Journal of Southeast University (English Edition) Vol. 22, No. 1, pp. 93 – 95 Mar. 2006 ISSN 1003—7985

Solving Navier-Stokes equation by mixed interpolation method

Wan Shui[1] Mogens Peter Nielsen[2]

([1] College of Transportation, Southeast University, Nanjing 210096, China)

([2] Department of Civil Engineering, Technical University of Denmark, Lyngby, Denmark)

Abstract: The operator splitting method is used to deal with the Navier-Stokes equation, in which the physical process described by the equation is decomposed into two processes: a diffusion process and a convection process; and the finite element equation is established. The velocity field in the element is described by the shape function of the isoparametric element with nine nodes and the pressure field is described by the interpolation function of the four nodes at the vertex of the isoparametric element with nine nodes. The subroutine of the element and the integrated finite element code are generated by the Finite Element Program Generator (FEPG) successfully. The numerical simulation about the incompressible viscous liquid flowing over a cylinder is carried out. The solution agrees with the experimental results very well.

Key words: Navier-Stokes equation; finite element method; incompressible viscous flow; mixed interpolation method

The equations are composed of a continuity equation, a constitutive equation and an equilibrium equation, which are described as follows:

Continuity equation

$$u_{i,i}=0 \tag{1}$$

where u_i is the velocity.

Constitutive equation

$$\sigma_{ji}=-p\delta_{ij}+2\mu d_{ij} \tag{2}$$

where σ_{ji} is the stress tensor, p is the pressure, μ is the viscosity parameter, and $d_{ij}=\frac{1}{2}(u_{i,j}+u_{j,i})$.

Equilibrium equation

$$\sigma_{ji,j}+f_i=(\rho u_i)_{,t}+(\rho u_j u_i)_{,j} \tag{3}$$

where ρ is the density of mass.

Boundary condition

Entrance boundary

$$u_i=u_{i0} \tag{4}$$

Outlet boundary

$$p=p_0 \tag{5}$$

The adhesive boundary on the fastness wall

$$u_i=0 \tag{6}$$

1 Operator Splitting Method

In order to avoid the difficulties in computation, the physical process described by the new theory is decomposed into two processes: a diffusion process and a convection process. They are analyzed separately. The operator splitting method (OSM) is used to deal with the equations. The OSM algorithm is described as follows:

Step 1 Solve the diffusion equations

$$\left.\begin{aligned}(\rho u_i)_{,t}&=f_i+\sigma_{ji,j}\\ u_{i,i}&=0\end{aligned}\right\} \tag{7}$$

Step 2 Solve the convection equation

$$(\rho u_i)_{,t}+(\rho u_j u_i)_{,j}=0 \tag{8}$$

At each time step, the diffusion equation (7) is solved first, and then the result is used in the solution of Eq. (8). The virtual work equation of the diffusion equation (7) is

$$\int_\Omega \rho u_{i,t}\delta u_i \mathrm{d}\Omega-\int_\Omega \sigma_{ji,j}\delta u_i \mathrm{d}\Omega-\int_\Omega u_{i,i}\delta p \mathrm{d}\Omega=\int_\Omega f_i\delta u_i \mathrm{d}\Omega \tag{9}$$

where Ω is the solution domain and δu_i is the virtual displacement of u_i.

Integrating by parts of Eq. (9) and noting the constitutive equation (2), we get

$$\int_\Omega \rho u_{i,t}\delta u_i \mathrm{d}\Omega+\int_\Omega 2\mu d_{ij}\delta u_{i,j}\mathrm{d}\Omega-\int_\Omega p\delta u_{i,i}\mathrm{d}\Omega-\int_\Gamma \sigma_{ji}n_j\delta u_i \mathrm{d}\Gamma-\int_\Omega u_{i,i}\delta p \mathrm{d}\Omega=\int_\Omega f_i\delta u_i \mathrm{d}\Omega \tag{10}$$

where Γ is the boundary of Ω, and n_j is the direction cosine of the outside normal vector on Γ.

For the adhesive boundary condition Eq. (6), δu_i equals zero, so Eq. (10) can be written as

$$\int_\Omega \rho u_{i,t}\,\delta u_i \mathrm{d}\Omega+\int_\Omega 2\mu d_{ij}\delta d_{ij}\mathrm{d}\Omega-\int_\Omega p\delta u_{i,i}\mathrm{d}\Omega-\int_\Omega u_{i,i}\delta p \mathrm{d}\Omega=\int_\Omega f_i\delta u_i \mathrm{d}\Omega \tag{11}$$

According to the standard FE assembling process, the FE equations of Eq. (11) can be written as

Received 2005-06-15.

Biography: Wan Shui (1960—), male, doctor, professor, wanshui60421@yahoo.com.

$$(\boldsymbol{M}+\boldsymbol{S}\Delta t)\boldsymbol{u}^{n+1}=\boldsymbol{M}\boldsymbol{u}^{n}+\boldsymbol{F}\Delta t \tag{12}$$

where $\boldsymbol{M}$, $\boldsymbol{S}$ are the mass matrix and the stiffness matrix, respectively; Δt is the time increment.

The convection equation (8) can be written as the following time-discrete equation

$$\rho u_i^{n+1}+\Delta t\rho(u_j^n u_i^{n+1})_{,j}=\rho u_i^n \tag{13}$$

The least square scheme is used to solve Eq. (13). The weak form is

$$\int_\Omega L(u_i)\delta L(u_i)\mathrm{d}\Omega=\int_\Omega \rho u_i^n\,\delta L(u_i)\mathrm{d}\Omega \tag{14}$$

where $L(u_i)=\rho u_i^{n+1}+\Delta t\rho(u_j^n u_i^{n+1})_{,j}$ and $\delta L(u_i)$ is the virtual displacement of $L(u_i)$.

2 Mixed Interpolation Method

The FE code is made according to Eqs. (12) and (14) based on the FEPG[1-5]. If the element with four nodes or eight nodes is used to solve Eqs. (12) and (14), the calculation will be unstable as the Reynolds number becomes greater. In order to make the computation stable and to get the convergence solution, the term $c\times|d|\times\int_\Omega p_{,i}\delta p_{,i}\mathrm{d}\Omega$ is added into Eq. (11), where c is a constant number, $|d|$ is the value of the Jacobian determinant of the element.

$$\int_\Omega \rho u_{i,i}\delta u_i\mathrm{d}\Omega+\int_\Omega 2\mu d_{ij}\delta d_{ij}\mathrm{d}\Omega-\int_\Omega p\delta u_{i,i}\mathrm{d}\Omega-\int_\Omega u_{i,i}\delta p\mathrm{d}\Omega-c\times|d|\times\int_\Omega p_{,i}\delta p_{,i}\mathrm{d}\Omega=\int_\Omega f_i\delta u_i\mathrm{d}\Omega \tag{15}$$

In engineering, we want to get the accurate pressure value that the fluid exerts on the structure. Although the term $c\times|d|\times\int_\Omega p_{,i}\delta p_{,i}\mathrm{d}\Omega$ can make the solution stable, it results in some errors in the computation. The greater the Reynolds number becomes, the greater the c will be taken and the more seriously the pressure calculated will deviate from its real value. In order to get the accurate pressure value and keep the stability, we use the mixed interpolation method to solve Eqs. (12) and (14). The isoparametric element with nine nodes is used to describe the velocity field. Fig. 1 shows the element.

The shape function of the element is

$$\left.\begin{aligned}
&N_i=\frac{1}{4}(\xi^2+\xi\xi_i)(\eta^2+\eta\eta_i) && i=1,3,5,7\\
&N_i=\frac{1}{2}[\eta_i^2(\eta^2-\eta\eta_i)(1-\xi^2)+\\
&\qquad\xi_i^2(\xi^2-\xi\xi_i)(1-\eta^2)] && i=2,4,6,8\\
&N_i=(1-\xi^2)(1-\eta^2) && i=9
\end{aligned}\right\} \tag{16}$$

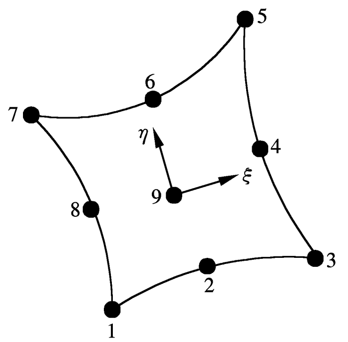

Fig. 1 Isoparametric element with nine nodes

Only four nodes at the vertex in the same element is used for the interpolation function to describe the pressure field. It is

$$N_i=\frac{1}{4}(1+\xi\xi_i)(1+\eta\eta_i)\qquad i=1,3,5,7 \tag{17}$$

In this way we need not add the additional term $c\times|d|\times\int_\Omega p_{,i}\delta p_{,i}\mathrm{d}\Omega$ and can obtain a stable solution.

3 Numerical Example

A numerical simulation of the incompressible viscous flow over a cylinder is made in this example. The Reynolds number is 1 000. The multi-frontal solver is used to solve the FE equation. The computational domain is a rectangle. It is 3 m long and 2 m wide (see Fig. 2). The diameter of the cylinder D is 0. 1 m. The grid is generated by software Gid. There are 5 528 elements, 22 292 nodes in the computational domain (see Fig. 3).

The physical parameters of the liquid are as follows: the viscosity parameter $\mu=10^{-3}$ kg/(m·s), the

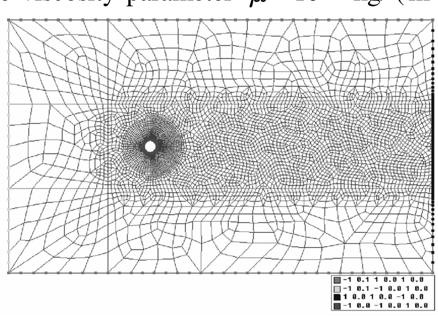
Fig. 2 Grid of the computational domain

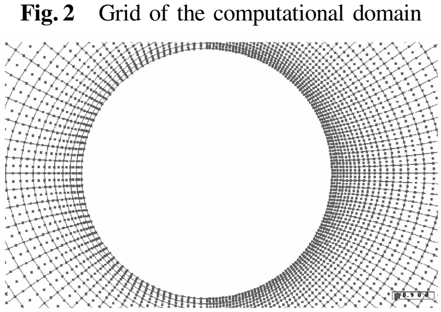
Fig. 3 Grid and element

density of the liquid $\rho = 1\,000$ kg/m^3. The outlet condition $p_0 = 0$ Pa. The velocity on the boundary of the cylinder is $u = v = 0$ m/s. The entrance flow velocity is $v_0 = 0$ m/s and $u_0 = \dfrac{\mu Re}{D\rho}$ (Re is the Reynolds number).

Solutions are obtained from the numerical simulation. Fig. 4 shows the pressure field made by the incompressible viscous flow over a cylinder. Fig. 5 shows two curves which change against the Reynolds number. One is about the parameter of drag force C_D and the other is about the parameter of the lift force C_L. From Fig. 5 we know that the average value of C_D equals 1.19. It agrees with the experimental results very well[6].

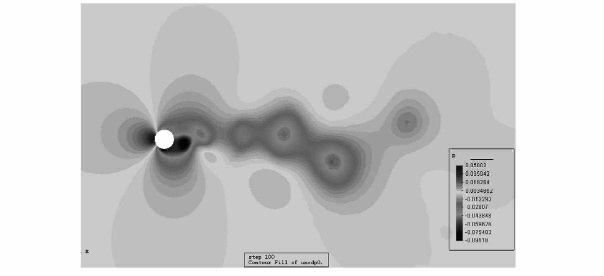

Fig. 4 Pressure field

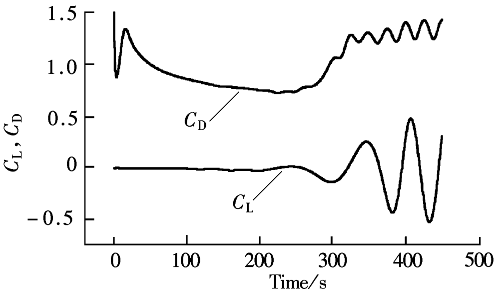

Fig. 5 Curves of C_D and C_L ($Re = 1\,000$)

Acknowledgements Thanks sincerely to professor Liang Guoping for his directions, Dr. Qian Huashan, Jia Feng and FEGENSOFT Co. Ltd. for their help and valuable material.

References

[1] Zienkiewicz O C, Taylor R L. *The finite element method*, vol. 3[M]. 5th ed. Butterworth Heinemann, 2000.

[2] Smith I M, Girffiths D V. *Programming the finite element method*[M]. 3rd ed . John Wiley & Sons, Inc. 1998.

[3] Taylor C, Hughes T G. Finite element programming of the Navier-Stokes equation [M]. Swansea: Pineridge Press Ltd, 1981.

[4] FEPG 5.0 user's guide [R]. FEPGENSOFT Co. Ltd., 2004. (in Chinese)

[5] FEPG 5.0 theory menu [R]. FEPGENSOFT Co. Ltd., 2004. (in Chinese)

[6] Zhao Xiaobao. Fluid mechanics with engineering applications [M]. Nanjing: Southeast University Press, 2004. (in Chinese)

混合插值法求解 Navier-Stokes 方程

万 水[1] Mogens Peter Nielsen[2]

([1] 东南大学交通学院,南京 210096)

([2] 丹麦技术大学土木工程系,丹麦林比)

摘要: 利用算子分裂法将粘性不可压流体 Navier-Stokes 方程表述的物理过程分解为扩散和对流 2 个过程,建立起它们的有限元方程. 采用 9 节点四边形等参元中的形函数描述单元中流体的速度场,而利用这种单元的 4 个顶点节点构成的插值函数描述单元中流体的压力场,通过有限元生成系统(FEPG)进行 Navier-Stokes 方程的单元子程序的生成,得到求解不可压粘性流体 Navier-Stokes 方程的有限元程序,并用它对粘性不可压流体的挠圆柱流动问题进行了数值模拟,分析结果与试验结果吻合.

关键词: Navier-Stokes 方程;有限元法;粘性不可压流;混合插值法

中图分类号: TU311.3