УДК 519.63; 519.684 A.Issakhov

DIRECT NUMERICAL SIMULATION (DNS) OF TURBULENT FLOWS BY USING PARALLEL TECHNOLOGY

Nowadays the turbulence remains one of the priority areas of science and one of the most complex objects of study aero- and hydrodynamics. For more than a century of study were invited a lot of different techniques and approaches that were promising directions of the relevant period of time. There are new approaches to its study, the number of models and approaches proposed for a better understanding of its properties, as well as the mechanisms of its origin and existence.

The need for study of turbulent flows due to the fact that they are the dominant form of movement, both in nature and in technology. The presence of turbulence in technical devices have a strong influence on the performance, durability and other important characteristics of the structures. Therefore, the study of unsteady phenomena characteristic of turbulent flows can explain the processes occurring in them, and greatly facilitate the work on creation of new devices /10,13/.

There are many approaches to modeling the turbulence. One of the most well-known approaches - the use of a semi-empirical models that are based on the hypothesis of the local Reynolds averaging by time of hydro-mechanical parameters of the flow. These models are used for the closure of the system of equations of various algebraic or differential eddy viscosity model, containing a number of empirical constants whose values need to vary in each case. A large amount of numerical studies using this approach will significantly clarify the picture of the processes taking place in a turbulent flow. These turbulence models continue to evolve at this time.

Recently, for the calculation of turbulent flows is intensively used LES (Large Eddy Simulation) modeling - modeling of large eddies. The idea of this method is to calculate the large-scale three-dimensional unsteady turbulent flow using the procedure of spatial filtering, which highlights the major vortex formation. This approach ignores the influence of small-scale turbulence on the flow pattern, so it is often used together with the subgrid model of turbulence (SGS model) /3/.

One of the priority areas nowadays, the calculation of turbulent flows is the numerical simulation based on the construction of difference methods for the calculation. A significant place in modern research has a direct numerical simulation (DNS), i.e. three-dimensional simulation of direct calculations of the program, solving the Navier - Stokes equations, without using any special turbulence models. The complexity of direct numerical simulation is due primarily to the fact that unsteady turbulent flows are characterized by a wide range of spatial and temporal scales.

Therefore, the calculations required for high-performance computing system and efficient numerical method to obtain reliable numerical results. It should be noted that the methods, which have a low order of approximation of spatial derivatives have a significant dissipation of the circuit and to obtain reliable results with these schemes require refinement of the difference grids, and as a consequence of an increase in machine time.

This work is devoted to direct numerical simulation using finite volume method. In the problems of turbulence is necessary to construct an irregular grid in order to explore the region with large gradients. For the direct numerical simulation of turbulence was more effective approach using approximation calculation of domains of different structures of regular and irregular meshes, the approximation that makes it easier to carry out concentration of cells in separate subdomains with large gradients. In this case, the original equation is more convenient to choose in integral form and approximate them to the design grid, consisting of cells of different shapes. The approximation equations in integral form on a regular and not regular grids are preferred. Approximation of the basic equations in integral form, based on the finite volume method /3,9/ leads to a system of linear and nonlinear algebraic equations /6,10,13/.

Equations

We are taking Navier - Stokes equations in the form of integral conservation laws for an arbitrary fixed Ω with a boundary $\partial\Omega$:/1,2,3,4,5,11,12/

$$\int_{\Omega}\left(\frac{\partial U}{\partial t}+\frac{\partial F_i}{\partial x_i}+\frac{\partial G_i}{\partial x_i}-B\right)d\Omega=0 \qquad (1)$$

where

$$U=\begin{pmatrix}0\\ v_j\end{pmatrix} \quad F_i=\begin{pmatrix}v_i\\ v_iv_j+p\delta_{ij}\end{pmatrix} \quad G_i=\begin{pmatrix}0\\ \tau_{ij}\end{pmatrix} \quad B=\begin{pmatrix}0\\ F_j\end{pmatrix}$$

We can write equation (1) like that

$$\int_{\Omega}\left(\frac{\partial U}{\partial t}-B\right)d\Omega+\oint_{\partial\Omega}(F_i+G_i)n_i d\Gamma=0 \qquad (2)$$

In our case B=0 and equation (2) writing like this

$$\int_{\Omega}\left(\frac{\partial U}{\partial t}\right)d\Omega+\oint_{\partial\Omega}(F_i+G_i)n_i d\Gamma=0 \qquad (3)$$

Grid functions will be determined at the center of the cells, and the value of flows across the border in fractional cells. Cell volume is denoted by grid functions.

Now we perform a discretization of equation (3) of the control volume (CV) and the control surface (CS)

$$\sum_{CV}\left(\frac{\Delta U}{\Delta t}\right)\Delta\Omega + \sum_{CS}(F_i + G_i)n_i\Delta\Gamma = 0 \tag{4}$$

or we can write like that

$$\sum_{CV}\Delta U\Delta\Omega + \sum_{CS}\Delta t(F_i + G_i)n_i\Delta\Gamma = 0 \tag{5}$$

Numerical algorithm

To solve equation (5) used a scheme of splitting by physical parameters. The first stage is assumed that the transfer of momentum is only due to convection and diffusion. The intermediate velocity field found by 5-steps Runge - Kutta method /3/. At the second stage, by found an intermediate velocity field, is found the pressure field. The Poisson equation for the pressure field is solved by Jacobi method /2,3,12/. In the third stage it is assumed that the transfer is carried out only by the pressure gradient. The numerical algorithm parallelized in a high-performance system with a 2D domain decomposition /7,8,14,15/.

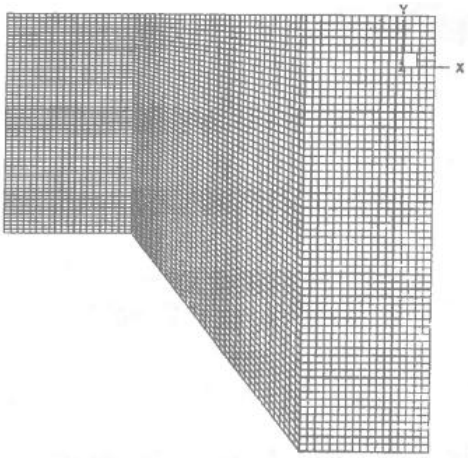
I) $$\int_{\Omega}\frac{\vec{u}^{*}-\vec{u}^{n}}{\tau}d\Omega = -\oint_{\partial\Omega}\left(\nabla\vec{u}^{n}\,\vec{u}^{*} - \nu\Delta\vec{u}^{*}\right)n_i d\Gamma$$

II) $$\oint_{\partial\Omega}(\Delta p)d\Gamma = \int_{\Omega}\frac{\nabla\vec{u}^{*}}{\tau}d\Omega$$

III) $$\frac{\vec{u}^{n+1}-\vec{u}^{*}}{\tau} = -\nabla p.$$

Computational setup

The grid is irregular and regular. Figure 1 shows a two-dimensional grid view, and figure 3 shows a three-dimensional grid view



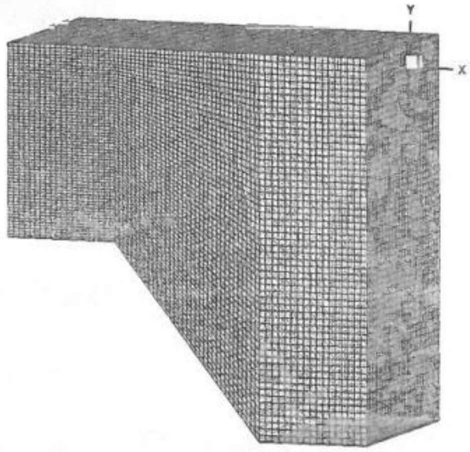


Figure1 Two – dimensional grid view Figure 2 Three – dimensional grid view

Results

To solve the problems have been given initial and boundary conditions. In the calculations used the mesh with size 72x60x41. Figure 3 shows the isosurface of distribution of kinetic energy at different times. The results show that the kinetic energy is distributed over a larger area, despite the fact that the Reynolds number is very small (Re = 500). Over time, a dynamic field decays, and the field energy is converted into a stationary state. From figure 3 we see that with direct numerical simulation can embrace a wide range of eddies.

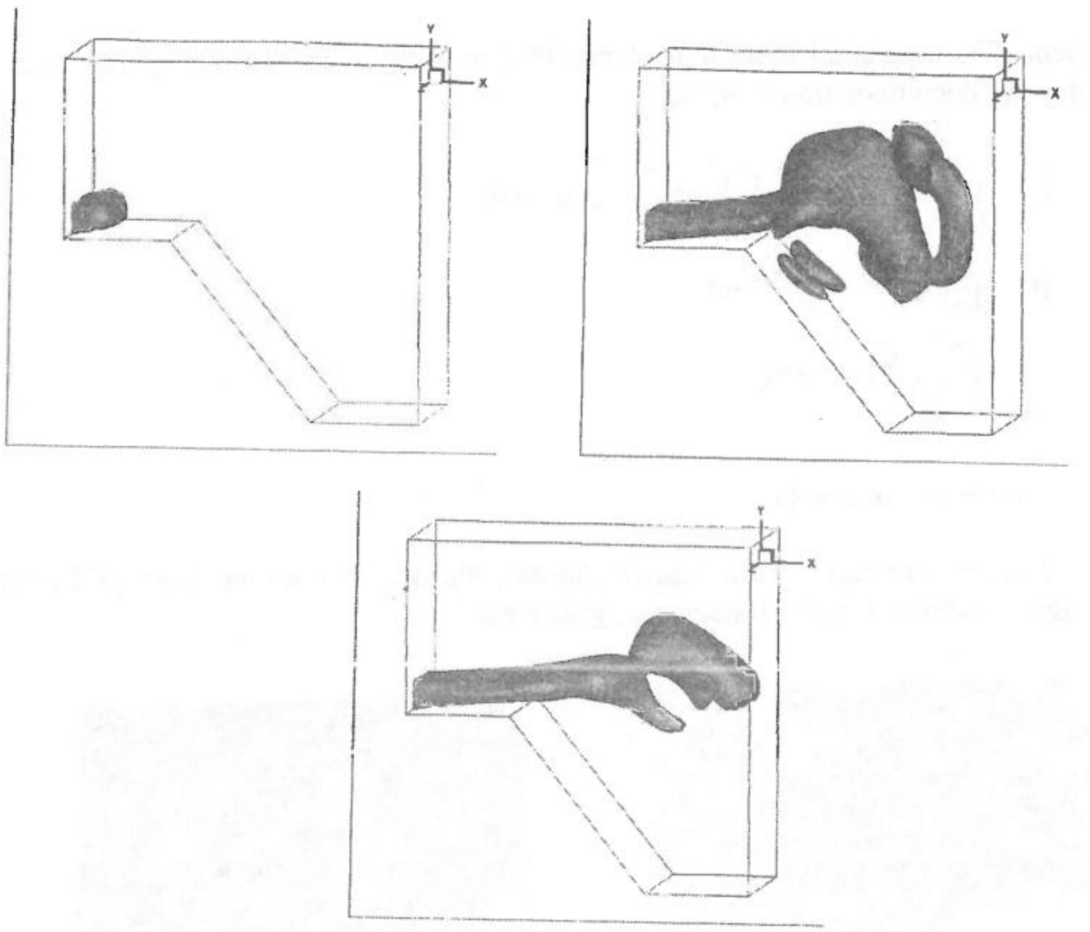


Figure 3 Distribution of isosurface of the kinetic energy at different times

References

1. Anderson, J.D., Jr. Computational Fluid Dynamics. New York: McGraw-Hill. 1995

2. Anderson, D.A., Tannehil, J.C., and Pletcher, R.H. Computational Fluid Mechanics and Heat Transfer. New York: McGraw-Hill. 1984

3. Chung, T.J. Computational Fluid Dynamics. Cambridge university press. 2002

4. Fletcher, C.A. Computational Techniques for Fluid Dynaimics. Vol 1: Fundamental and General Techniques, Berlin: Springer-Verlag. 1988

5. Fletcher, C.A. Computational Techniques for Fluid Dynaimics. Vol 2: Special Techniques for Differential Flow Categories, Berlin: Springer-Verlag.[1] 1988

6. Hinze, J.O. Turbulence. An introduction to its mechanism and theory. New York: McGraw-Hill. 1959

7. Karniadakis, G. E. Parallel Scientific Computing in C++ and MPI. 2000

8. Pacheco, P.Parallel Programming with MPI. Morgan Kaufmann. 1996

9. Peyret, R., Taylor, D. Th. Computational Methods for Fluid Flow. New York: Berlin: Springer-Verlag. 1983

10. Pope, S. B.Turbulent Flows. Cambridge University Press. 2000

11. Roache, P.J. Computational Fluid Dynamics, Albuquerque, NM: Hermosa Publications. 1972

12. Tannehill, J.C., Anderson, D.A., and Pletcher, R.H. Computational Fluid Mechanics and Heat Transfer. 2nd ed., New York: McGraw-Hill.[2] 1997

13. Tennekes, H., Lumley, J.L. A first course in turbulence. The MIT Press. 1972

14. Issakhov A. Large eddy simulation of turbulent mixing by using 3D decomposition method Issue 4 (2011) J. Phys.: Conf. Ser. 318 042051

15. Issakhov A., Zhumagulov B. Parallel implementation of numerical methods for solving turbulent flows Вестник НИА РК. – 2012, – № 1 (43) – С.12-24