COMPUTATIONAL MECHANICS
ISCM2007, *July 30-August 1, 2007, Beijing, China*

Three-Dimensional Coupling Compact Finite Difference Methods for Navier-Stokes Equations

Li Zhang[1]*, Dengbin Tang [2], Yingzhao Yang[1]

[1] *Thirteenth Flight Academy of Air Force, Bengbu, 233000 China*
[2] *College of Aerospace Engineering, Nanjing University of Aeronautics and Astronautics, Nanjing, 210016 China*

Email: bbzhli6@sohu.com

Abstract The higher-order three- dimensional coupling compact finite difference methods are proposed for solving the three- dimensional time-dependent incompressible Navier-Stokes equations in the paper. And then the generation, development and evolution of the turbulent spots in the channel flow are directly simulated using this difference method. The results are in agreement with the experiment and prove availability of this difference method.

Key words: Navier-Stokes equations, turbulent spots, numerical method, three-dimensional coupling finite difference scheme, turbulence

INTRODUCTION

Navier-Stokes equations are main governing equations and one of principal tools analyzing fluid flow in fluid mechanics. Nevertheless, owing to it is nonlinear partial differential equations, therefore, only at the simple flow condition can its analytic solution be found. The complex question is solved usually by means of numerical methods [1]. Present, direct numerical simulation of turbulence with a range of space and time is developing quickly [2]. These requirements have led to the quick development of numerical schemes for solving Navier-Stokes equations with higher accuracy and higher resolution. In the paper, the fourth-order three-dimensional coupling compact finite difference schemes [3] for the Poisson equation and Hemholtz equation, which not only have higher accuracy and resolution, but also applies to calculate the near boundary nodes, are presented, thereby overcomes the difficult of the general higher-order central finite difference scheme not applied to the near boundary nodes. Combining these coupling compact finite difference schemes with the time splitting method [4], the compact finite difference methods for the time-dependent incompressible Navier-Stokes equations are constructed. This finite difference method can be applied to general boundary conditions and flows in more complex domains. And the generation, development and evolution of the turbulent spots induced by ridged wall are directly simulated using this difference method, and prove availability of this difference method.

NUMERICAL METHOD

The non-dimensional three-dimensional time-dependent incompressible Navier-Stokes equations are as fellows:

$$\frac{\partial U}{\partial t}+(U\cdot\nabla)U=-\nabla p+\frac{1}{\mathrm{Re}}\nabla^2 U \tag{1a}$$

$$\nabla \cdot U = 0 \tag{1b}$$

In equations (1) and (2), $U = (u,\ v,\ w)^{\mathrm{T}}$, is the nondimensional velocity vector; p and t are the nondimensional static pressure and time respectively; Re is Reynolds number.

We will present the fourth-order compact difference methods for solving above equations and prove its availability by actual application in later part.

1. Temporal discretization The fourth-order time splitting methods are used in the temporal discretization of equation (1a), and the semi-discrete system of equations are given by

$$\frac{U' - \sum_{q=0}^{J-1} \alpha_q U^{n-q}}{\Delta t} = -\sum_{q=0}^{J-1} \beta_q (U^{n-q} \cdot \nabla) U^{n-q} \tag{2a}$$

$$\nabla^2 p^{n+1} = \frac{\nabla \cdot U'}{\Delta t} \tag{2b}$$

$$\frac{U'' - U'}{\Delta t} = -\nabla p^{n+1} \tag{2c}$$

$$\frac{\gamma_0 U^{n+1} - U''}{\Delta t} = \frac{1}{\mathrm{Re}} \nabla^2 U^{n+1} \tag{2d}$$

Where U' and U'' are the intermediate velocity fields, Δt is temporal step, n the number of time layer, α_q, β_q, γ_0 proper weight factors, J an accuracy parameter, $q = 0,\ 1, ..., J$. Where, J = 4, $\alpha_0 = 8$, $\alpha_1 = -6$, $\alpha_2 = 8/3$, $\alpha_3 = -1/2$, $\beta_0 = 4$, $\beta_1 = -6$, $\beta_2 = 4$, $\beta_3 = -1$, $\gamma_0 = 25/6$.

These schemes not only possess higher accuracy and better stability, but also have separate linear and nonlinear terms so that make numerically solving conveniently.

2. Spatial discretization Where, we will continuously make spatial discretization to above semi-discretized equations, and give solving methods.

2.1 *Equation* (2a): In this equation, the fourth-order upwind-biased compact difference scheme is used. We illustrate its discrete method instanced $u\partial u/\partial x$. Let

$$a_1 = (u + |u|)/2; \qquad a_2 = (u - |u|)/2$$

$$\delta^{\pm} u_i = \pm(u_{i\pm1} - u_i)$$

$$u^{\pm} = \frac{3u_{i\mp1} + 13u_i - 5u_{i+1} - 6u_{i\pm2} + u_{i\pm2}}{12h}$$

then

$$u\frac{\partial u}{\partial x} = a_1 \delta^- u^- + a_2 \delta^+ u^+ \tag{3a}$$

Other nonlinear items are done with same method; and the fourth-order explicit upwind-biased finite difference schemes on U' are obtained.

$$U' = \sum_{q=0}^{J-1} \alpha_q U^{n-q} - \Delta t \sum_{q=0}^{J-1} \beta_q (a_1 \delta^- U^- + a_2 \delta^+ U^+)^{n-q} \tag{3b}$$

The U' can be obtained by this scheme and given initial value U^n, U^{n-1} and U^{n-2}.

2.2 *Equation* (2b): This equation is common Poisson equation. Its general fourth-order central difference scheme is given by

$$\left(\frac{\delta_x^2}{h_x^2} - \frac{\delta_x^4}{12h_x^2}\right)p^{n+1} + \left(\frac{\delta_y^2}{h_y^2} - \frac{\delta_y^4}{12h_y^2}\right)p^{n+1} + \left(\frac{\delta_z^2}{h_z^2} - \frac{\delta_z^4}{12h_z^2}\right)p^{n+1} = f(x, y, z) \tag{4a}$$

Where h_x, h_y and h_z are the step sizes in x, y and z, respectively, δ^2 is a difference operator,

$\delta^2 p_i = p_{i-1} - 2p_i + p_{i+1}$, $\delta^4 = \delta^2 \cdot \delta^2$, $f(x,y,z) = \dfrac{\nabla \cdot U'}{\nabla t}$.

Although equation (4a) possesses fourth-order, it not can be used in the near boundary nodes. Therefore, the near boundary nodes must be calculated by using lower-order difference schemes, so the accuracy and resolution of whole systems are reduced that not can be used in direct numerical simulation on turbulence. For overcoming this difficulty, we propose fourth-order three-dimensional coupling difference schemes given by

$$(\frac{12\delta_x^2 + \delta_x^2\delta_y^2 + \delta_x^2\delta_z^2}{h_x^2})p^{n+1} + (\frac{12\delta_y^2 + \delta_x^2\delta_y^2 + \delta_z^2\delta_y^2}{h_y^2})p^{n+1}$$
$$+(\frac{12\delta_z^2 + \delta_z^2\delta_x^2 + \delta_z^2\delta_y^2}{h_z^2})p^{n+1} = 12f(x,y,z) + (\delta_x^2 + \delta_y^2 + \delta_z^2)f(x,y,z) \tag{4b}$$

This is a three-dimensional coupling fourth-order central difference scheme on Poisson equation, which not only has higher accuracy, but also has the higher resolution and applied to the near boundary nodes. The p^{n+1} can be found out by using equation (4b), and then U'' is found out by equation (2c).

2.3 *Equation* (2d): This equation is common Hemlholtz equation. Let

$$f = -\frac{\mathrm{Re}\, U''}{\nabla t}, \qquad k = \frac{\gamma_0 \,\mathrm{Re}}{dt}, \qquad f' = ku^{n+1} - f$$

then equation (2d) becomes

$$\nabla^2 U^{n+1} = f' \tag{5a}$$

This equation is similar to equation (2b), and can be studied using same method. Then it can be written as following discrete form:

$$\left(\frac{12\delta_x^2 + \delta_x^2\delta_y^2 + \delta_x^2\delta_z^2}{h_x^2}\right)U^{n+1} + \left(\frac{12\delta_y^2 + \delta_x^2\delta_y^2 + \delta_z^2\delta_y^2}{h_y^2}\right)U^{n+1}$$
$$+\left(\frac{12\delta_z^2 + \delta_z^2\delta_x^2 + \delta_z^2\delta_y^2}{h_z^2}\right)U^{n+1} - 12kU^{n+1} + k\left(\delta_x^2 + \delta_y^2 + \delta_z^2\right)U^{n+1} \tag{5b}$$
$$= 12f(x,y,z) + \left(\delta_x^2 + \delta_y^2 + \delta_z^2\right)f(x,y,z)$$

This is a three-dimensional coupling fourth-order central difference scheme on Hemlholtz equation. U^{n+1} can be found out by this equation.

Equation (3b), (4b), and (5b) have composed a equation systems for solving the Navier-Stokes equations. The boundary conditions and initial value will be given according to actual subjects.

NUMERICAL EXAMPLES

Where, we will directly simulate the development and evolution of the turbulent spots in the channel flow using above difference equation systems.

Simulations were conducted in domain that size is $15\hbar \times 2\hbar \times 4\hbar$ in stream, wall-normal and span direction respectively and on grids of $101 \times 101 \times 41$ nodes. ($\hbar$ is channel half-height). The time step is 0.015. The boundary conditions are given by

$$\left(\frac{\partial U}{\partial x}\right)_{x=0} = 0$$

$$\left[\frac{\partial U}{\partial t}+U\cdot\nabla(U+\bar{U})+\bar{U}\cdot\nabla U\right]_{x=15}=0$$

$$\left(\frac{\partial U}{\partial x}\right)_{z=0}=\left(\frac{\partial U}{\partial x}\right)_{z=4}=0,\qquad U_{y=0}=U_{y=2}=0$$

$$\left(p^{n+1}\right)_{x=0}=p_{01},$$

$$\left(\frac{\partial p^{n+1}}{\partial x}\right)_{x=15}=-\frac{1}{\mathrm{Re}}\sum_{q=0}^{J-1}\beta_q\left[\left(\frac{\partial^2 v}{\partial x\partial y}-\frac{\partial^2 u}{\partial y^2}-\frac{\partial^2 u}{\partial z^2}+\frac{\partial^2 w}{\partial z\partial y}\right)_{x=15}\right]^{n-q}$$

$$\left(\frac{\partial p^{n+1}}{\partial y}\right)_{y=0}=-\frac{1}{\mathrm{Re}}\sum_{q=0}^{J-1}\beta_q\left[\left(\frac{\partial^2 u}{\partial x\partial y}+\frac{\partial^2 w}{\partial z\partial y}\right)_{y=0}\right]^{n-q}$$

$$\left(\frac{\partial p^{n+1}}{\partial y}\right)_{y=2}=-\frac{1}{\mathrm{Re}}\sum_{q=0}^{J-1}\beta_q\left[\left(\frac{\partial^2 u}{\partial x\partial y}+\frac{\partial^2 w}{\partial z\partial y}\right)_{y=Ly}\right]^{n-q}$$

$$\left(\frac{\partial p^{n+1}}{\partial z}\right)_{x=0}=\left(\frac{\partial p^{n+1}}{\partial z}\right)_{x=4}=0$$

First, we let the Reynolds number Re = $u_{\max}\,\hbar/\nu$ = 50000 and simulate the stable flow in the channel as the basic flow field. Where $u_{\max}$ is streamwise the maximum velocity, and ν is the kinematic viscosity.

And then initial perturbation is leaded into the channel inlet. The initial perturbation fields is a negative velocity pulse given by $u = -0.5$, $v = w = 0$, $(0.15< x < 0.45,\ 0 < y < 0.02,\ 1.8< z <2.4)$. We directly simulate process that the perturbation evolves into turbulent spots and gradually develops. The significant evolutional experiences are shown in Figs. 1-6.

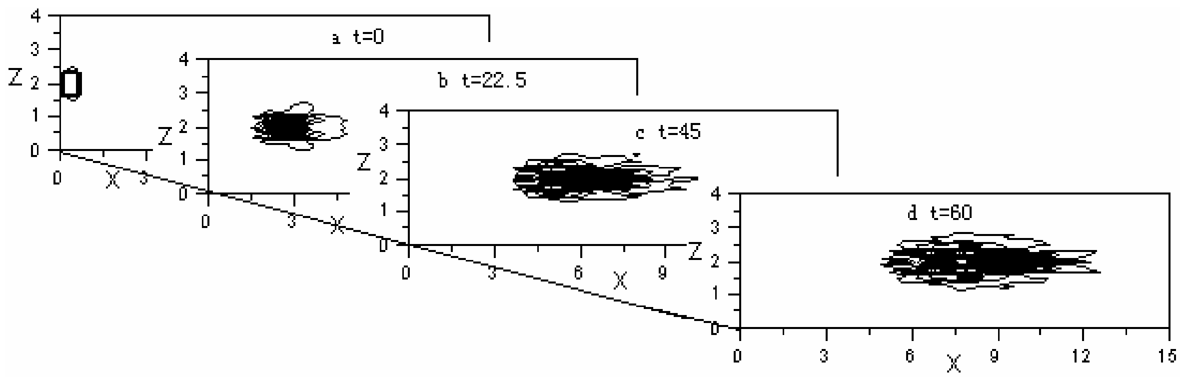

Figure 1: The evolution of normal direction section of turbulent spots

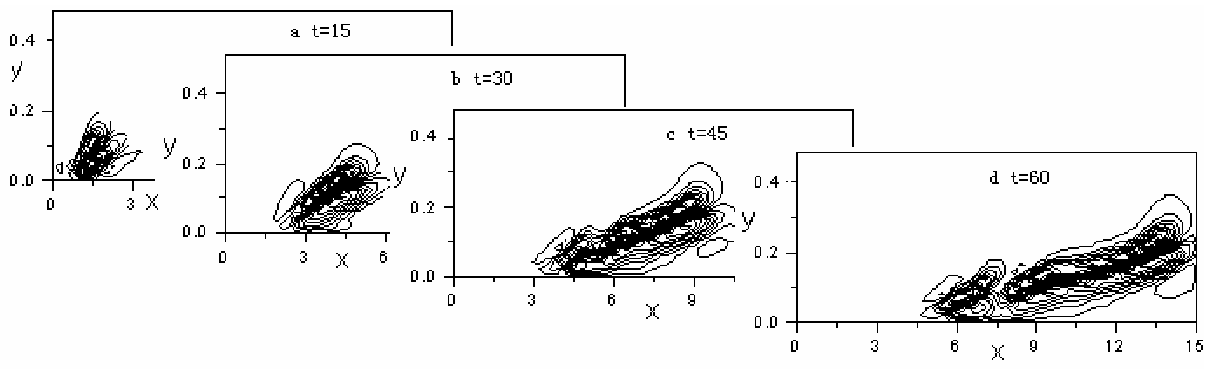

Figure 2: The evolution of stream-wise section of turbulent spots

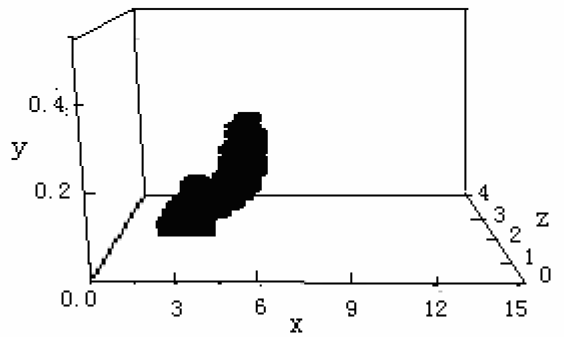

Figure 3: Three-dimensional picture of turbulent spots (t = 35)

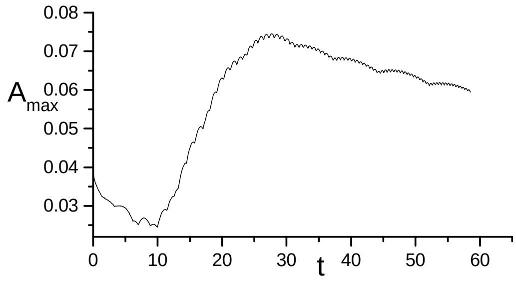

Figure 4: Evolution of amplitude (Re = 50000)

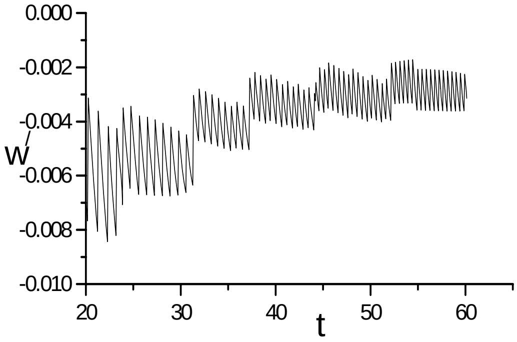

Figure 5: The evolutions of w'

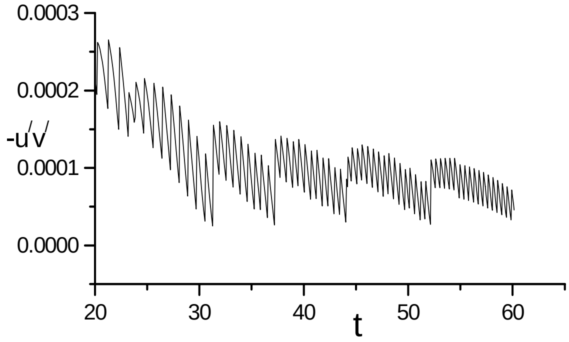

Figure 6: The evolutions of $-u'v'$

Where, $U' = (u', v', w')^T$, is disturbed velocity, $A_{\max} = \max\sqrt{|u'|^2 + |v'|^2 + w'^2}$ is the amplitude of the disturbed intensity.

Fig. 1 and Fig. 2 show the evolutional process of normal direction and stream direction section of the disturbed region, respectively. These indicate turbulent spots continually draw energy of surrounding fluid into this region and make it steadily amplify, arise and form a pipe-like turbulent region oblique to the wall as they travel downstream. Fig. 3 shows three-dimensional picture of turbulent spots at t = 35. These correspond with the experiments on turbulent spots in wall-bounded shear flows [5].

The evolution of the amplitude of the disturbed intensity is given in Figure 4, which shows that the disturbed intensity in disturbed region is obvious high after t > 15. The evolutions of spanwise disturbance velocity (w') and Reynolds stresses ($-u'v'$) in the disturbed region are shown in Fig. 5 and Fig. 6 respectively, which show random pulse in the disturbed region. These indicate that disturbed region has possessed turbulence properties and turbulent spots have been formed.

CONCLUSION

The simulated results show that the three-dimensional coupling fourth-order compact difference schemes developed in the paper has high accuracy, high resolution, and better stability, and can be applied to directly simulate the generation, development and evolution of turbulence in the flow region with complex boundary conditions. It is conducive to deeply researching on turbulent mechanism.

Acknowledgements

The support of the Doctoral Foundation of Ministry of Education of China (Grant No: 20030287003) is gratefully acknowledged.

REFERENCES

1. Moin Mahesh. Direct numerical simulation: a tool in turbulence research. Annu. Rev. Fluid Mech., 1998; **30**: 539-578.

2. Rai Moin. Direct simulation of turbulent flow using finite-difference schemes. J. Comp. Phys., 1991; **96**: 15-53.
3. Lele SK. Compact finite difference schemes with spectral-like resolution. J. Comp. Phys., 1992; **103**: 16-42.
4. Karniakis Israeli, Orszag SA. High-order splitting methods for the incompressible Navier-Stokes equation. J. Comp. Phys., 1991; 97 : 414-443.
5. Wu WY. *Fluid Mechanics* (II). Peking University Press, Beijing, China, 1983, pp.378-379 (in Chinese).