INT. J. NUM. METHODS HEAT FLUID FLOW, VOL. 1, 77–94 (1991)

A FRACTIONAL STEP FINITE ELEMENT METHOD FOR CONDUCTIVE–CONVECTIVE HEAT TRANSFER PROBLEMS

KATSUMORI HATANAKA AND MUTSUTO KAWAHARA

Department of Civil Engineering, Chuo University, Kasuga 1-13-27, Bunkyo-ku, Tokyo 112, Japan

ABSTRACT

A new fractional step method in conjunction with the finite element method is proposed for the analysis of the thermal convection and conduction in a fluid region expressed by the momentum equations, the equation of continuity and the energy equation. This paper focuses on the features of the present finite element method which gives a simple way of treating the Neumann boundary condition for the pressure Poisson equation. The applicability and effectiveness of the proposed scheme are illustrated through the numerical examples of the two-dimensional natural convection flow in enclosures with several Rayleigh numbers.

KEY WORDS Finite element method Fractional step method Natural convection flow Laminar incompressible flow Navier–Stokes equations Pressure Poisson equation

INTRODUCTION

The numerical solution of the problems involving the flow of laminar natural convection in enclosures is an important and an interesting area not only in the computational fluid dynamics but also in many other engineering practices. There are many important engineering applications of the natural convection in enclosures, for instance, free convection in the cases of a passive solar room heated and cooled on two opposing vertical walls, heat transfer through double-glazed window, general circulation planetary atmospheres and so on. The behaviour of the natural convection flow has, however, received limited attention because of its three-dimensionality and the difficulty in the numerical simulation, e.g. the combination of the time-scale-difference of the flow fields and temperature fields.

So far, the numerical solutions for this type of problem have been carried out successfully by the various techniques based on the finite difference method and have been done, principally, in rectangular or cubic enclosures[1–4]. However, there are still remaining some difficulties in their applications to the practical problems. The most notable one is the inconvenience of making FDM grid with accurately describing complex boundary shapes which often appear in the practical problems. Because of this, the use of the finite element method has been of interest in the computational simulations of the natural convection problems in recent years. Since the finite element method involves a remarkable feature in the treatment of the natural-boundary-condition and provides a convenience of making much complex mesh configuration to the practical problems, it is apparent that the finite element method provides a workable approach to the solution of non-linear coupled physical phenomena.

In this paper, the fractional step finite element method is applied to the thermal viscous fluid flow analysis. In recent years, several finite element analyses have already been presented based

0961–5539/91/010077–18$2.00

Received October 1990
Revised February 1991

on the fractional step method. The difficulty of the solution method for the problems of non-linear thermal flow arises from the continuity equation, i.e. the continuity equation does not include the pressure term in case of the incompressible flow. To overcome the difficulty, the fractional step method has been originated to obtain the pressure equation, which can solve the pressure field using the velocities computed by the previous time step in a certain implicit manner. The fractional step method consists of a two-step formulation. In the first step, the velocity fields are solved based on the velocities computed at the previous time point. Using the computed velocities, the pressure field can be computed.

The formulation of the fractional step method previously published can be divided into two groups. One formulation employs that the fractional step formulation is applied after discretizing the governing equations. In this formulation, Donea[5,6], Gresho[7–9] and others are included. Contrary to this, the other formulation is that the governing equations are transformed into the fractional step formulation then discretization procedure is applied. Kawahara and his group[10–13], Mizukami and Tsuchiya[14] and others use this formulation. The most advantageous point of the second fractional step formulation is that the same interpolation function can be used for pressure and velocity fields. This fact considerably reduces the computational efforts. Thus, this paper employed the second fractional step method. However, it is important to note that the transformed equations can be coincident with the basic governing equations only if the proper boundary condition can be applied.

The major emphasis of the present fractional step finite element method is the treatment of the Neumann boundary conditions for the pressure field. In the conventional fractional step method, the pressure gradients on boundaries $\partial p/\partial n_i$ must be given exactly to solve the pressure Poisson equation. However, it is not straightforward to estimate the exact $\partial p/\partial n_i$ on boundaries and because of this, it is difficult to give the exact Neumann boundary condition for the pressure field. The present fractional step finite element method provides a simple way of estimating $\partial p/\partial n_i$ by making use of the momentum equations. The Neumann condition to the pressure Poisson equation is transformed into the time-derivative term of velocity fields by the present method from which the computation of the pressure Poisson equation becomes easier than before. In the point of view of the situations cited in References 15–17, a new fractional step finite element method has been presented for solving laminar incompressible viscous fluid flow problems and many numerical solutions have been carried out by this method successfully and sufficiently[15–17]. In this paper, the authors present the application of the present finite element method to the numerical solution of non-linear coupled physical phenomena, such as the laminar natural convection due to the temperature-induced buoyancy in a finite two-dimensional enclosures i.e., cavity. The adaptability and applicability of the present scheme are obtained through several numerical examples. The computed results agree well with the physical phenomena.

GOVERNING EQUATIONS

Let Ω be a fluid flow domain which is surrounded by a piecewise smooth boundary Γ. The continuity, momentum and energy equations in a two-dimensional form, governing a laminar unsteady flow of a Newtonian fluid with the gravity acceleration acting in the vertical direction can be written as:

$$\frac{\partial \rho}{\partial t^*}+\frac{\partial}{\partial x^*}(\rho u^*)+\frac{\partial}{\partial y^*}(\rho v^*)=0 \quad \text{in } \Omega \tag{1}$$

$$\frac{\partial}{\partial t^*}(\rho u^*)+\frac{\partial}{\partial x^*}(\rho u^{*2})+\frac{\partial}{\partial y^*}(\rho u^* v^*)=-\frac{\partial p^*}{\partial x^*}+\frac{\partial}{\partial x^*}\left(\mu\frac{\partial u^*}{\partial x^*}\right)+\frac{\partial}{\partial y^*}\left(\mu\frac{\partial u^*}{\partial y^*}\right) \quad \text{in } \Omega \tag{2}$$

$$\frac{\partial}{\partial t^*}(\rho v^*)+\frac{\partial}{\partial x^*}(\rho u^* v^*)+\frac{\partial}{\partial y^*}(\rho v^{*2})=-\frac{\partial p^*}{\partial y^*}-\rho g+\frac{\partial}{\partial x^*}\left(\mu\frac{\partial v^*}{\partial x}\right)+\frac{\partial}{\partial y^*}\left(\mu\frac{\partial v^*}{\partial y}\right) \quad \text{in } \Omega \tag{3}$$

$$\frac{\partial}{\partial t^*}(\rho c_p T^*) + \frac{\partial}{\partial x^*}(\rho u^* c_p T^*) + \frac{\partial}{\partial y^*}(\rho v^* c_p T^*) = \frac{\partial}{\partial x^*}\left(k\frac{\partial T^*}{\partial x^*}\right) + \frac{\partial}{\partial y^*}\left(k\frac{\partial T^*}{\partial y^*}\right) \quad \text{in } \Omega \tag{4}$$

where t is time, x, y are the Cartesian coordinates, ρ is the density, u, v are velocities, p is pressure, μ is the dynamic viscosity, g is the gravity acceleration, c_p is the principal specific heat at constant pressure, T is temperature, k is the thermal conductivity of fluid and * denotes the dimensional variables.

In this paper, the fluid motion is further assumed to be laminar and the fluids under consideration are assumed to be incompressible. The dimensionless form of the governing equations in transient form with the Boussinesq approximation can be rewritten in the following forms. Here and henceforth, the equations are described using indicial notation and the summation convention for the repeated indices.

$$u_{i,i} = 0 \quad \text{in } \Omega \tag{5}$$

$$\frac{\partial u_i}{\partial t} + u_j u_{i,j} - \sigma_{ij,j} = PrRaTf_i \quad \text{in } \Omega \tag{6}$$

$$\frac{\partial T}{\partial t} + u_j T_{,j} - T_{,jj} = 0 \quad \text{in } \Omega \tag{7}$$

$$\sigma_{ij} = -p\delta_{ij} + d_{ij} \tag{8}$$

$$d_{ij} = 2Pre_{ij} \tag{9}$$

$$e_{ij} = \tfrac{1}{2}(u_{i,j} + u_{j,i}) \tag{10}$$

where

$$Pr = \frac{\nu}{\alpha}$$

$$Gr = \frac{\beta g \Delta T L^3}{\nu^2}$$

$$Ra = GrPr = \frac{\beta g \Delta T L^3}{\nu\alpha}$$

$$t = \frac{\alpha t^*}{L^2}$$

$$x_i = \frac{x_i^*}{L}$$

$$u_i = \frac{u_i^* L}{\alpha}$$

$$T = \frac{T^* - (\Delta T^*/2)}{\Delta T^*}$$

$$p = \frac{p^* - \rho g f_i x_i}{\rho\alpha^2/L^2}$$

and σ_{ij} is total stress tensor, d_{ij} is deviatoric stress tensor, e_{ij} is the rate-of-strain tensor, δ_{ij} is Kronecker's delta function, α is thermal diffusivity, β is the coefficient of thermal expansion of the fluid, ν is kinematic viscosity, ΔT^* is temperature difference between hot and cold walls, L is characteristic length, Pr, Gr and Ra are the Prandtl, Grashof and Rayleigh numbers, respectively.

To complete the formulation of the governing equations, a set of general boundary conditions are specified as:

$$u_i=\hat{u}_i \quad \text{on } \Gamma \tag{11}$$

$$t_i=\sigma_{ij}n_j=\hat{t}_i \quad \text{on } \Gamma_2 \tag{12}$$

$$T=\hat{T} \quad \text{on } \Gamma_3 \tag{13}$$

$$q=T_{,i}n_i=\hat{q} \quad \text{on } \Gamma_4 \tag{14}$$

In (12) and (14), n_i is the direction cosines with respect to a set of axes, and the ^ denotes the function which is given on the boundaries. Moreover, the subsets Γ_1, Γ_2, Γ_3 and Γ_4 of Γ satisfy the following relations:

$$\overline{\Gamma_1 \cup \Gamma_2}=\Gamma \tag{15}$$

$$\Gamma_1 \cap \Gamma_2=\theta \tag{16}$$

and

$$\overline{\Gamma_3 \cup \Gamma_4}=\Gamma \tag{17}$$

$$\Gamma_3 \cap \Gamma_4=\theta \tag{18}$$

The superposed bar in (15) and (17) represents the total boundary enclosing the fluid and the energy transfer region, respectively. Notation θ in (16) and (18) denotes the empty set.

The initial conditions for the non-linear coupled phenomena consist of specifying the value of velocity and temperature fields at the initial time:

$$u_i(x_i, 0)=u_i^{(0)}(x_i) \tag{19}$$

$$T(x_i,0)=T^{(0)}(x_i) \tag{20}$$

with the initial velocities $u_i^{(0)}(x_i)$ satisfying the incompressibility condition:

$$u_{i,i}^{(0)}=0 \tag{21}$$

and

$$u_i^{(0)}n_i=\hat{u}_i^{(0)}n_i \quad \text{on } \Gamma_1 \tag{22}$$

DISCRETIZATION IN TIME

The time discretization for the governing equations is described in this section. Let u_i^n and T^n be the known variables of the velocity and temperature fields at time t^n where $t^n=t^{n-1}+\Delta t$ ($n=1,2,\ldots$). In the procedure of present fractional step method, the unknown velocity fields u_i^{n+1} which have been accelerated by the pressure at the advanced time level with satisfying the incompressibility constraint are calculated from the following equations:

$$\frac{u_i^{n+1}-u_i^n}{\Delta t}+u_j^n u_{i,j}^n+p_{,i}^{n+1}-Pr(u_{i,j}^n+u_{j,i}^n)_{,j}-PrRaT^n f_i=0 \tag{23}$$

$$u_{i,i}^{n+1}=0 \tag{24}$$

The boundary conditions for this system become:

$$u_i^{n+1}=\hat{u}_i \quad \text{on } \Gamma_1 \tag{25}$$

$$t_i=\{-p^{n+1}\delta_{ij}+Pr(u_{i,j}^n+u_{j,i}^n)\}\cdot n_j=\hat{t}_i \quad \text{on } \Gamma_2 \tag{26}$$

Because the pressure field p^{n+1} is solved from the pressure Poisson equation by the fractional step method in advance, p^{n+1} in the left-hand-side of (23) is assumed as an unknown variable at this stage.

After taking divergence from both sides of momentum equation (23) with satisfying the incompressibility condition such as the discrete continuum equation (24), the following pressure Poisson equation can be derived:

$$p^{n+1}_{,ii}=\frac{1}{\Delta t}u^n_{i,i}-u^n_{j,i}u^n_{i,j}-u^n_ju^n_{i,ij}+Pr(u^n_{i,j}+u^n_{j,i})_{,ij}+PrRa(T^nf_i)_{,i} \tag{27}$$

To solve (27), the following boundary conditions must be imposed:

$$p^{n+1}=\hat{p}\quad\text{on }\Gamma_2 \tag{28}$$

$$p^{n+1}_in_i=\hat{\gamma}_i\quad\text{on }\Gamma_1 \tag{29}$$

Once the pressure field has been determined from (27), the velocities u^{n+1}_i can be computed from the discrete momentum equation (23).

Finally, the temperature field T^{n+1} at all nodal points can be solved by employing the explicit Euler's first order scheme applied to the transport equation (7):

$$\frac{T^{n+1}-T^n}{\Delta t}+u^n_jT^n_{,j}-T^n_{,jj}=0 \tag{30}$$

The boundary conditions for this system are

$$T^{n+1}=\hat{T}\quad\text{on }\Gamma_3 \tag{31}$$

$$q=T^{n+1}_{,i}n_i=\hat{T}_{,i}n_i\quad\text{on }\Gamma_4 \tag{32}$$

The computational procedure of the present fractional step method for the problems under consideration can be summarized as follows:

(a) assume the initial condition for velocity, pressure and temperature fields,
(b) solve pressure field p^{n+1} from (27),
(c) solve velocity fields u^{n+1}_i from (23),
(d) solve temperature field T^{n+1} from (30),
(e) forward one time point and iterate from (b) to (d).

The basic idea of the present fractional step scheme is that; if a solution is found to the equations of momentum, continuum and energy, then the pressure field must satisfy the pressure Poisson equation (27). In other words, if a solution can be found whose pressure field satisfy the pressure Poisson equation and whose velocity fields satisfy the momentum equations, a unique solution exists for the Navier–Stokes problem, and in that case, the incompressibility constraint such as the divergence free condition is valid in a flow region.

FINITE ELEMENT METHOD

Weighted residual formulations

The weak formulation of the equations governing the coupled heat transfer problem is obtained by the method of weighted residuals. The scalar product of the weighting functions p^*, u^*_i and T^* with (27), (23) and (30), and performing an integration over the domain Ω give the following weighted residual equations:

$$\begin{aligned}\int_\Gamma p^*p^{n+1}_{,i}n_i\,d\Gamma-\int_\Omega p^*_{,i}p^{n+1}_{,i}\,d\Omega=\frac{1}{\Delta t}\int_\Omega p^*u^n_{i,i}\,d\Omega+\int_\Omega p^*_{,i}u^n_ju^n_{i,j}\,d\Omega-\\ Pr\int_\Omega p^*_{,i}(u^n_{i,j}+u^n_{j,i})_{,j}\,d\Omega-PrRa\int_\Omega p^*_{,i}(T^nf_i)\,d\Omega+\\ \int_\Gamma p^*\{-u^n_ju^n_{i,j}+Pr(u^n_{i,j}+u^n_{j,i})_{,j}+PrRa(T^nf_i)\}\cdot n_i\,d\Gamma\end{aligned} \tag{33}$$

$$\int_\Omega u_i^* u_i^{n+1}\,d\Omega = \int_\Omega u_i^* u_i^n\,d\Omega - \Delta t\left[\int_\Omega u_i^* u_j^n u_{i,j}^n\,d\Omega - \int_\Omega u_{i,i}^* p^{n+1}\,d\Omega + Pr\int_\Omega u_{i,j}^*(u_{i,j}^n + u_{j,i}^n)\,d\Omega + PrRa\int_\Omega u_i^*(T^n f_i)\,d\Omega + \int_\Gamma u_i^*\{-p^{n+1}\delta_{ij} + Pr(u_{i,j}^n + u_{j,i}^n)\}\cdot n_j\,d\Gamma\right] \tag{34}$$

$$\int_\Omega T^* T^{n+1}\,d\Omega = \int_\Omega T^* T^n\,d\Omega - \Delta t\left[\int_\Omega T^* u_j^n T_{,j}^n\,d\Omega + \int_\Omega T_{,i}^* T_{,i}^n\,d\Omega - \int_\Gamma T^* T_{,i}^n n_i\,d\Gamma\right] \tag{35}$$

where the line integral terms appeared in (33)–(35) are obtained by the divergence theorem through an integration by parts.

The following problem in the weak formulation of the pressure Poisson equation still exists. Since there is the line integral term in the left-hand-side of (33), the exact pressure gradients $\partial p^{n+1}/\partial n_i$ must be given along the boundary to get the pressure distribution exactly. In other words, the exact values of $\hat{\gamma}_i$ in (29) must be computed on the boundary, otherwise the resultant pressure field will not be correct at all and, consequently, a suspicious solution for the Navier–Stokes problem will be obtained. However, it is not easy to estimate the exact values of the Neumann boundary conditions for the pressure field in any cases, even in a simple case i.e., the fluid flow in a two-dimensional cavity, and those were treated as merely equal to zero in the conventional computations.

The idea of the present fractional step finite element scheme is that the normal gradients of the pressure field $\partial p^{n+1}/\partial n_i$ can be calculated from:

$$p_{,i}^{n+1}\cdot n_i = -\left\{\frac{u_i^{n+1} - u_i^n}{\Delta t} + u_j^n u_{i,j}^n + Pr(u_{i,j}^n + u_{j,i}^n)_{,j} + PrRaT^n f_i\right\}\cdot n_i \tag{36}$$

Substituting this equation into the line integral term in (33) properly, the weak formulation of (27) is reformulated as:

$$\int_\Omega p_{,i}^* p_{,i}^{n+1}\,d\Omega = -\frac{1}{\Delta t}\int_\Omega p^* u_{i,i}^n\,d\Omega - \int_\Omega p_{,i}^* u_j^n u_{i,j}^n\,d\Omega + Pr\int_\Omega p_{,i}^*(u_{i,j}^n + u_{j,i}^n)_{,j}\,d\Omega + PrRa\int_\Omega p_{,i}^*(T^n f_i)\,d\Omega - \int_\Gamma p^*\left(\frac{u_i^{n+1} - u_i^n}{\Delta t}\right)\cdot n_i\,d\Gamma \tag{37}$$

The Neumann boundary conditions $\partial p^{n+1}/\partial n_i$ in (33) are transformed into the term as $\{(u_i^{n+1} - u_i^n)/\Delta t\}\cdot n_i$. This manipulation is useful for solving the Navier–Stokes problems in enclosures because the non-slip condition specified on walls gives zero to the line integral terms in (37) and this is the exact Neumann boundary condition for the transformed pressure Poisson equation.

It is not straightforward to give the exact Neumann boundary conditions to (37) in case of the problems which have the open or artificial boundaries. However, it is still possible to estimate the approximate values of these conditions by proper computational techniques, e.g., iterative technique etc. (see, for example, Reference 12).

The Neumann boundary conditions to (34) are not a problem for the closed cavity flow model. Also, in other cases, these will be given as usual.

Finite element formulations

Let the pressure, velocity and temperature fields be represented within an element by the interpolation function N_α as:

$$p = N_\alpha p_\alpha \tag{38}$$

$$u_i = N_\alpha u_{\alpha i} \tag{39}$$

$$T = N_\alpha T_\alpha \tag{40}$$

and the corresponding weighting functions are:

$$p^* = N_\alpha p^*_\alpha \tag{41}$$

$$u^*_i = N_\alpha u^*_{\alpha i} \tag{42}$$

$$T^* = N_\alpha T^*_\alpha \tag{43}$$

where subscript α represents the nodal value of the αth node of the finite element in the ith direction. Substituting (38)–(43) into (37), (34) and (35), considering the arbitrariness of the weighting functions and rearranging the terms, the finite element formulations for the systems are represented as:

$$A_{\alpha i\beta i} p^{n+1}_\beta = -\frac{1}{\Delta t} H_{\alpha\beta i} u^n_{\beta i} - K_{\alpha i\beta\gamma j} u^n_{\beta j} u^n_{\gamma i} + PrRaN_{\alpha i} T^n_\alpha f_i - \hat{\Omega}_{\alpha i} \tag{44}$$

$$\bar{M}_{\alpha\beta} u^{n+1}_{\beta i} = \bar{M}_{\alpha\beta} u^n_{\beta i} - \Delta\{K_{\alpha\beta\gamma j} u^n_{\beta j} u^n_{\gamma i} - H_{\alpha i\beta} p^{n+1}_\beta + PrS_{\alpha i\beta} u^n_{\beta j} + PrRaH_{\alpha i\beta} T^n_\beta f_i - \hat{\Sigma}^{n+1}_{\alpha i}\} \tag{45}$$

$$\bar{M}_{\alpha\beta} T^{n+1}_\beta = \bar{M}_{\alpha\beta} T^n_\beta - \Delta t\{K_{\alpha\beta\gamma j} u^n_{\beta j} T^n_{\gamma j} - H_{\alpha j\beta j} T^n_\beta - \hat{\Sigma}^{n+1}_{1\alpha}\} \tag{46}$$

where

$$M_{\alpha\beta} = \int_\Omega N_\alpha N_\beta \, \mathrm{d}\Omega$$

$$A_{\alpha i\beta j} = \int_\Omega N_{\alpha,i} N_{\beta,j} \, \mathrm{d}\Omega$$

$$H_{\alpha\beta i} = \int_\Omega N_\alpha N_{\beta,i} \, \mathrm{d}\Omega$$

$$K_{\alpha\beta\gamma j} = \int_\Omega N_\alpha N_\beta N_{\gamma j} \, \mathrm{d}\Omega$$

$$S_{\alpha i\beta j} = \int_\Omega N_{\alpha,i} N_{\beta,j} \, \mathrm{d}\Omega + \int_\Omega N_{\alpha,k} N_{\beta,k} \delta_{ij} \, \mathrm{d}\Omega$$

$$\hat{\Omega}_{\alpha i} = \int_\Gamma N_\alpha \left(\frac{u^{n+1}_i - u^n_i}{\Delta t} \right) \cdot n_i \, \mathrm{d}\Gamma$$

$$\hat{\Sigma}^{n+1}_{\alpha i} = \int_\Gamma N_\alpha \hat{\gamma} \, \mathrm{d}\Gamma$$

$$\hat{\Sigma}^{n+1}_{1\alpha} = \int_\Gamma N_\alpha \hat{q} \, \mathrm{d}\Gamma$$

In (45) and (46), $\bar{M}_{\alpha\beta}$ is the lumped coefficient matrix which is adding all terms of each row of the consistent coefficient matrix $M_{\alpha\beta}$ and placing the diagonal terms with the resulted terms.

NUMERICAL RESULTS

The numerical example of the laminar incompressible viscous fluid flow problems involving the flow of natural convection due to the temperature induced buoyancy in a two-dimensional square cavity is discussed in this section. The square cavity with isothermal vertical walls at different temperature and horizontal adiabatic walls is shown in *Figure 1*. The boundary condition for fluid flow is the non-slip condition at walls and the referential value of the pressure field is specified as equal to zero at the bottom centre. Computations were carried out for the fixed Prandtl number and several Rayleigh numbers using the uniform mesh divided by the four nodes bilinear isoparametric finite elements. The mesh used in this computation has 441 nodes and 400 elements.

The Prandtl number of the solutions is 0.71 and the computed results with $Ra=10^3$, 10^4, 10^5, 10^6 and 10^7 are shown in *Figures 2* to *6*. *Figure 7* shows the comparison of the vertical and horizontal velocity profiles at the middle of cavity on these Ra numbers. The computed results of $Pr=5.12$ with $Ra=10^5$, 10^6 and 10^7 are shown in *Figures 8* to *10*.

For the flow of low Rayleigh number $Ra \leqslant 10^3$, the isothermal lines are almost linear everywhere inside such that $T=x$. For the lower Rayleigh numbers heat transfer is conduction dominated, while for higher Ra numbers the action is concentrated close to the boundaries. As the Rayleigh number increases, the convective effects become more apparent and the isothermal lines progressibly distorted. For a higher Rayleigh number flow $Ra=10^7$, the boundary layer can be observed near the walls. The present fractional step finite element scheme clearly demonstrates these phenomena.

CONCLUSIONS

In this paper, a new fractional step finite element method has been presented for solving the conductive–convective heat transfer problems. The following remarks can be summarized through the numerical example.

(1) In the time discretization of the governing equations, the pressure Poisson equation are employed by the fractional step manner. This makes the computational scheme extremely simple in algorithmic structure.

(2) In the weak formulation of the pressure Poisson equation, the Neumann boundary condition of the pressure field has been transformed into the time-derivative term of the velocity

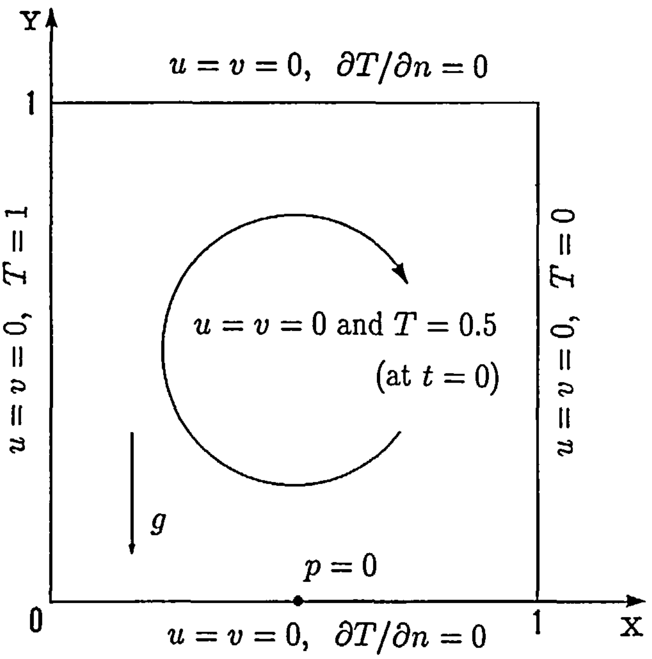

Figure 1 Analytical domain

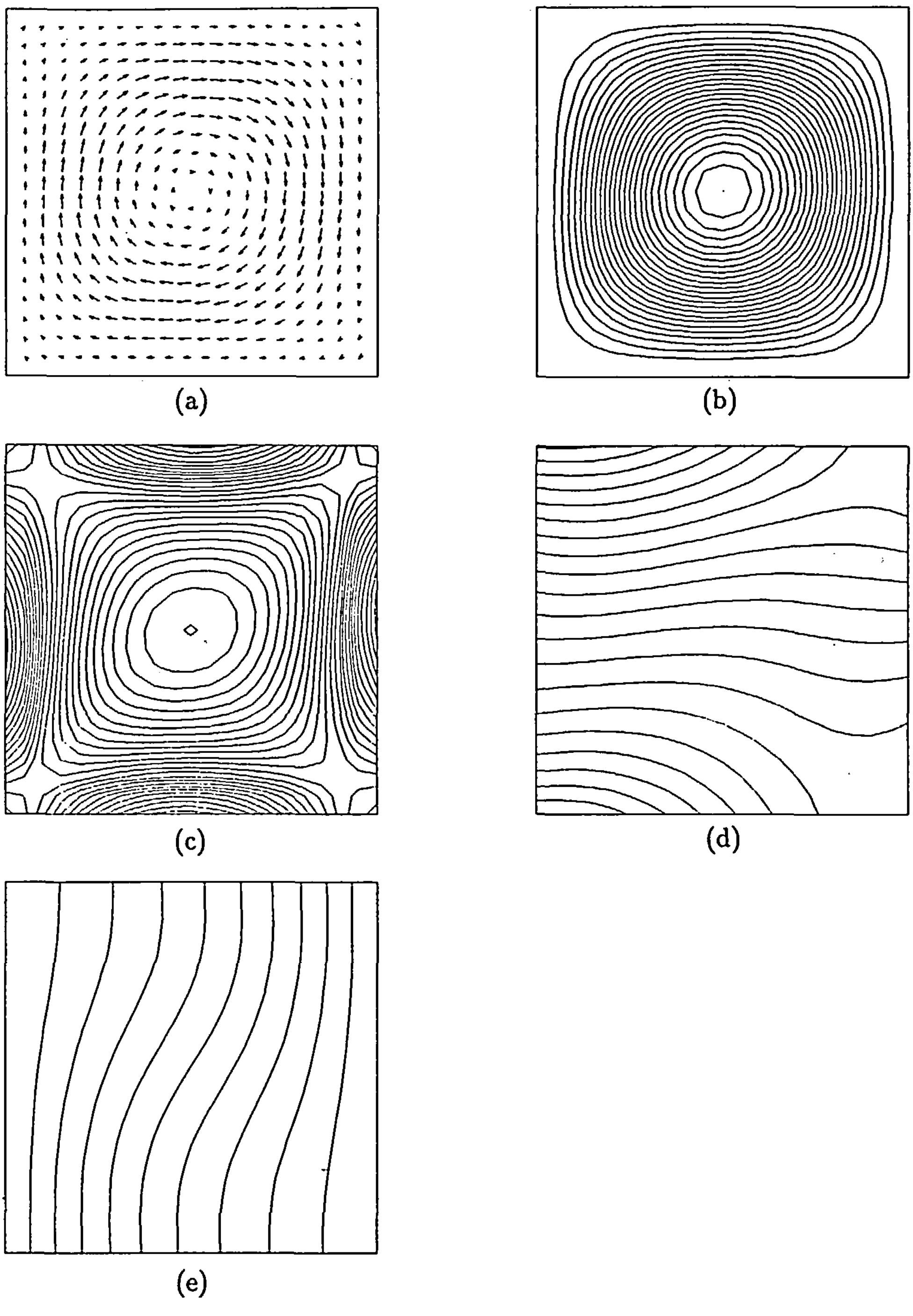

Figure 2 Computed results ($Pr=0.71$, $Ra=10^3$). (a) velocity vector; (b) stream line; (c) vorticity distribution; (d) pressure distribution; (e) temperature distribution

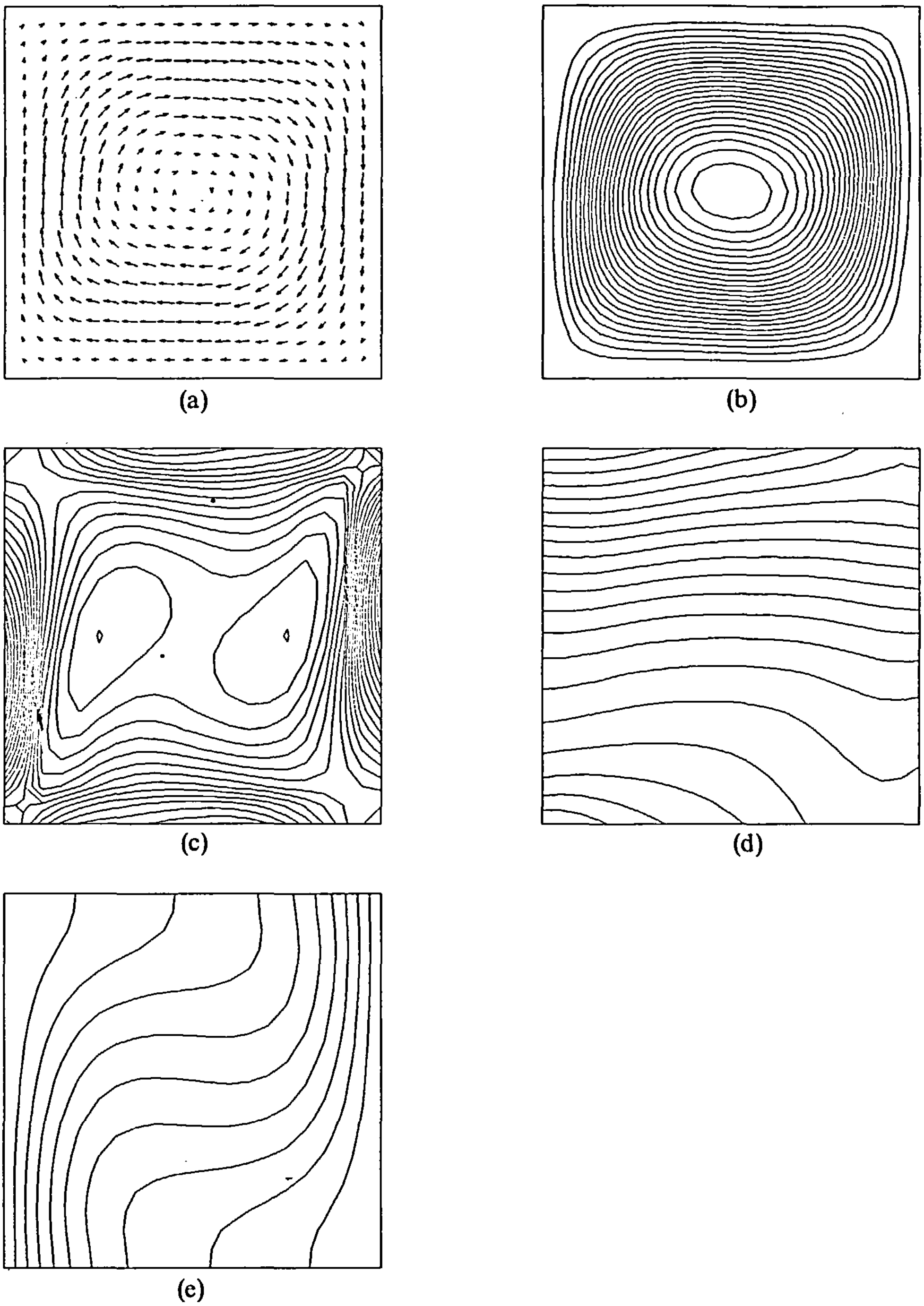

Figure 3 Computed results ($Pr=0.71$, $Ra=10^4$). (a) velocity vector; (b) stream line; (c) vorticity distribution; (d) pressure distribution; (e) temperature distribution

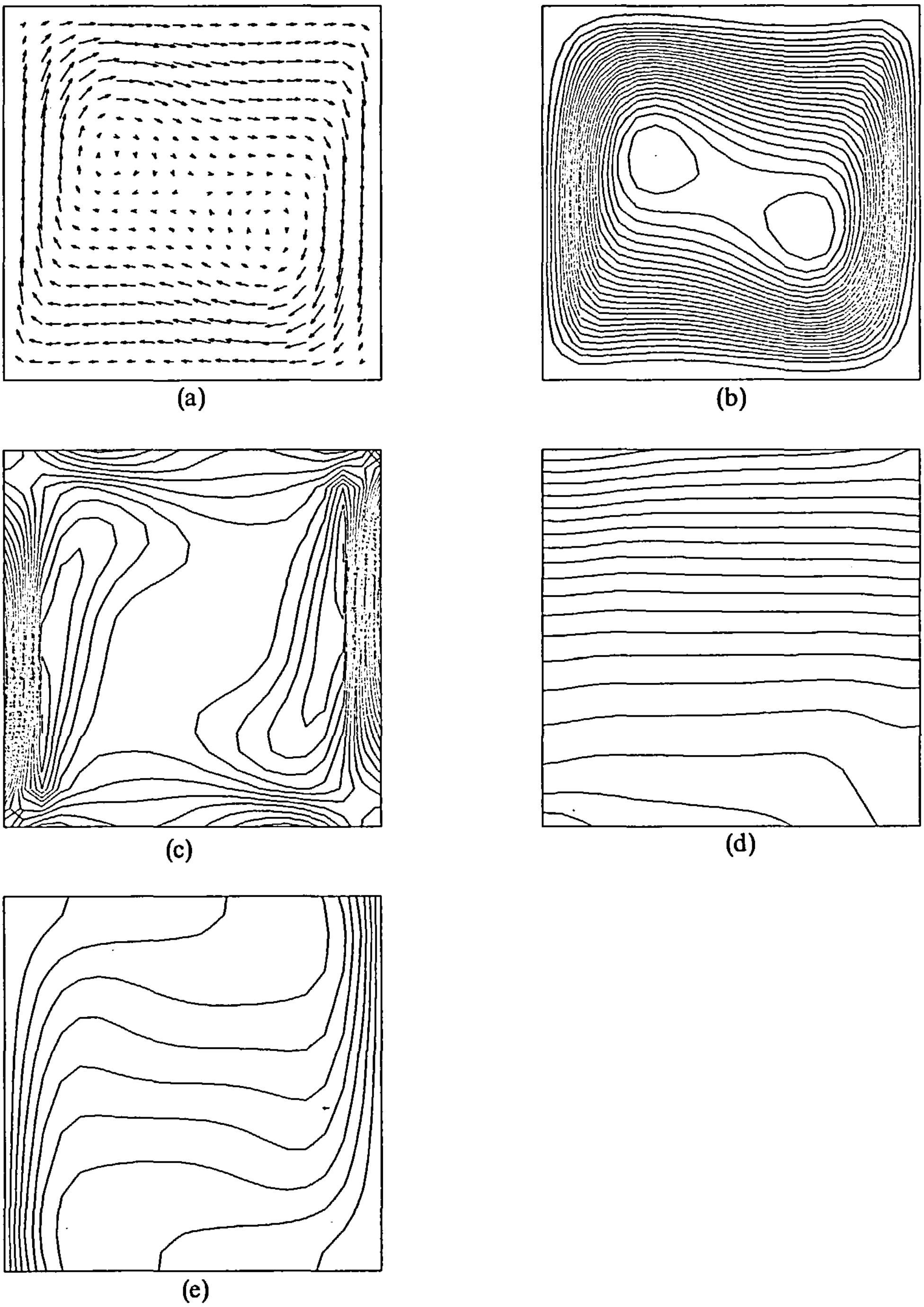

Figure 4 Computed results ($Pr=0.71$, $Ra=10^5$). (a) velocity vector; (b) streamline; (c) vorticity distribution; (d) pressure distribution; (e) temperature distribution

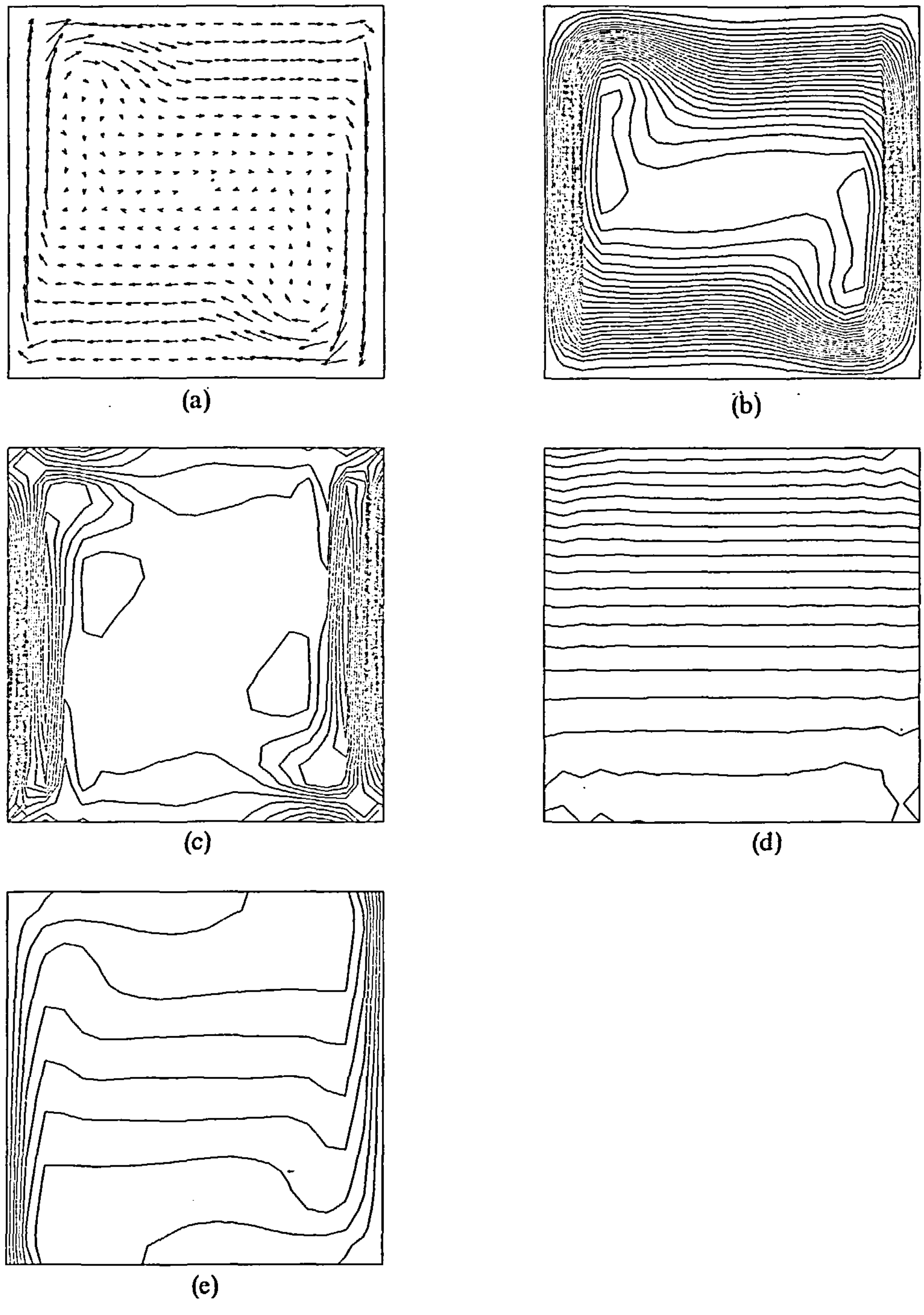

Figure 5 Computed results (Pr=0.71, Ra=10^6). (a) velocity vector; (b) stream line; (c) vorticity distribution; (d) pressure distribution; (e) temperature distribution

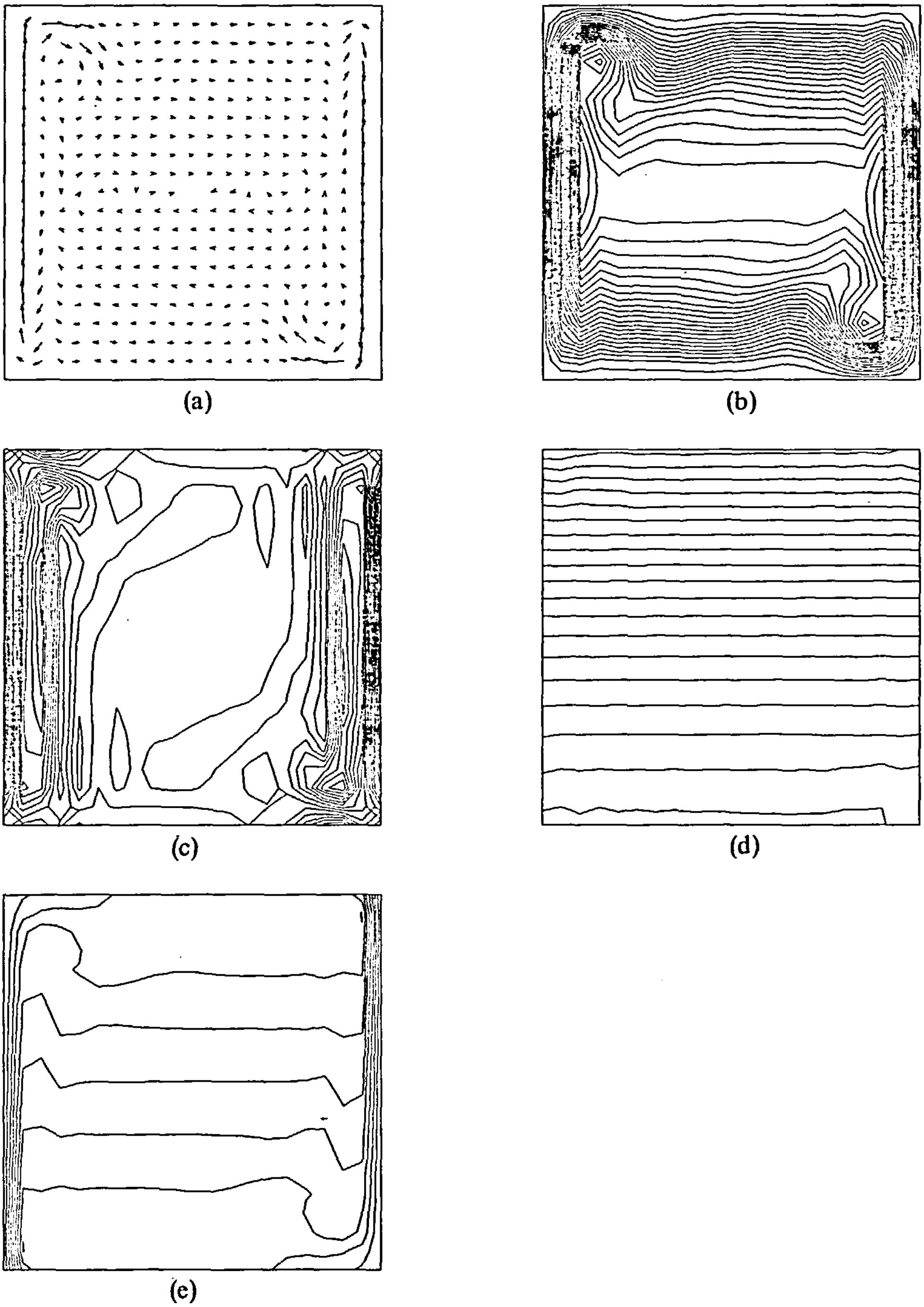

Figure 6 Computed results ($Pr=0.71$, $Ra=10^7$). (a) velocity vector; (b) stream line; (c) vorticity distribution; (d) pressure distribution; (e) temperature distribution

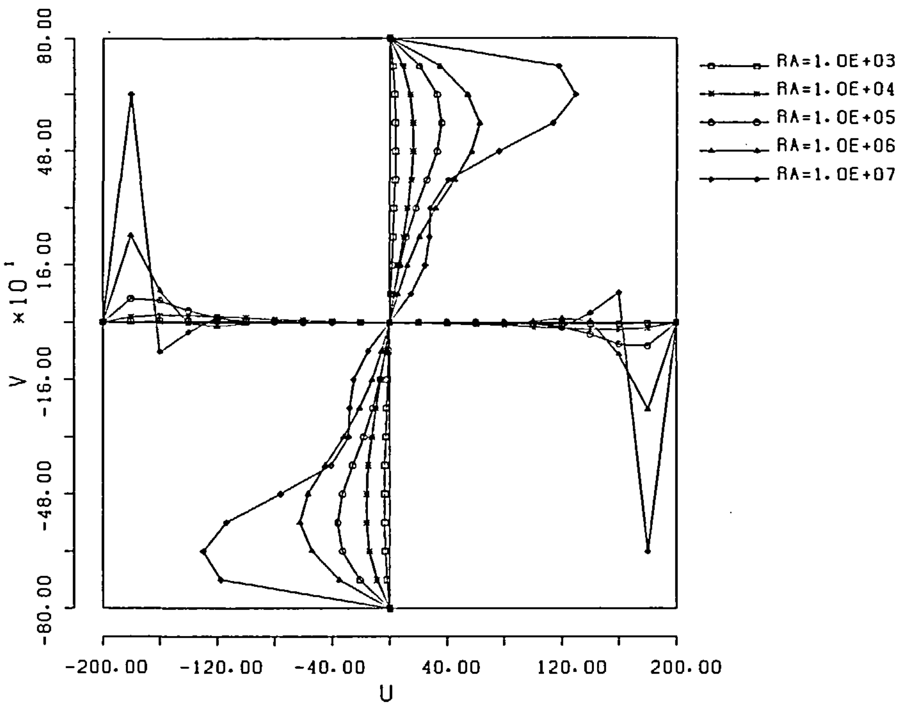

Figure 7 Comparison of computed vertical and horizontal velocity profiles at the centre of cavity on several Rayleigh numbers ($Pr=0.71$)

fields. This makes the computation much easier than before because it is not needed to estimate the special pressure flux on boundaries.

(3) The natural convection due to the temperature induced buoyancy in a two dimensional square cavity has been solved as the numerical example. Computed results show the applicability and addaptability of the present finite element method.

The calculations of the flow of high Rayleigh numbers, the flow governed by the turbulent model, the three-dimentional flow, etc. are future subjects for research from, the present method that can be extended.

ACKNOWLEDGEMENTS

Part of the computation in this research has been carried out by Mr Masahiro Hayashi, graduate student of Chuo University. The authors are grateful for the discussions with Dr. Krzysztof Wisniewski, Polish Academy of Science, former visiting research fellow of Chuo University. This research was supported by Chuo Grant for Special Research in 1990. The computations have been carried out using FACOM VP-30 supercomputer of Chuo University and Hitac S-820 supercomputer of university of Tokyo.

REFERENCES

1 Elder, J. W. Numerical experiments with free convection in a vertical slot, *J. Fluid Mech*, **24**, 823–838 (1966)
2 De Vahl Davis, G. Laminar natural convection in an enclosed rectangular cavity, *Int. J. Heat Mass Transfer*, **11**, 1675–1693 (1968)
3 Ostrach, S. Natural convection in enclosures, *Adv. Heat Transfer*, **8**, 161–227 (1972)

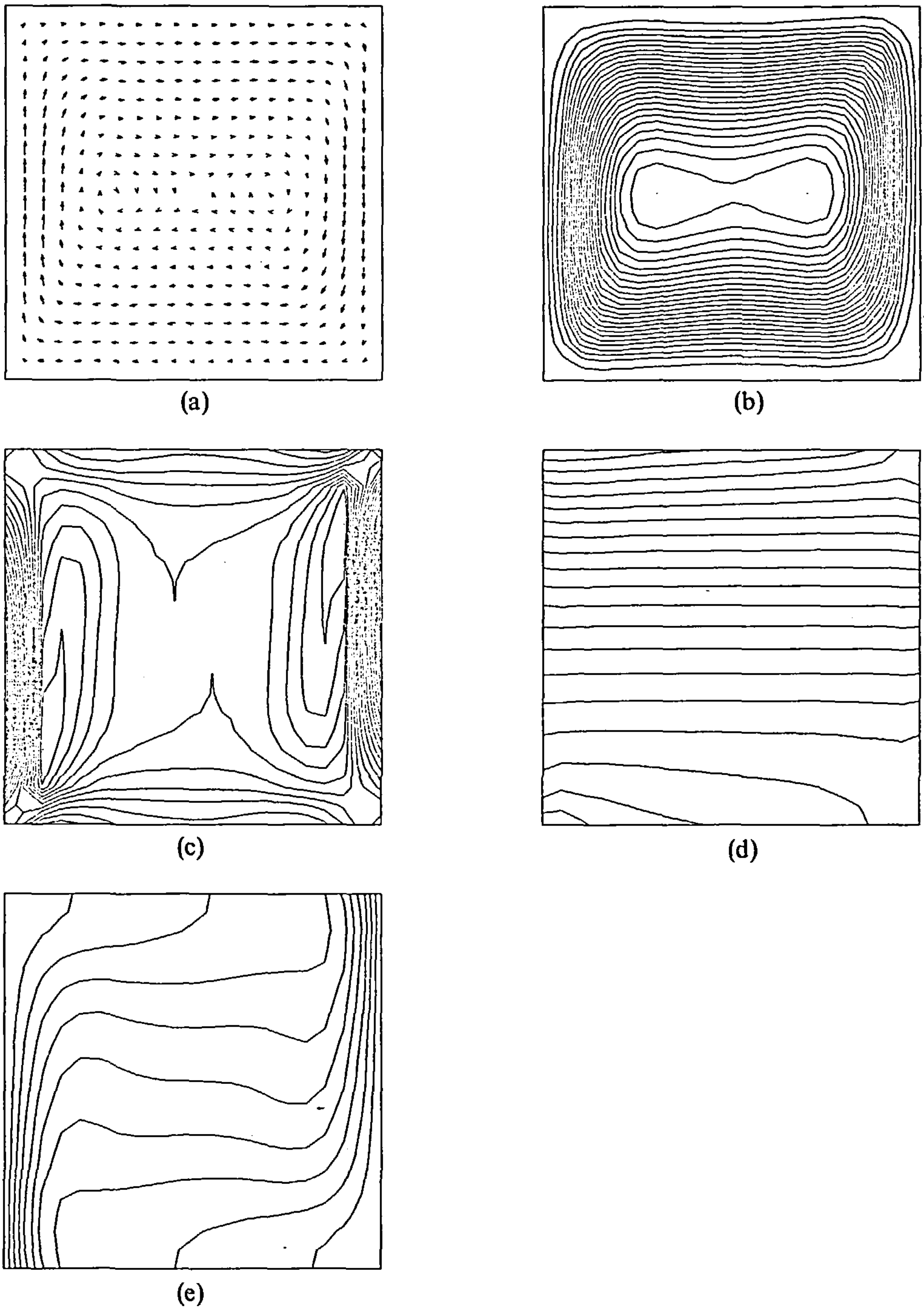

Figure 8 Computed results ($Pr=5.12$, $Ra=10^5$). (a) velocity vector; (b) stream line; (c) vorticity distribution; (d) pressure distribution; (e) temperature distribution

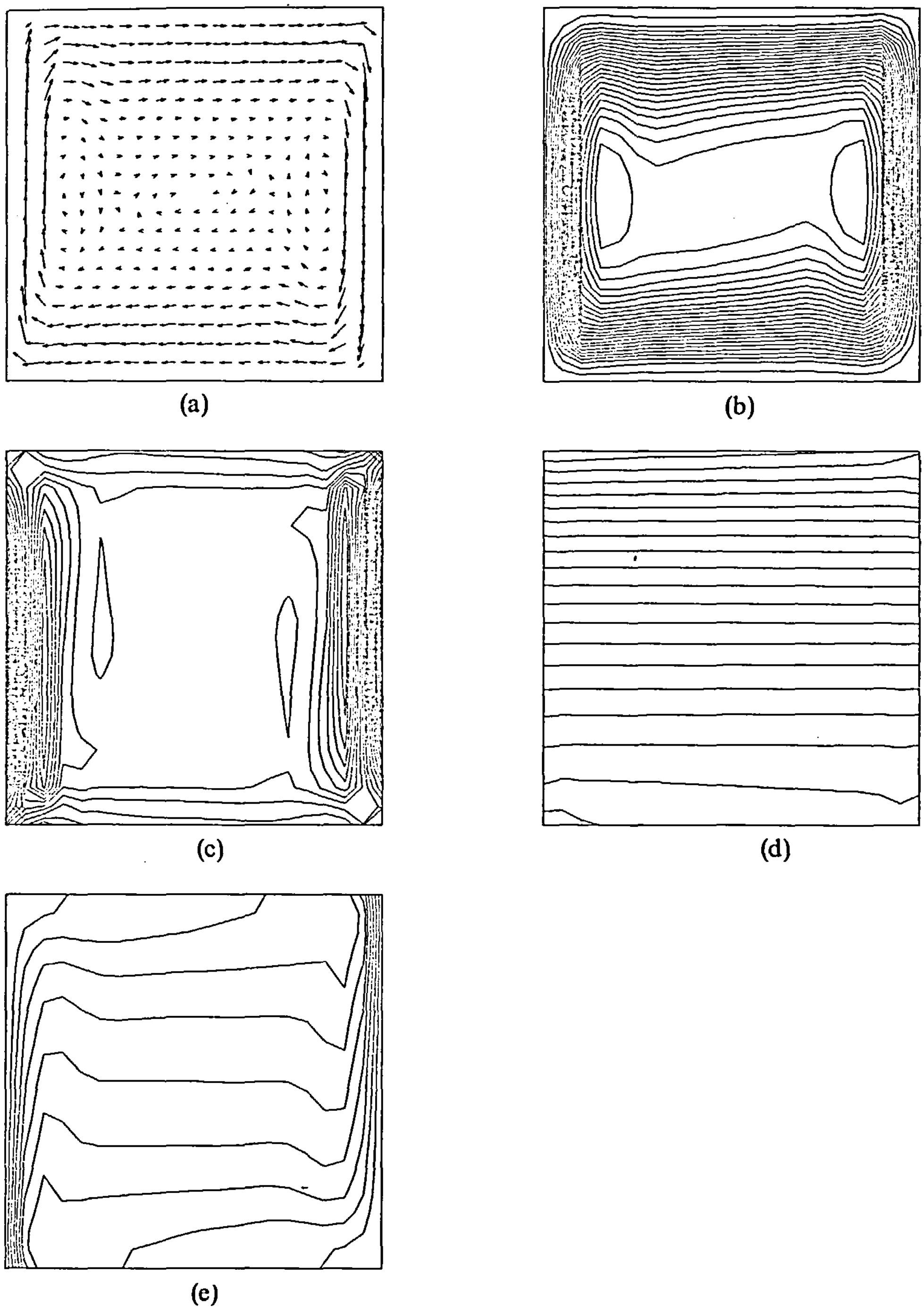

Figure 9 Computed results ($Pr = 5.12$, $Ra = 10^6$). (a) velocity vector; (b) stream line; (c) vorticity distribution; (d) pressure distribution; (e) temperature distribution

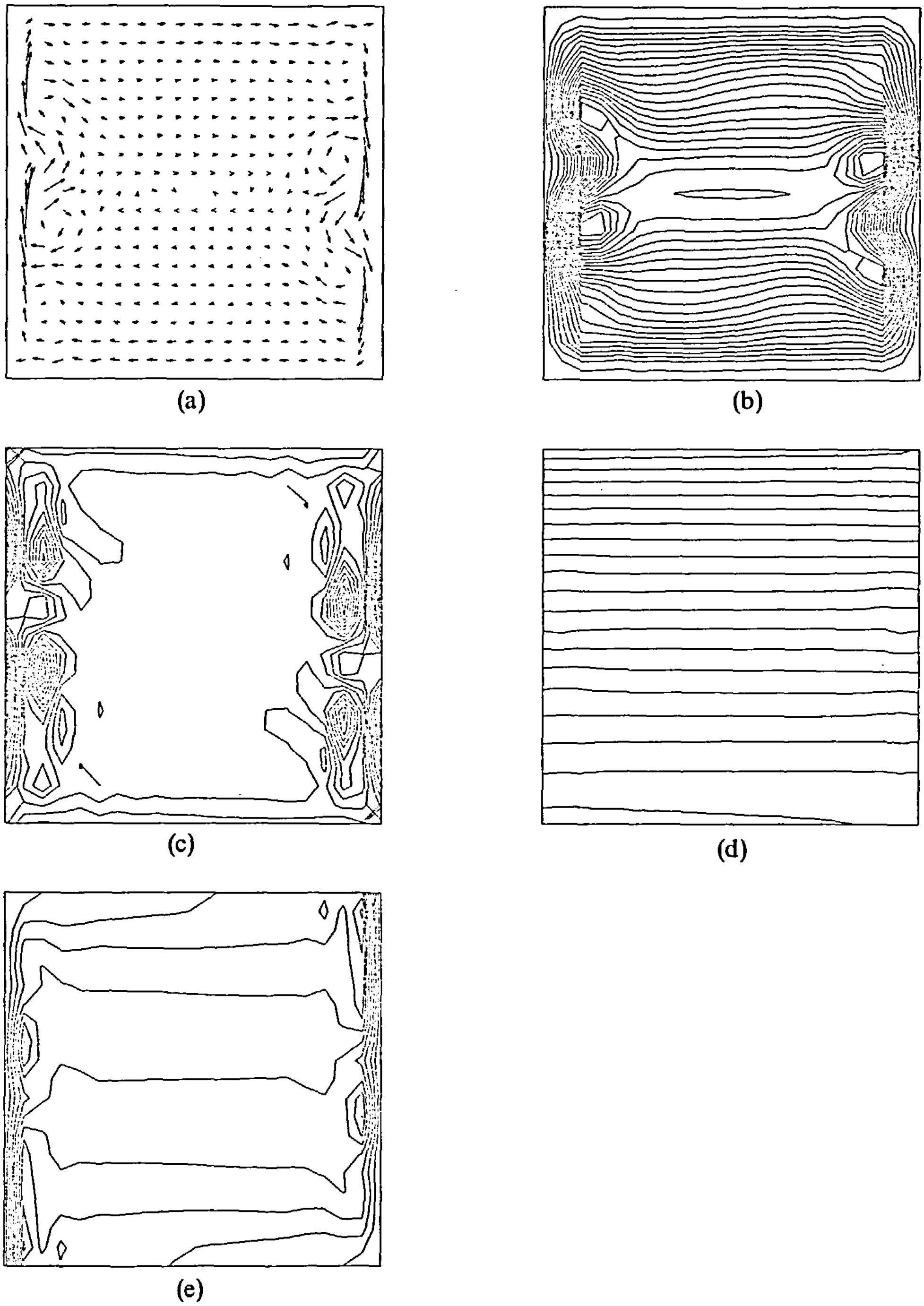

Figure 10 Computed results ($Pr=5.12$, $Ra=10^7$). (a) velocity vector; (b) stream line; (c) vorticity distribution; (d) pressure distribution; (e) temperature distribution

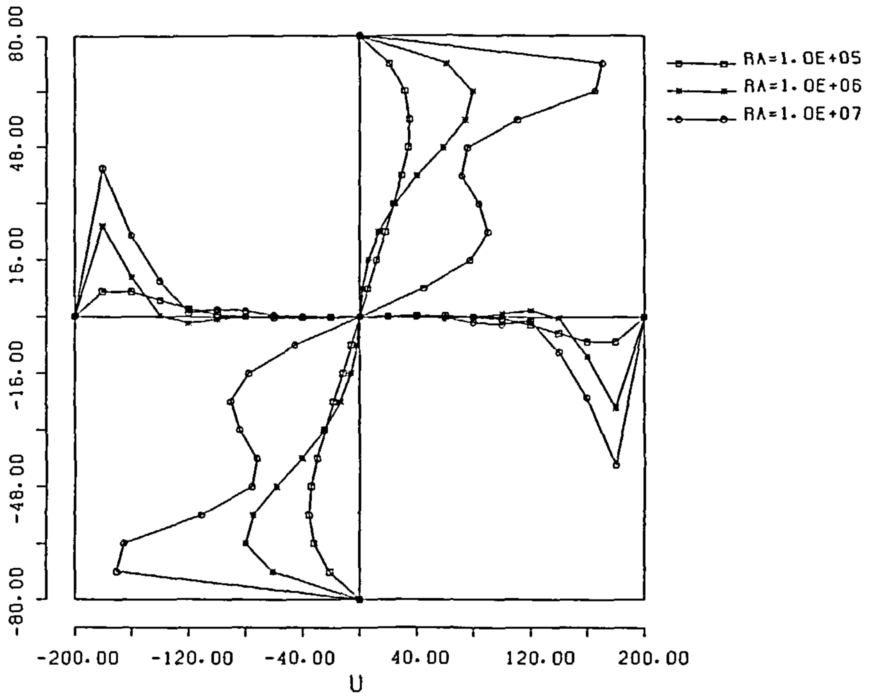

Figure 11 Comparison of computed vertical and horizontal velocity profiles at the centre of cavity on several Rayleigh numbers ($Pr=5.12$)

4 Ozoe, H. *et al.* Numerical calculations of laminar and turbulent natural convection in water rectangular channels heated and cooled isothermally on the opposing vertical walls, *Int. J. Heat Mass Transfer*, **28**, 125–138 (1985)
5 Donea, J. *et al.* Solution of the unsteady Navier–Stokes equations by a finite element projection method, *Recent Advances in Numerical Methods in Fluids*, (eds C. Taylor and K. Morgan), Vol. 24, pp. 97–132, Pineridge Press, Swansea (1981)
6 Donea, J. *et al.* Finite element solution of the unsteady Navier-Stokes equations by a fractional step method, *Comp. Meth. Appl. Mech. Eng.*, **30**, 53–73 (1982)
7 Gresho, P. M. *et al.* A modified finite element method for solving the time-dependent incompressible Navier-Stokes equations. Part 1: Theory, *Int. J. Num. Mech. Fluids*, **4**, 557–598 (1984)
8 Gresho, P. M. *et al.* A modified finite element method for solving the time-dependent incompressible Navier–Stokes equations. Part 2: Applications, *Int. J. Num. Meth. Fluids*, **4**, 619–640 (1984)
9 Gresho, P. M. *et al.* The finite element method in viscous incompressible flow, *Recent Advances in Computational Fluid Dynamics* (*Lecture Notes Eng. 43*), Springer, Berlin, pp. 148–190 (1989)
10 Kawahara, M. and Ohmiya, K. Finite element analysis of density flow using the velocity correction method, *Int. J. Num. Meth. Fluids*, **5**, 981–993 (1985)
11 Ramaswamy, B. and Kawahara, M. Arbitrary Lagrangian-Eulerian finite element method for unsteady, convective, incompressible viscous free surface fluid flow, *Int. J. Num. Meth. Fluids*, **7**, 1053–1075 (1987)
12 Shimura, M. and Kawahara, M. Two dimensional finite element flow analysis using the velocity correction method, *Proc. JSCE*, (398) I-10 (1988)
13 Hatanaka, K. and Kawahara, M. Thermal flow analysis by velocity correction method, *Proc. CMFA Okayama*, **2**, 1178–1185 (1988)
14 Mizukami, A. and Tsuchiya, M. A finite element method for the three dimensional nonsteady Navier–Stokes equations, *Int. J. Num. Meth. Fluids*, **4**, 349–357 (1984)
15 Hatanaka, K., Hayashi, M. and Kawahara, M. Numerical investigations for solving unsteady incompressible viscous fluid flow by finite element method, *Proc. ISCFD, Nagoya*, pp. 335–340 (1989)
16 Kawahara, M., Hatnaka, K. and Hayashi, M. Numerical investigations for solving unsteady incompressible viscous fluid flow by finite element method, *Comp. Meth. Surface Hydrol*, pp. 331–338 (1989)
17 Hayashi, M., Hatanaka, K. and Kawahara, M. Lagrangian finite element method for free surface Navier–Stokes flow using fractional step method, *Int. J. Num. Meth. Fluids*, in press