Vol 15 No 10, October 2006
1009-1963/2006/15(10)/2352-05

Chinese Physics

© 2006 Chin. Phys. Soc.
and IOP Publishing Ltd

An analysis of the incompressible viscous flows problem by meshless method*

Xiong Yuan-Bo(熊渊博) and Wang Hao(王 浩)†

Department of Engineering Mechanics, Hunan University, Changsha 410082, *China*

(Received 14 November 2005; revised manuscript received 21 March 2006)

Generally the incompressible viscous flow problem is described by the Navier–Stokes equation. Based on the weighted residual method the discrete formulation of element-free Galerkin is inferred in this paper. By the step-by-step computation in the field of time, and adopting the least-square estimation of the-same-order shift, this paper has calculated both velocity and pressure from the decoupling independent equations. Each time fraction Newton–Raphson iterative method is applied for the velocity and pressure. Finally, this paper puts the method into practice of the shear-drive cavity flow, verifying the validity, high accuracy and stability.

Keywords: incompressible viscous flows, element-free Galerkin method, Navier–Stokes equation, weighted residual method

PACC: 4715F

1. Introduction

The incompressible viscous flows problem is of high importance in the aerospace, ocean freight, petrochemical industry, energy industry, etc. Important and effective methods of analysis are numerical methods, such as the finite difference method, finite element method, finite volume method. The MM (meshless method) has emerged for solving large deformation, moving boundary and all other complex problems lately. Atluri *et al* have used the meshless local Petrov–Galerkin method to solve the problems of steady state incompressible viscous flows recently,[1] which indicated that MM is promising in the application of analysing complicated hydromechanics problems.

The element-free Galerkin method (EFGM)[2] is a meshless method of good compatibility and stability. We use it in incompressible Navier–Stokes equations and for solving undisturbed flow problems.[3] In this paper the EFGM is extended to address the problems of undisturbed incompressible viscous flows. Because of the fraction method in the discretization of space, variable velocity and pressure can be obtained by the same-order moving least-square (MLS) interpolation approximation, leading to the discrete Navier–Stokes equations based on the Galerkin weighted residual method. Finally, the method is applied to calculate the instance of the shear-drive cavity inner flow, to verify the validity and accuracy of this study.

2. Basic equation and step-by-step form

The mathematical description of the general incompressible viscous flow problem is the Navier–Stokes equations of motion and the equation of continuity,

$$\frac{\partial u_i}{\partial t} + \frac{\partial u_i}{\partial x_j}u_j = f_i - \frac{1}{\rho}\frac{\partial p}{\partial x_i} + \frac{\mu}{\rho}\frac{\partial^2 u_i}{\partial x_j^2}, \qquad (1)$$

$$\frac{\partial u_i}{\partial x_i} = 0, \qquad (2)$$

where i, j=1, 2, 3, u_i, ρ and p represent material velocity of fluid particle, fluid density and pressure respectively; μ is fluid viscosity parameter; f_i is body force.When reference frame is non-inertia, inertial-force item should be merged into body-force item, it is still denoted by f_i.

In order to make sure the above equation's solution, corresponding threshold condition and initial condition must be given. Consider the threshold conditions of 2 basic types on the curve Γ which encloses

*Projects Supported by the Project Fund of the Hubei Province Key Lab. of Mechanical Transmission & Manufacturing Engineering Wuhan University of Science & Technology (Grant No 2003A16) is gratefully acknowledged.

†E-mail: catboywh@163.com

the computation domain Ω. The Dirichlet boundary condition:

$$u_i = \bar{u}_i \quad \text{on} \quad \Gamma_u. \tag{3}$$

The Neumann boundary condition:

$$p = \bar{p} \quad \text{on} \quad \Gamma_t, \tag{4}$$

where $\bar{u}_i$ and $\bar{p}$ are fixed value of velocity and pressure, $\Gamma = \Gamma_u \cup \Gamma_t$ and $\Gamma_u \cap \Gamma_t = \Phi$.

Through the equations mentioned above, the incompressible viscous flow problem can be worked out in principle. But the nonlinear convection terms in the equations are hard to process mathematically. In Eqs.(1) and (2) velocity of flow and pressure are coupled. The unknown pressure p is not included in the continuity equation (2) which depicts the pressure field. So velocity u_i and pressure p cannot be attained without certain discretization techniques and a number-seeking method.

In order to simplify the computation form, reduce the workload of calculation and improve solving efficiency in the analysis of MM, we use the step-by-step method of time domain as stated in Ref.[4] to discretize the equation of description of incompressible viscous flows. Equation (1) and Eq. (2) in the form of time discretization are as follows:

$$u_i^{n+1} = u_i^n - \Delta t\left[u_j^n \frac{\partial u_i^n}{\partial x_j} + \frac{1}{\rho}\frac{\partial p^{n+1}}{\partial x_i} - \frac{\mu}{\rho}\frac{\partial^2 u_i^n}{\partial x_j^2} - f_i^{n+1}\right], \tag{5}$$

$$\frac{\partial u_i^{n+1}}{\partial x_i} = 0. \tag{6}$$

Discarding the pressure p's increment term in Eq.(5), and inducting the intermediate velocity, we get

$$\tilde{u}_i^{n+1} = u_i^n - \Delta t\left[u_j^n \frac{\partial u_i^n}{\partial x_j} + \frac{1}{\rho}\frac{\partial p^n}{\partial x_i} - \frac{\mu}{\rho}\frac{\partial^2 u_i^n}{\partial x_j^2} - f_i^{n+1}\right]. \tag{7}$$

Making use of the continuity equation further to deduce the velocity and pressure formulae we have

$$u_i^{n+1} = \tilde{u}_i^{n+1} - \frac{\Delta t}{\rho}\left(\frac{\partial p^{n+1}}{\partial x_i} - \frac{\partial p^n}{\partial x_i}\right), \tag{8}$$

$$\frac{\partial^2 p^{n+1}}{\partial x_i^2} = \frac{\rho}{\Delta t}\frac{\partial \tilde{u}_i^{n+1}}{\partial x_i} + \frac{\partial^2 p^n}{\partial x_i^2}. \tag{9}$$

As the matter stands, the incompressible viscous flow equation is transformed into fractional form in which velocity and pressure are expressed independently, namely, an explicit form of velocity as Eq.(8) and implicit form of pressure as Eq.(9). The computation procedures of corresponding step-by-step method are: calculate the intermediate velocity $\tilde{u}_i^{n+1}$ by Eq.(7); then work out pressure p^{n+1} by Eq.(9); next calculate the velocity of time $n+1$ step by Eq.(8); finally use u_i^{n+1} replace u_i^n in Eq.(7) and enter the loop of next time step.

3. Equation for discretization and number implementation

With the EFGM, space discretization is directly applied to the equation of incompressible viscous flow fractional-step form obtained earlier. Velocity and pressure are solved by the decoupling co-independent equations, the computation carried out in step-by-step form in time domain. The same-order moving interpolation can be taken. Velocity and pressure shape functions are generated by MLS approximation function[2,3]

$$u_i(\mathbf{x}) = \sum_{\alpha=1}^{n} \varphi_\alpha(\mathbf{x})\hat{u}_{\alpha i}, \tag{10}$$

$$p(\mathbf{x}) = \sum_{\alpha=1}^{n} \varphi_\alpha(\mathbf{x})\hat{p}_\alpha \tag{11}$$

where n is the number of nodes in the influence domain Ω_s of point x; $\hat{u}_{\alpha i}$ and $\hat{p}_\alpha$ denote the value of the ith component of nominal velocity of the αth nodes x_α in the sub-domain and nominal pressure; φ_α is the mobile MLS function in interpolated form similar to the finite element method (FEM) function in the same form, which constitutes the selected m-dimension complete monomial radix $g^{\mathrm{T}}(x)$ and weight function $w_i(x)(>0)$ of MLS approximate function at the point x_i. See Ref.[5] for more details. Analysing two-dimensional flow problems we take the cubic radix function

$$g^{\mathrm{T}}(x) = [1\ x_1\ x_2\ x_1^2\ x_1\ x_2\ x_2^2\ x_1^3\ x_1^2\ x_2\ x_1\ x_2^2\ x_2^3],$$
$$m = 10 \tag{12}$$

four spline weight function

$$w_\alpha(x)=\begin{cases}1-6\left(\dfrac{d_\alpha}{r_\alpha}\right)^2+8\left(\dfrac{d_\alpha}{r_\alpha}\right)^3-3\left(\dfrac{d_\alpha}{r_\alpha}\right)^4, & 0\le d_\alpha\le r_\alpha;\\ 0, & d_\alpha\ge r_\alpha,\end{cases} \tag{13}$$

where $d_\alpha=\|x-x_\alpha\|$, r_α is the radius of the support domain of point x_α.

Concluded from the incompressible viscous flows step-by-step form equations (7), (8), (9) obtained by the Galerkin weighted residual method and the threshold conditions introduced by a penalty factor and MLS interpolation, functions (10), (11) are enforced by this method, and the number-discrete equations of the EFGM are as follows:

$$\begin{aligned}M^{n+1}_{\alpha\beta}\tilde{u}^{n+1}_{\beta i}=&M^n_{\alpha\beta}u^n_{\beta i}-\Delta t\Big[B^n_{\alpha\beta}u^n_{\beta i}+\frac{1}{\rho}C^n_{\alpha\beta i}p^n_\beta\\&+D^n_{\alpha i\beta j}u^n_{\beta j}-F^{n+1}_{\alpha i}-\hat{E}^{n+1}_{\alpha i}\Big],\end{aligned} \tag{14}$$

$$\begin{aligned}&(M^{n+1}_{\alpha\beta}-\lambda_1H^{n+1}_{\alpha\beta})u^{n+1}_{\beta i}\\=&M^{n+1}_{\alpha\beta}\tilde{u}^{n+1}_{\beta i}-\frac{\Delta t}{\rho}\left[C^{n+1}_{\alpha\beta i}p^{n+1}_\beta-C^n_{\alpha\beta i}p^n_\beta\right]\\&-\lambda_1\hat{U}^{n+1}_\alpha,\end{aligned} \tag{15}$$

$$\begin{aligned}&(A^{n+1}_{\alpha\beta}+\lambda_2R^{n+1}_{\alpha\beta})p^{n+1}\\=&-\frac{\rho}{\Delta t}C^{n+1}_{\alpha\beta i}\tilde{u}^{n+1}_{\beta i}+A^n_{\alpha\beta}p^n+\hat{Q}^{n+1}_\alpha\\&-\hat{Q}^n_\alpha+\lambda_2\hat{S}^{n+1}_\alpha.\end{aligned} \tag{16}$$

In the above equations

$$\begin{aligned}M^s_{\alpha\beta}&=\int_\Omega\varphi_\alpha\varphi_\beta\mathrm{d}\Omega,\\B^s_{\alpha\beta}&=\int_\Omega\varphi_\alpha\varphi_{\beta,j}u^n_j\mathrm{d}\Omega,\\C^s_{\alpha\beta i}&=\int_\Omega\varphi_\alpha\varphi_{\beta,i}\mathrm{d}\Omega,\\D^s_{\alpha i\beta j}&=\int_\Omega\varphi_{\alpha,j}\mu\varphi_{\beta,j}\mathrm{d}\Omega,\\F^s_{\alpha i}&=\int_\Omega\varphi_\alpha f_i\mathrm{d}\Omega,\\\hat{E}^s_{\alpha i}&=\frac{1}{\rho}\int_\Gamma\varphi_\alpha\mu\varphi_{\beta,j}n_j\mathrm{d}\Gamma,\\A^s_{\alpha\beta}&=\int_\Omega\varphi_{\alpha,i}\varphi_{\beta,i}\mathrm{d}\Omega,\\\hat{Q}^s_\alpha&=\int_\Gamma\varphi_\alpha\varphi_{\beta,i}n_i\mathrm{d}\Gamma,\\R^s_{\alpha\beta}&=\int_{\Gamma_t}\varphi_\alpha\varphi_\beta\mathrm{d}\Gamma,\\\hat{U}^s_\alpha&=\int_{\Gamma_u}\varphi_\alpha\bar{u}\mathrm{d}\Gamma,\\\hat{S}^s_\alpha&=\int_{\Gamma_t}\varphi_\alpha\bar{p}\mathrm{d}\Gamma,\\H^s_{\alpha\beta}&=\int_{\Gamma_u}\varphi_\alpha\varphi_\beta\mathrm{d}\Gamma,\end{aligned}$$

where α, β are the numbers of nodes in the subdomain, i, j denote the dimension of pending problem (two-dimensional problem i,j=1, 2), λ_1, λ_2 are two penalty factors, and n_j is the outward normal unit vector on the boundary. So the incompressible viscous flow problem's EFGM number implementation is changed into the solution of the above three algebraic equations respectively, its detailed processes of realization and computation are just as stated in Refs.[3, 5].

In order to avoid the possible shift or instability caused by the various integrals mentioned above, Eqs.(15), (16) should be iteratively solved with Newton–Raphson increment. The method is the same as in Ref.[3] and this paper will not be tautological.

4. Numerical examples

The shear-drive two-dimensional cavity flow problem is a well-known benchmark for a large number of verifying arithmetics and analytical models, as Refs.[6, 7]; there is a relatively complicated move of flow field but a simple boundary. We analyse this flow to show the validity and reliability by using EFGM to solve incompressible viscous flow problems.

This is a typical laminar flow calculation problem. The three-dimensional fixed boundary is defined as: $u_x=u_y=0$. Horizontal flow velocity of cavity cover is invariable: that is to say, suddenly increasing to constant U from initial value zero. We use the EFGM to calculate the result when the flow is invariable, and define the Reynolds parameter as $R_\mathrm{e}=\rho U\cdot H/\mu$, computed as R_e=1000 and R_e=5000.

At first 361 (19×19) evenly-deposited-regular-node scheme is adopted, with background grid made up of 400 (20×20) regular tetragon cells. Select (3×3) Gauss integral nodes in each cell and the radius γ_α of support domain takes the value of $0.25H$, together

with time step increment $\Delta t = 0.01$s. Setting the number of iterations as 40, the results of the flow field of the whole solution space are shown in Fig.2. An obvious clockwise host vortex is generated by flow field near the middle of the cavity, as stated in this figure, for the viscosity of fluid and the movement of cover board. And each of the three anticlockwise rotation quadric vortexes lies at the left top corner, left root corner, right root corner of cavity respectively (Fig.1). This result is the same as the one from FEM of PJM1 in Ref.[7].

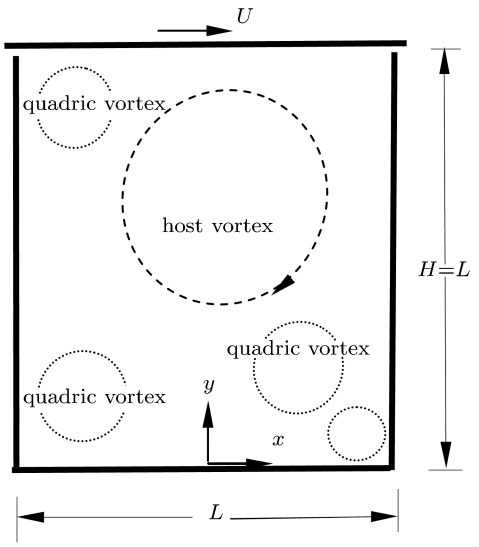


Fig.1. Shear drive cavity problem and boundary condition.

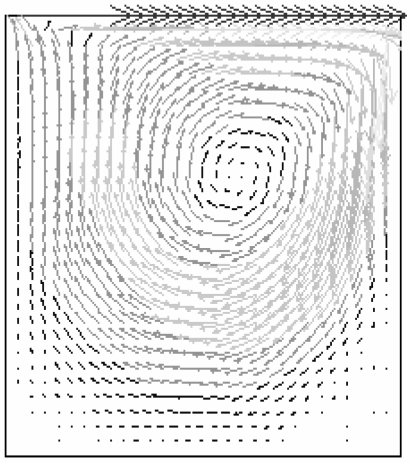


Fig.2. Velocity vector field in shear drive cavity.

Figures 3 and 4 show the distribution charts of horizontal and vertical velocities at the centreline of cavity u_x/U, u_y/U. From these figures, the velocity grids are much bigger in the small regions near the left, right and bottom immovable walls than in other regions. At the same time, Table 1 shows the maximum and minimum flow velocities and corresponding places calculated in this paper when R_e=5000, including the results of other researchers for comparison. Among all, it is considered that the result of Ghia *et al*[6] is most precise. The results obtained by the present method are of high accuracy and in good agreement with others, compared with those given in the Table 1.

In addition, in order to prove the validity of the method and the precision of the algorithm further, we calculate with the schemes of irregular meshes of 361 nodes and sparse regular meshes of 225 (15×15) nodes respectively, still with the background grid made up of 400 (20×20) regular tetragon cells. The maximum and minimum velocity results are both included in Table 1. It can be concluded that the irregular layout of nodes have little to do with the results; the calculation accuracy and efficiency of the EFGM bears a reference to the numbers of nodes (sparse or dense) in computation regions. And high accuracy can also be obtained even with quite sparse meshes of nodes.

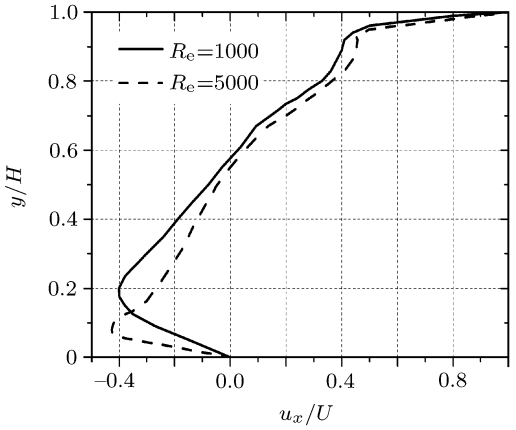


Fig.3. Velocity u_x/U distribution of level centreline.

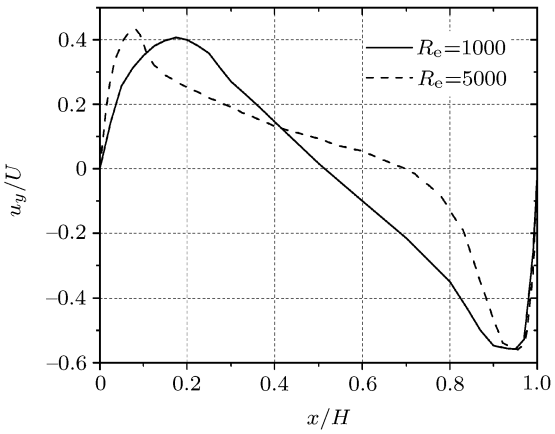


Fig.4. Velocity u_y/U distribution of cross centreline.

Table 1. Max and min velocity in shear drive cavity clamped.

Max and min velocity		$(u_x/U)_{\min}$	$(x/H=0)$	$(u_y/U)_{\max}$	$(y/H=0)$	$(u_y/U)_{\min}$	$(y/H=0)$
		$(u_x/U)_{\min}$	y/H	$(u_y/U)_{\max}$	x/H	$(u_y/U)_{\min}$	x/H
Present method	(Regular nodes 361)	–0.428	0.0773	0.431	0.0826	–0.561	0.956
	(Irregular nodes 361)	–0.426	0.0774	0.432	0.0829	–0.560	0.957
	(Regular nodes 225)	–0.343	0.0692	0.347	0.0765	–0.516	0.940
Ref.[6]		–0.436	0.070	0.436	0.078	–0.554	0.953
Ref.[7]		–0.348	0.083	0.355	0.095	–0.533	0.949
Ref.[8]		–0.377	0.074	0.357	0.074	–0.504	0.953

5. Conclusions

The basic concept and implementation of the EFGM approach for analysing and solving the incompressible viscous flow problem have been presented in the paper. The step-by-step method from FEM in the field of time was adopted for discretizing the incompressible Navier–Stokes equation. So velocity and pressure can use the-same-order-shift least-square (MLS) function interpolations. The Newton–Raphson iterative method is applied in each time fraction to ensure accuracy. Numerical calculations for the computation example of shear flow in the drive cavity show that the method presented in this paper is efficient and reliable, besides offering an excellent calculation precision and an accurate exposition of the flow distribution of flow field.

The EFGM is a relatively sophisticated numerical method in MM. It has much in common with FEM in seeking solution. This approach only needs the information of nodes instead of cells and the simple pre-processing. What is more, it can also construct random high order continuous approximate field functions without the compatibility issues in FEM. This approach promises a bright future in computing complex fluid dynamics problem such as great sloshings, interaction between fluid and structure, etc.

References

[1] Lin H and Atluri S N 2001 *Computer Modeling in Engineering & Sciences* (CMES) **2** 117

[2] Ted Belytschko, Lu Y Y and Gu L 1994 *Int. J. Numer. Meth. Eng.* **37** 229

[3] Xiong Y B, Long S Y and Hu D A 2004 *J. Hunan University* (Natural Sciences) **31** 93 (in Chinese)

[4] Hayashi M, Hatanaka K and Kawahara M 1991 *Int. J. Numer. Meth. Fluids* **13** 805

[5] Pang Z H, Ge X R, Zheng H and Wang S L 1999 *Chin. J. Comput. Mech.* **16** 320 (in Chinese)

[6] Ghia U, Chia K N and Shin C T 1982 *J. Comput. Phys.* **48** 387

[7] Chen D H and Li W 2001 *Journal of Hydrodynamics* **16** 442 (in Chinese)

[8] Gresho P M and Chan S T 1990 *Int. J. Numer. Meth. Fluids* **11** 621