Numerical Modelling with the Usage of High Order Accuracy Methods

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

Abstract. In this contribution we consider the application of the WENO scheme and the ILES method to simulations of 3D laminar and turbulent incompressible flow. As a test problem a 3D lid-driven cavity flow was modelled. The results of the three-dimensional Taylor-Green vortex example were carried out. For the low and high Reynolds numbers the case of flow past an obstacle were simulated. The used model consists of the Navier–Stokes equations for the laminar flow and the continuity equation in the form for incompressible fluid flow. To solve the system of the equations the conservative high order methods were used. 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 explicit TVD Runge-Kutta scheme. The ILES, as a turbulent model, was employed.

Introduction

Any atmospheric flow within the atmospheric boundary layer is turbulent. In this contribution, we focus on the problem of laminar and turbulent flow in 3D.

High-order accuracy is required in the simulation of turbulence in order to capture both the large- and small-scale structures. 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. The turbulent model ILES 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 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.

To test the applicability of this approach, we chose a problem involving flow in a cavity, around a cube in a channel. And a 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) and (Don et al. 2002). The lid-driven cavity flow case was computed for high Reynolds number. In the flow past an obstacle, we modelled for several values of Reynolds numbers. The TGV problem is solved for different grids, Courant–Friedrichs–Lewy (CFL) computational stability conditions and Reynolds numbers.

Governing system

To describe a fluid motion the equations representing the conservation laws of various quantities are used. 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.

\[
\begin{align*}
\frac{\partial v_i}{\partial t} + \frac{\partial v_i v_j}{\partial x_j} &= -\frac{\partial p}{\partial x_i} + \frac{1}{Re} \Delta v_i + \frac{\Delta v_j}{Re} \\
\frac{\partial v_i}{\partial x_i} &= 0
\end{align*}
\]

Numerical method

For the spatial discretization of the governing equations (1–2) the finite volume method is used (see, for example, Ferziger, Perić 1997 or 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) and (Shu, 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 (Titarev, 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, 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 used 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

In this contribution the three cases of 3D non-linear flow have been simulated. The first one is a lid-driven cavity flow, the second case deals with the flow past an obstacle and the third one is Taylor-Green vortex problem.

3D Lid-driven cavity flow

The sketch describing the cross-section xz geometry of the solved problem is displayed in Figure 1 together with the boundary conditions used.

The boundary conditions are stated as the Dirichlet boundary condition for the components of velocity, \( v=w=0 \) on all boundaries, \( u=0 \) in the side and bottom boundaries, \( u=1 \) on the top boundary. The computations were carried out for the \( Re \) equal to 10 000.

In Figure 2, there are depicted cross-section xz of flow field in time \( t=36 \) for 220^3 computational cells described with streamlines on the left and y-component of vorticity on the right. The vectors of velocity in 3D view and xz cross-section are shown in Figure 3.

Flow past an obstacle

In this case it is modelled 3D flow past a cube, which is located in the middle of computed area (area has these proportions in dimensionless form: length \( L=4 \) with widths \( H_2=12 \) and \( H_3=12 \) in distance 12 from the inflow. As entry conditions there is used Dirichlet condition \( v=w=0 \), \( u=1 \). At the exit of flow are stated Neumann conditions \( \frac{\partial u}{\partial x} = \frac{\partial v}{\partial x} = 0 \). In the sides of computed area there are used periodic boundary conditions.

Figure 1. Configuration used in lid-driven cavity problem and coordinate system, cross-section x-z.

Figure 2. Lid-driven cavity case, cross-section xz, medium cuts, streamlines (left) and y-component of vorticity (right).
Figure 3. Lid-driven cavity case, cross-section xz velocity vectors, medium cut (left) and 3D velocity view (right).

Figure 4. Flow past an obstacle, cross-section xz, medium cut, y-component of vorticity for Re 200.

Figure 5. Flow past an obstacle, cross-section xz, medium cut, y-component of vorticity: Re 5000 (left) and Re 20000 (right).

The spatial step is 0.125. Reynolds numbers are chosen 200, 5000 and 20 000.

In Figure 4, there is depicted stable flow for Re 200. With the compare of 2D of the same case stable flow is pronounced for Re 30 and smaller. It was also computed for higher Reynolds number 5000 and 20 000 and the results are shown in Figure 5. In contrast with case of low Reynolds number the vortex street behind obstacle is pronounced. All figures are shown in non-dimensional time $t=50$.

Taylor-Green vortex case

Configuration of this case here involves triple-periodic boundary conditions enforced on a cubical domain with box side length $2\pi$ using $160^3$ evenly spaced computational cells. The initial conditions for this flow are written in (3):

$$
\vec{u}(t = 0) = \begin{bmatrix}
\sin(x)\cos(y)\cos(z) \\
0 \\
-\cos(x)\sin(y)\cos(z)
\end{bmatrix}
$$

Computation is executed for several values of Reynolds number. In Figure 6, there is depicted decay of the total vorticity in time for Re 5000.

The evolution in time of the kinetic energy $K$, where $K = 1/2 \langle u^2 \rangle$, $\langle \rangle$ denotes mean (volumetric average) and $u$ is velocity vector, is demonstrated in Figure 7 as the dependence of several values of CFL for Re=1000 and in Figure 8 in the dependence on Reynolds number.
**Conclusion**

We employed a fifth-order WENO reconstruction of the convective terms of Navier–Stokes equations and the method ILES to compute 3D incompressible, turbulent flow motion. The three tested cases were modeled for...
different meshes, CFL’s and Reynolds numbers. As turbulent flow examples, we computed lid-driven cavity flow for Re=10,000 with well developed vortexes. Next we computed flow past a cube for several Reynolds numbers. For low value of Re the flow became steady. For high Re’s the vortex street behind the obstacle were pronounced. Taylor-Green vortex case was solved for some values of Re numbers and stability conditions CFL’s. With the decreasing values of Reynolds numbers the faster decay of kinetic energy was observed. For the lowest value of CFL the solution was different against other used CFL’s.

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

References