 5. There are depicted results for Re 1000 and compared the Strouhal number (the frequency of vortexes) with literature. The results are 10 % lower than in literature. It can be caused by coarse grid and unused the law of wall.

Figure 2. Usage of boundary conditions, the whole CV is equal to zero (left) and solid wall leads in the middle of CV (right).

Figure 3. Lid-driven cavity case, Re 1000, the streamlines (left) and positions of vortexes (right).

Figure 4. Description of derivation on the outflow border.

Figure 5. Flow past an obstacle, velocity \( v^2 \) (left), Strouhal numbers compared with literature (right).
Taylor-Green vortex case

In 3D cases the Crank-Nicholson method is employed instead of explicitly given 2nd derivation. It provides almost 10 times better stability. It is important for computational time. The initial conditions for TGV are written in (3):

\[
\pi(t = 0) = \begin{bmatrix}
\sin(x) \cos(y) \cos(z) \\
0 \\
-\cos(x) \sin(y) \cos(z)
\end{bmatrix}
\]

Configuration of case involves triple-periodic boundary conditions enforced on a cubical domain with box side length \(2\pi\) using 1603 evenly spaced computational cells. The periodic condition must be used also for pressure variable. The usage of the Neumann conditions (pressure is constant on borders) as in 2D leads into instabilities and divergence of the solution. The derivations and interpolations are compute with the step \(\Delta x/2\) (see Figure 6). It provides better stability and it is more accurate approach as already mentioned above (lid-driven cavity case).

The evolution in time of the kinetic energy \(E_K\) is shown in Figure 7 as the dependence on Reynolds number, where \(E_K = 1/2 < u^2 >\), \(< >\) denotes mean (volumetric average) and \(u\) is velocity vector.

Variation of Reynolds numbers makes differences in the decreasing of normalized kinetic energy. The fastest decrease is for \(Re = 500\) and the slowest for \(Re = 5000\). The decrease of the Kinetic energy continues also after time \(t=7.5\), in which the computation is stopped.

Flow past an obstacle in 3D

The flow past an obstacle in 3D is the extension of 2D case discussed above (flow past an obstacle in 2D). At the borders there are used the periodic conditions. The input and outflow conditions are issues in this case. In 2D there is used velocity equal to zero as input condition in the whole area except the inflow. In 3D this approach leads into instabilities. It helps use the same values as inflow for \(Re < 5000\) as input condition. For higher values it is used the flow without obstacle for \(Re = 5000\) as the input conditions. There are problems also with the outflow. The approach used in 2D doesn’t suppress reflexes in higher dimension. There is employed the buffer function [Guerits, 2003]. It straightens the velocities components. Results are depicted in Figure 8 for \(Re = 1000\) and \(100 000\), vortex streets behind the obstacle are pronounced.
Conclusion

We employed a fifth-order WENO reconstruction of the convective terms of Navier-Stokes equations and to compute 2D and 3D incompressible flow motion. The tested cases were modeled for different meshes, CFL’s and Reynolds numbers. In all cases there had to be made improvements to reach better stability and convergence to the right solutions. It is clear that it doesn’t work as a universal approach to all cases. It is necessary to upgrade and change the schemes from case to case. The right usage of the grid is also important.

As 2D cases the lid driven and flow past an obstacle were used. As turbulent flow examples, we computed flow past a cube for several Reynolds numbers. Taylor-Green vortex case was solved for some values of Re numbers and CFL stability conditions. With the decreasing values of Reynolds numbers the faster decay of kinetic energy was observes. All the results were compared with the literature with a good agreement.

Acknowledgments. This research was supported by the Czech Ministry of Education, Youth and Sports under the framework of Research Plan MSM0021620860.

References