Computer Assisted Mechanics and Engineering Sciences, 2: 87–104, 1995.

On improvement of computational efficiency in FEM calculations of incompressible fluid flow and heat transfer[1]

Jerzy Banaszek
Institute of Heat Engineering, Warsaw University of Technology
ul. Nowowiejska 25, 00-665 Warsaw, Poland

(Received October 20, 1994)

The operator splitting algorithm has been applied in FEM analysis of fluid flow and heat transfer to improve the computational efficiency through the use of the optimum FEM models and the optimum solvers independently for convection and diffusion. The need for decoupling convection and diffusion operators in FEM calculations comes from the behavioural error analysis, where conditions have been studied for a proper representation of major physical features of the convective-diffusive transport phenomenon on a coarse grid. The accuracy and efficiency of the algorithm have been verified by solving two pertinent benchmark problems of recirculating flow and free convection. The results obtained show that solutions of both equal- and unequal-order FEM interpolations are free from wiggles and spurious pressure modes and they fit fairly well the results reported elsewhere.

1. Introduction

A major impediment in widespread use of the Finite Element Method (FEM) in modelling fluid flow and heat transfer problems is its poor computational efficiency in comparison with the FDM simulation. It results from a denser form of FEM matrices, coming from the use of irregular grids and high-order polynomial interpolations of unknown field quantities. It also comes from the application of a simultaneous solution algorithm commonly used in the early FEM analysis of incompressible fluid flow.

There are two possible ways to improve computational efficiency in the FEM simulation of field theory problems.

The first one is simply to use a coarse mesh. This is justified by the fact that the finite element discretization technique provides simple means for an accurate approximation of curvilinear geometry on a sparse grid. This feature cannot be overestimated if one takes into account the fact that the solution of many field theory problems is sensitive to even minor change in the shape of a geometrical domain. However, the problem of accuracy of the FEM solution arises here. It turns out that not only quantitative but even qualitative errors can appear on a coarse grid unless a proper discretization model is used [7,9,10]. Moreover, the classical error analysis does not give enough information about errors on a coarse mesh because it provides the convergence estimation only for an asymptotic case when grid increments tend to zero.

In this context, the present paper shows that the behavioural error analysis, where conditions are studied for a proper discrete representation of major physical features of a convective-diffusive transport phenomenon, provides means for setting up FEM models of an acceptable accuracy on a coarse mesh. To obtain such models, conditions for a proper discrete representation of both the conservation principle and the maximum principle as well as means to reduce the numerical dispersion error significantly are considered here.

[1]The paper has been presented at 2nd Joint Polish-Japanese Seminar on Advanced Modelling and Simulation in Engineering, Pułtusk, Poland, May 29–June 1, 1994.

The second possible way to improve the FEM computational efficiency lies in the use of special sophisticated acceleration techniques for the solving process. Most of them take their origin from the FDM analysis but their efficient utilization in the FEM context requires means which make possible the exploitation of the advantages of these techniques along with the retention of the commonly appreciated versatility of the finite element discretization procedure.

Such attempts are still in progress. L. Hayes [19] has proposed a patch interpolation of the Jacobian matrix determinant of isoparametric mappings to adopt the classical ADI technique in the FEM analysis for a curvilinear arbitrary domain. The multiplicative and additive correction procedure has been designed by C.N. Chen and L.C. Welford [13] to accelerate convergence by solving a sequence of problems on increasingly coarser meshes. The correct solution in a coarse grid is obtained by finding multiplicative and additive correction coefficients interpolated in accordance with the piece-wise FEM interpolation procedure. J.M. Winget and T.J.R. Hughes [35] have proposed the element-by-element algorithm (EBE), which results from the combination of the FEM discretization procedure and the approximate factorization technique. The EBE retains the generality of the FEM simulation as it does not impose any restrictions on a domain geometry and its spatial discretization. Moreover, the EBE can be easily implemented on a multi-processor computer [35]. In the context of parallel processing in FEM calculations the overlapping subdomain technique (Schwartz alternating method), originally designed for the FDM analysis, can also be used.

In incompressible flows pressure is an implicit variable that instantaneously adjusts itself in such a way that the incompressibility condition is satisfied. This implies that proper velocity and pressure fields can be obtained by means of a simultaneous solution of a coupled nonlinear set of discrete momentum and continuity equations. Such a solving technique was commonly used in the early FEM analysis of incompressible fluids (e.g. [34]). Unfortunately, this algorithm is not a reasonable choice in terms of the cost-effectiveness of calculations. Indeed, it requires a large amount of computer storage and CPU time. Therefore, the segregated solution techniques, where pressure and velocity fields are separated and momentum and mass balance equations are solved consecutively, have been elaborated for the FDM calculations [24]. This is a key reason why the FDM analysis is economical in comparison with the one inherent in those FEM calculations, where the simultaneous solution algorithm is taken. In the fractional step method (also called velocity correction method), originated by A.J. Chorin [14], the momentum equation is first solved disregarding the pressure gradient term. Then the provisional velocity field thus obtained is corrected by taking into account the pressure contribution through the enforcement of the incompressibility requirement. The most popular technique in FDM analysis is the segregated velocity-pressure solution algorithm ("guess and correct" method) of S.V. Patankar [24], where, for guessed initial values of unknowns, pressure and velocity components are found in a sequence of iterations. In each iteration cycle, the discrete momentum equations and the Poisson-like pressure correction equation, resulting from the imposition of the continuity constraint, are solved in a sequential manner and finally an updating procedure is used to obtain a velocity field that satisfies the incompressibility condition. Both above techniques were recently successfully applied in the finite element analysis, based on the Galerkin weighted residual method [15, 32], to improve the economy of calculations.

The FEM analysis of transient fluid flow and heat transfer problems can also be speeded up with the use of the splitting-up method [23], in which the discrete convection and diffusion operator is split according to physical processes and the contribution from each of these processes is calculated separately in the time integration procedure. B. Ramaswamy [25, 26, 27] has reported a successful use of this technique in the Galerkin FEM calculations whereas J. Banaszek [8] has used it succesfully in the control-volume formulation of FEM.

This splitting up technique, in which the convection and diffusion in the momentum and energy balance equations are treated in two distinct consecutive phases, is used in the present study so that the optimum numerical models and the most efficient solvers of a set of algebraic equations can be chosen independently for convection and diffusion.

The above mentioned behavioural error analysis leads to the general conclusion that accuracy and computational efficiency of FEM approximation for simultaneous convection and diffusion of a field quantity depend on the proper choice of a discrete model which is due to the intensity of both forms of transport. For example, in diffusion-type problems the lumped capacity (mass) matrix model is a reasonable choice from the point of view of stability and early-time solution accuracy [3, 5]. On the other hand, in the case when convection dominates (moderate and high Peclet numbers), the consistent capacity (mass) matrix model significantly reduces numerical dissipation and dispersion [6, 9]. Hence, there is no general FEM approximation equally accurate and efficient for both problems. Each of them needs individual approach, for each of them different optimum discrete models should be used to obtain an efficient numerical technique. In fluid flow and heat transfer problems, where a coupled convection-diffusion occurs, such a technique can be developed through decomposition of both forms of momentum and energy transfer in the time integration procedure of the splitting-up method.

Therefore, to utilize the results drawn from the theoretical error analysis in FEM calculations of fluid flow and heat transfer, the splitting-up technique has been used here along with Chorin's velocity correction method. First, the momentum equations with the disregarded pressure terms are solved consecutively for each provisional velocity component in two steps: pure diffusion and then pure convection. Next, the pressure is calculated from the Poisson equation resulting from a combination of the continuity equation and divergence of the momentum equation. Finally, the provisional velocity field is corrected by taking into account the pressure contribution to acceleration through the enforcement of the incompressibility condition. This solution algorithm has been tested on several benchmark problems, commonly used to verify the accuracy and economy of numerical solutions for incompressible fluids. The exemplifying results of two selected problems, i.e. driven cavity flow and free convection in a square enclosure, are given in the paper.

2. FEM EQUATIONS FOR CONVECTION-DIFFUSION PROBLEMS

The balance equation for the scalar quantity Φ, which can denote mass, momentum components or enthalpy, transferred by convection and diffusion in a small but finite control-volume Ω_k confined by the surface Γ_k, has the following integral form,

$$\int_{\Omega_k} \rho \frac{\partial \Phi}{\partial t}\, d\Omega + \int_{\Gamma_k} \left(\rho u_i \Phi - \lambda_{ij} \frac{\partial \Phi}{\partial x_j} \right) n_i\, d\Gamma = \int_{\Omega_k} S\, d\Omega, \tag{1}$$

which states that the change rate of Φ is influenced both by convective-diffusive flux through the boundary Γ_k and sources occurring in the domain Ω_k. This equation is a basic one for setting up nodal equations for the control-volume based FEM (CVFEM) [2, 4, 31]. For an infinitesimally small control-volume the above balance equation assumes a local conservative form

$$\rho \frac{\partial \Phi}{\partial t} + \frac{\partial}{\partial x_i} \left(\rho u_i \Phi - \lambda_{ij} \frac{\partial \Phi}{\partial x_j} \right) = S \qquad \text{for } i, j = 1, 2, 3, \tag{2}$$

which, through the incompressibility condition

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{3}$$

is differentially equivalent to its non-conservative form

$$\rho \frac{\partial \Phi}{\partial t} + \rho u_i \frac{\partial \Phi}{\partial x_i} - \frac{\partial}{\partial x_i} \left(\lambda_{ij} \frac{\partial \Phi}{\partial x_j} \right) = S \qquad \text{for } i, j = 1, 2, 3. \tag{4}$$

An integral weak formulation is obtained here by weighting the residual R_Ω of the above partial differential equation and the residual R_Γ of the boundary conditions (resulting from an element-wise interpolation of unknowns) in the whole domain through the weighting functions W_k defined in a local element basis [36]. Thus, a set of nodal equations for the Petrov-Galerkin FEM (PGFEM) is established

$$\int_\Omega W_k R_\Omega \,\mathrm{d}\Omega + \int_\Gamma W_k R_\Gamma \,\mathrm{d}\Gamma = 0 \qquad \text{for } k = 1, 2, \ldots, NP_\Phi . \tag{5}$$

To obtain a discrete form of Eq. (1) or Eq. (5) two consecutive discretization steps are carried out. Firstly, the FEM discretization is defined by the piece-wise independent interpolations of the domain geometry, velocity components and the field quantity sought,

$$\begin{aligned} x_i(\bar{\zeta}) &= M_p(\bar{\zeta})\, x_{ip} , \\ u_i(\bar{\zeta}, t) &= P_l(\bar{\zeta})\, u_{il}(t) , \\ \Phi(\bar{\zeta}, t) &= N_m(\bar{\zeta})\, \Phi_m(t) , \end{aligned} \tag{6}$$

$$\text{for} \quad i = 1, 2, 3; \quad l = 1, 2, \ldots, NP; \quad p = 1, 2, \ldots, NP_g ; \quad m = 1, 2, \ldots, NP_\Phi .$$

Next, a set of ordinary differential equations for the semi-discrete FEM model thus obtained, which can be written in a concise matrix form

$$\mathbf{C}\, \frac{\mathrm{d}\{\mathbf{\Phi}\}}{dt} + (\mathbf{K} + \mathbf{K_u})\{\mathbf{\Phi}\} = \mathbf{R} \tag{7}$$

where

$\mathbf{C}$ – capacity (mass) matrix,

$\mathbf{K}$ – diffusion matrix,

$\mathbf{K_u}$ – convection matrix,

$\mathbf{R}$ – r.h.s. vector resulting from internal sources and natural boundary conditions,

is integrated in time by means of the one-step implicit finite difference scheme [29],

$$\{\mathbf{\Phi}\}^{n+1} = \{\mathbf{\Phi}\}^n + \Theta \Delta t \left\{ \frac{\mathrm{d}\mathbf{\Phi}}{\mathrm{dt}} \right\}^{n+1} + (1 - \Theta)\Delta t \left\{ \frac{\mathrm{d}\mathbf{\Phi}}{\mathrm{dt}} \right\}^{n} , \tag{8}$$

where the scheme impliciteness parameter $0 \le \Theta \le 1$.

This leads to a set of algebraic equations of a fully discrete FEM model, which can be written in the following matrix form, common for both the CVFEM and PGFEM,

$$[\mathbf{C} + \Theta \Delta t(\mathbf{K} + \mathbf{K_u})]\, \{\mathbf{\Phi}\}^{n+1} = [\mathbf{C} - (1 - \Theta)\Delta t(\mathbf{K} + \mathbf{K_u})]\, \{\mathbf{\Phi}\}^{n} + \bar{\mathbf{R}} , \tag{9}$$

where $\bar{\mathbf{R}} = \Delta t\, [\Theta \mathbf{R}^{n+1} + (1 - \Theta)\mathbf{R}^n]$.

The way chosen for spatial approximation of a time derivative of the field quantity sought determines the form of the capacity matrix, which in turn significantly influences the accuracy and stability of FEM solutions to transient convection-diffusion problems.

When a spatial approximation of a time derivative of Φ is assumed in accordance with the FEM interpolation technique

$$\frac{\partial \Phi(\bar{\zeta}, t)}{\partial t} = N_m(\bar{\zeta})\, \frac{\mathrm{d}\Phi_m(t)}{\mathrm{d}t} ,$$

the so-called consistent capacity (mass) model, further labelled as C-model,

$$\begin{aligned} C_{km} &= \sum_{e=1}^{NE} \int_{\Omega_k^e} \rho N_m \,\mathrm{d}\Omega , \\ C_{km} &= \sum_{e=1}^{NE} \int_{\Omega^e} \rho W_k N_m \,\mathrm{d}\Omega , \end{aligned} \tag{10}$$

is obtained for the CVFEM and PGFEM, respectively.

On the other hand, averaging spatial changes of the time derivative over a control-volume or with special lumping techniques [17,36] leads to the lumped capacity (mass) model,

$$C_{kk} = \sum_{e=1}^{NE} \int_{\Omega^e} \rho \, d\Omega\,, \qquad C_{km} = 0;$$
$$C_{kk} = \sum_{e=1}^{NE} \int_{\Omega^e} \rho W_k \, d\Omega\,, \qquad C_{km} = 0; \tag{11}$$

further labelled as L-model, for the CVFEM and PGFEM, respectively.

3. Behavioural error analysis

The commonly used error analysis for a discrete model gives an estimation of the truncation error, referring to the inaccuracy caused by ignoring high order terms in an infinite Taylor series expansion of an unknown. This error, which is due to the finite step sizes of both spatial and temporal discretizations, vanishes when the grid increments Δx_i and Δt tend to zero. This is, however, a rather dubious consolation if one takes into account the fact that a complex mathematical description of fluid flow and heat transfer problems, given by a set of coupled nonlinear partial differential equations, usually forces the use of grids of rather moderate density. Otherwise, the computational economy of a numerical analysis is not retained. Unfortunately, finite sizes of space-time discretization steps can cause unacceptable quantitative and frequently even qualitative errors of an approximate solution, unless a proper numerical model is developed. To make up such FEM models, the behavioural error analysis, originated by S.P. Roache in the FDM context [30], is used here. This means that the quality of the numerical analogue is judged in terms of how the physical features of the convection-diffusion transport phenomenon are reflected within a FE grid [7,10]. In fact, for engineers and physicists such appraisal seems to be the most convincing way of verifying the correctness of a numerical model employed.

Certain comments and conclusions drawn from this error analysis are given further to show simple and time-saving means of setting up proper FEM models on a coarse mesh.

3.1. Conservative property of FEM models

The conservative property, which arises directly from the conservation principle for a scalar field quantity, is one of the most meaningful and desired features of a credible numerical analogue. If a discrete model possesses this property, the field quantity sought is correctly balanced within a considered domain and/or within its subdomains, irrespective of the discretization pattern and grid density used [30].

It is shown above that the FEM nodal equations can be established in two different ways, either by means of the weighted residual approach to the governing PDE (GFEM or PGFEM) or by setting up the integral balance equation for the scalar quantity within the control-volume surrounding each nodal point (CVFEM). This gives reason to distinguish two different forms of the conservative property for the FE analogues, namely the global and local ones [7,10].

The Global Conservative Property (GCP) is obtained if a FEM solution satisfies the integral balance equation (Eq. (1)) within the whole region Ω, irrespective of the discretization pattern, the element shape, weighting and interpolating functions used. Furthermore, when Eq. (1) is applied to each control volume Ω_k bounded by imaginary and/or real boundary Γ_k, the Local Conservative Property (LCP) is achieved.

The weighted residual technique consists in a global minimization of the PDE residual within the whole domain Ω (Eq. (5)). Therefore, its individual nodal equation cannot, in general, be considered as the local balance equation. Therefore, only the global form of the conservative property can be associated with this model [7,10]. The sufficient condition for the GCP requires the weak formulation given in Eq. (5) to be established on the basis of the conservative form of PDE (Eq. (2)) and the sum of all weighting functions W_k to be equal to unity at any point of the considered domain [10].

On the other hand, the CVFEM provides a numerical analogue inherently possessing the LCP, and in consequence also the GCP, owing to the fact that its nodal equation is obtained by setting up the local integral balance of Φ within the control-volume Ω_k, confined by the boundary Γ_k running inside those elements which share the common node k.

Although not indispensable, the LCP is desirable for it frequently offers better accuracy and less stringent stability requirements of the FEM solution in comparison with the ones obtained from only globally conservative weighted residual FEM models. This is shown in [3, 5, 10], where accuracy of the GFEM and CVFEM solutions is compared for the selected steady-state and transient diffusion problems.

In the FEM analysis of convection-dominated problems, the correctness of both mass and convected scalar quantity balances should be taken into account simultaneously [10]. Otherwise, poor interpolation of a solenoidal velocity field over a coarse low-order FE grid can cause a mass balance error which might significantly influence the accuracy of the FEM solution for the scalar quantity sought.

To reduce this inaccuracy, the PGFEM (GFEM) nodal equations can be established on the basis of the non-conservative form of PDE (Eq. (4)). This somewhat confusing conclusion can be explained as follows [10]. As the FEM interpolation of a velocity field (given in Eq. $(6)_2$) does not generally satisfy the incompressibility condition at any point of an element domain,

$$u_i(\bar{\zeta}) = P_l(\bar{\zeta})\, u_{il} \implies \frac{\partial P_l(\bar{\zeta})}{\partial x_i}\, u_{il} \neq 0\,,$$

a redundant mass occurs within an infinitesimally small volume

$$\mathrm{d}m = \rho\, \frac{\partial P_l(\bar{\zeta})}{\partial x_i}\, u_{il}\, \mathrm{d}\Omega\,,$$

which gives an additional amount of the field quantity in Ω_e,

$$\Delta E = \int_{\Omega^e} \rho\, \frac{\partial P_l(\bar{\zeta})}{\partial x_i}\, u_{il}(N_m \Phi_m)\, \mathrm{d}\Omega\,. \tag{12}$$

Comparing the above integral with the sum of the element convection integrals for the PGFEM based on the conservative form of PDE (Eq. (2)),

$$\begin{aligned}\sum_k \int_{\Omega^e} \rho W_k \frac{\partial(u_i\Phi)}{\partial x_i}\, \mathrm{d}\Omega &= \int_{\Omega^e} \rho \frac{\partial}{\partial x_i}\left[(P_l u_{il})(N_m\Phi_m)\right] \mathrm{d}\Omega \\ &= \int_{\Omega^e} \rho\left[(P_l u_{il})\left(\frac{\partial N_m}{\partial x_i}\Phi_m\right)\right]\mathrm{d}\Omega + \int_{\Omega^e}\rho\left[\left(\frac{\partial P_l}{\partial x_i}u_{il}\right)(N_m\Phi_m)\right]\mathrm{d}\Omega\,,\end{aligned} \tag{13}$$

one can conclude that the integral (12) coincides with the second one on the right-hand side of Eq. (13). The second integral exists in the above conservative formulation but it does not appear in the non-conservative model and thus the impact of the mass balance error on the FEM solution for Φ is reduced in the formulation based on Eq. (4).

In the CVFEM, the same effect is achieved when the local integral balance of Φ within the control volume is set up for a corrected amount of mass comprised in this subdomain [10], i.e. when a redundant amount of the field quantity resulting from redundant mass coming through the boundary Γ_k,

$$\Delta m = \int_{\Gamma_k} \rho[P_l u_{il}] n_i\, \mathrm{d}\Gamma \implies \Delta E_k = \left(\int_{\Gamma_k} \rho[P_l u_{il}] n_i\, \mathrm{d}\Gamma\right)\Phi_k\,, \tag{14}$$

is subtracted from the nodal balance equation (1).

The FEM models thus obtained are not likely to satisfy the conservative property but they can provide more accurate results because the impact of inaccuracy in the approximation of the continuity condition over a FE grid on the calculated field quantity is significantly reduced. In consequence, an acceptable accuracy of the FEM solution can often be achieved on a coarser mesh as it is shown in [10] for a steady-state convection problem. This, in turn, leads to considerable savings in both computer storage and CPU time [10].

3.2. Discrete maximum principle

One of the most important properties of the diffusive transport phenomenon is the maximum principle, which imposes physical limits on the extreme values of the unknown. For instance, when heat conduction with no heat source is considered, the principle states that both the maximum and the minimum values of temperature can only occur on a domain boundary or at an initial time. If a numerical model does not preserve this principle within a division grid, its solution may exhibit strong spatial and temporal oscillations and it may even take values which are outside of a physically justified range [3, 7, 21, 28]. To avoid this error in the FEM analysis one should select a space-time discretization pattern in a way which ensures the fulfilment of the Ciarlet matrix criterion defined in the FDM context in [12], which imposes inequality relations for individual terms of the capacity and diffusion matrices [7]. These relations put restrictions on the grid Fourier number, on the ratio of spatial division steps and on the parameter Θ of the time marching scheme (given in Eq. (8)). Their thorough examination leads to the conclusion that they are less stringent for the CVFEM than for the GFEM. Furthermore, the L-model of the capacity matrix has only the upper bound for the grid Fourier number whereas the C-model possesses both lower and upper limits [7, 28].

Convection is a one-way transport phenomenon, i.e. the convected scalar quantity travels only in the direction of the velocity vector, downwind to the flow. Unfortunately, the classical FDM and FEM models do not necessarily obey this physically meaningful requirement unless the discretization grid is sufficiently dense. This can cause considerable spatial oscillations of the numerical solution, called wiggles [7, 11, 21], particularly in the case where a strong gradient of the transported variable occurs along streamlines. In consequence, the FEM solution often takes values which are outside of physically justified limits. This can also be viewed as a violation of the maximum principle in the convective-diffusive transport of the scalar field quantity [21]. To suppress wiggles either a very dense FE grid should be used or special upwind techniques, similar to those commonly incorporated in the FDM models (e.g. [24]), should be developed. In the PGFEM, the convective motion upwind to the flow is eliminated with the use of high order unsymmetric polynomial weighting functions (e.g. [11]). This way cannot be used, however, in the CVFEM where the weighting technique is not exploited. The effect of upwinding can be obtained here by developing a flow-oriented form of the interpolation function for the field quantity Φ, i.e. a form which depends on both the direction and intensity of convection. Although several such techniques have been proposed in the literature for triangular [2] and rectangular [4, 20, 31] FE grids, there is no general way to obtain consistent approximations of all terms in the balance equation (Eq. (1)). Therefore, when choosing an upwinding technique in the control-volume FEM formulation one should take into account some characteristic features of the problem considered, like the scale of expected flow recirculations, contributions of the source terms and direction of the convective transport [4, 31].

It is a common way to use the upwind procedures, developed for a steady-state problem, in numerical modelling of transient convection dominated flows, in the hope that they also perform quite well in this case. Unfortunately, this is not necessarily true due to the considerable dispersion error which may occur when approximation of the temporal term in the balance equation (Eq. (1) or Eq. (2)) is not properly defined. The source of this error and some remedies available are further discussed in the subsequent paragraph.

3.3. Numerical dispersion — Fourier mode analysis

Fourier mode analysis, commonly used in the von Neumann stability method, also provides a convenient tool for the examination of the behaviour of FEM solutions on a coarse mesh. It is based on the Fourier series representation of both the continuous solution and its numerical counterpart. By virtue of the linearity assumption Fourier modes are mutually independent. Therefore, the analysis can focus on one wave only, taken from a discrete spectrum of modes based on a discretization grid.

For any initial wave

$$\Phi_0(\bar{x}) = \phi \exp\left(\vec{i} k_{u_j} u_j\right) , \tag{15}$$

where $\vec{i} = \sqrt{-1}$ and k_{u_j} is a component of the wave number vector $\mathbf{k_u}$, the following form of the Fourier mode for a continuous model is assumed,

$$\Phi(\bar{x}, t) = \Phi_0(\bar{x}) \exp\left[\left(-\beta - \omega \vec{i}\right) t\right] , \tag{16}$$

and making use of Eq. (2) with $S = 0$, the damping and dispersion relations for a continuous model are obtained:

$$\beta = \lambda_{ij} k_{ui} k_{uj} , \tag{17}$$

$$\omega = k_{u_j} u_j . \tag{18}$$

The first relation (Eq. (17)) is a parabolic function of the components of the wave number vector $\mathbf{k_u}$, whereas the second one is a linear function of this vector with a constant phase speed c.

$$\omega = c|k_{u_j}| = c\left[\sum_j (k_{u_j})^2\right]^{\frac{1}{2}} . \tag{19}$$

These relations are further compared with those for discrete models which are relevant here.

For a two-dimensional semi-discrete model in a regular grid the nodal FEM equation can be written in a general operator form

$$\mathbf{A}_t \left(\frac{\mathrm{d}\Phi_{ij}}{\mathrm{d}t}\right) = \mathbf{A}_s(\Phi_{ij}) \tag{20}$$

where

$\mathbf{A}_t$ – spatial shifting operator for temporal derivative,

$\mathbf{A}_s$ – spatial shifting operator for convection and diffusion terms,

and the Fourier mode takes the form

$$\begin{aligned}\Phi_h(i\Delta x_1, j\Delta x_2, t) = \phi \exp\left[(k_{u1} i\Delta x_1 + k_{u2} j\Delta x_2)\vec{i}\right] \exp\left[\left(-\bar{\beta} - \bar{\omega}\vec{i}\right) t\right] = \\ = \Phi(x_{1i}, x_{2j}, t) \exp\left\{\left[(\beta - \bar{\beta}) + (\omega - \bar{\omega})\vec{i}\right] t\right\}\end{aligned} \tag{21}$$

where

$\Phi(x_{1i}, x_{2j}, t)$ – the continuous solution at (x_{1i}, x_{2j}, t),

$\bar{\beta}$ – damping parameter for semi-discrete model,

$\bar{\omega}$ – wave frequency for semi-discrete model.

The expression $\exp[(\beta - \bar{\beta})t]$ is a measure of additional dissipation, whereas the expression $(\omega - \bar{\omega})t$ determines the phase-shift error resulting from the FEM spatial discretization. In Figs. 1 and 2 the damping parameter and the phase speed are compared for a continuous model and semi-discrete FEM models on a bilinear square grid for different dimensionless wave numbers $k_{u1} = k_{u2} = k$.

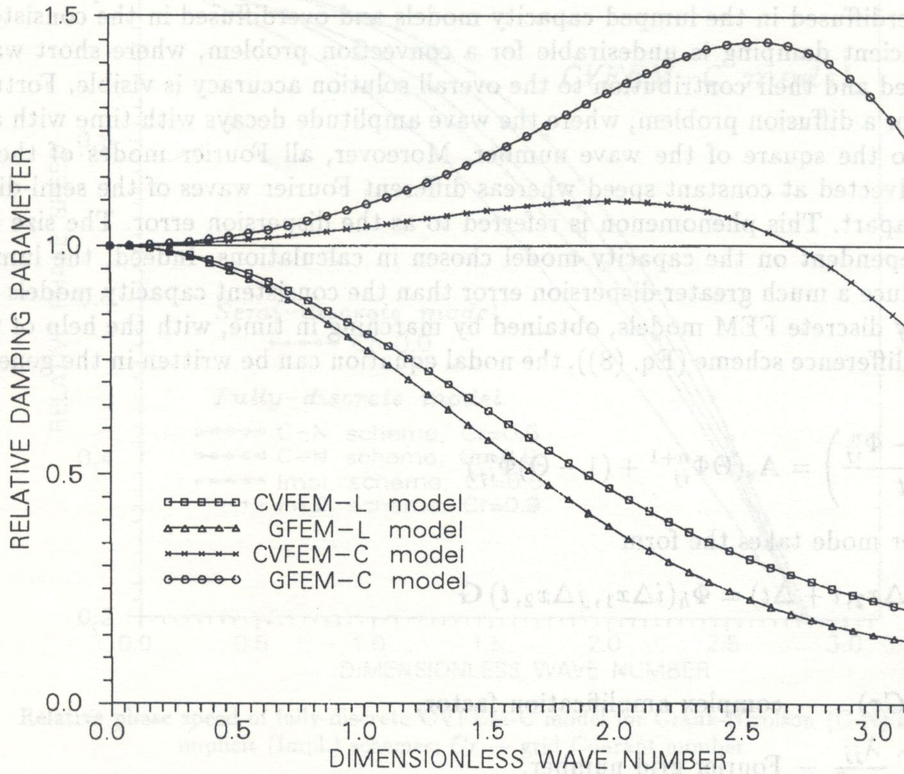

Fig. 1. Relative damping parameter for semi-discrete FEM models

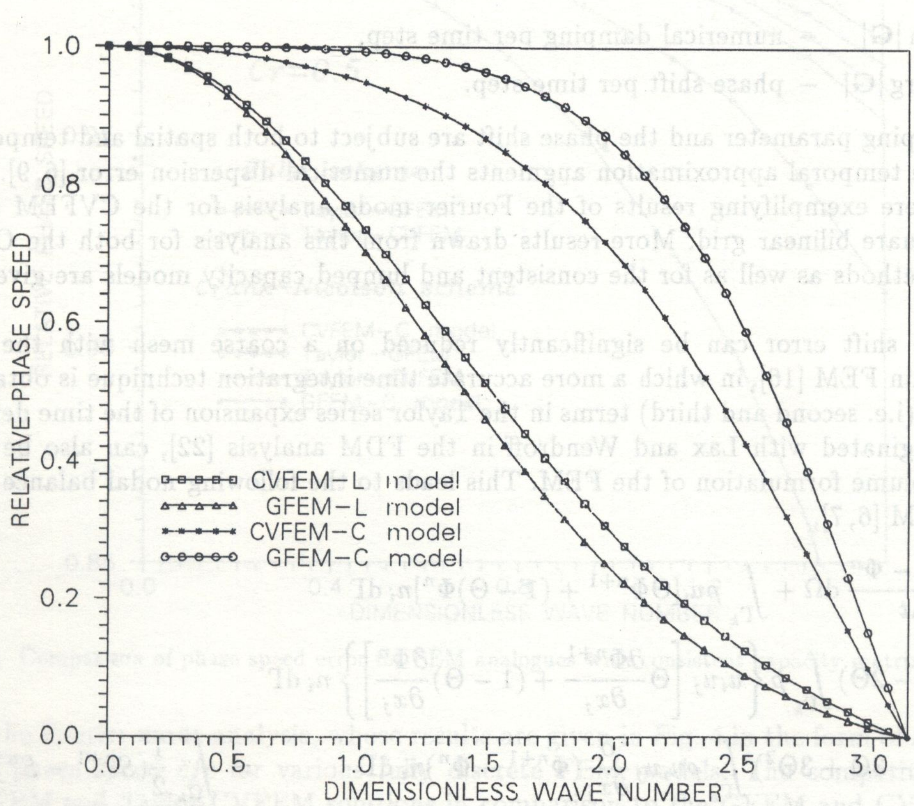

Fig. 2. Relative phase speed for semi-discrete FEM models

Their thorough analysis reveals that long waves are represented more accurately, whereas short waves are underdiffused in the lumped capacity models and overdiffused in the consistent capacity models. Insufficient damping is undesirable for a convection problem, where short waves are too slowly convected and their contribution to the overall solution accuracy is visible. Fortunately, it is not observed for a diffusion problem, where the wave amplitude decays with time with an exponent proportional to the square of the wave number. Moreover, all Fourier modes of the continuous solution are advected at constant speed whereas different Fourier waves of the semi-discrete FEM model spread apart. This phenomenon is referred to as the dispersion error. The size of this error is primarily dependent on the capacity model chosen in calculations. Indeed, the lumped models (L-models) induce a much greater dispersion error than the consistent capacity models (C-models).

For the fully discrete FEM models, obtained by marching in time, with the help of the one-step implicit finite difference scheme (Eq. (8)), the nodal equation can be written in the general operator form

$$\mathbf{A}_t\left(\frac{\Phi_{ij}^{n+1}-\Phi_{ij}^{n}}{\Delta t}\right)=\mathbf{A}_s(\Theta\Phi_{ij}^{n+1}+(1-\Theta)\Phi_{ij}^{n}) \tag{22}$$

and the Fourier mode takes the form

$$\Phi_\Delta(i\Delta x_1,j\Delta x_2,t+\Delta t)=\Phi_h(i\Delta x_1,j\Delta x_2,t)\,\mathbf{G} \tag{23}$$

where

$\mathbf{G}=f(Fo,Cr)$ – complex amplification factor,

$$Fo=\Delta t\sum_j\frac{\lambda_{jj}}{\Delta x_j^2} \quad \text{– Fourier grid number,} \tag{24}$$

$$Cr=\Delta t\sum_j\frac{\rho u_j}{\Delta x_j} \quad \text{– Courant grid number,} \tag{25}$$

$$\bar{\beta}\Delta t=-\ln|\mathbf{G}| \quad \text{– numerical damping per time step,} \tag{26}$$

$$\bar{\omega}\Delta t=-\arg|\mathbf{G}| \quad \text{– phase shift per time step.} \tag{27}$$

Here the damping parameter and the phase shift are subject to both spatial and temporal approximations. The temporal approximation augments the numerical dispersion error [6, 9]. It is visible in Fig. 3, where exemplifying results of the Fourier mode analysis for the CVFEM solution are given on a square bilinear grid. More results drawn from this analysis for both the CVFEM and the GFEM methods as well as for the consistent and lumped capacity models are given elsewhere (e.g. [9]).

The phase shift error can be significantly reduced on a coarse mesh with the use of the Taylor-Galerkin FEM [16], in which a more accurate time integration technique is obtained by applying higher (i.e. second and third) terms in the Taylor series expansion of the time derivative $\frac{\partial\Phi}{\partial t}$. This idea, originated with Lax and Wendroff in the FDM analysis [22], can also be adopted to the control-volume formulation of the FEM. This leads to the following nodal balance equation of Taylor-CVFEM [6, 7],

$$\begin{aligned}\int_{\Omega_k}\rho\frac{\Phi^{n+1}-\Phi^n}{\Delta t}\,d\Omega+\int_{\Gamma_k}\rho u_i[\Theta\Phi^{n+1}+(1-\Theta)\Phi^n]n_i\,d\Gamma\\-\frac{\Delta t}{2}(1-2\Theta)\int_{\Gamma_k}\rho\left\{u_iu_j\left[\Theta\frac{\partial\Phi^{n+1}}{\partial x_j}+(1-\Theta)\frac{\partial\Phi^n}{\partial x_j}\right]\right\}n_i\,d\Gamma\\-\frac{\Delta t}{6}(1-3\Theta+3\Theta^2)\int_{\Gamma_k}\rho u_iu_j\frac{\partial}{\partial x_j}(\Phi^{n+1}-\Phi^n)n_i\,d\Gamma=\int_{\Omega_k}\frac{1}{2}(S^{n+1}-S^n)\,d\Omega,\end{aligned} \tag{28}$$

where an additional, weighted in time, diffusion flux appears. It gives the necessary streamline-upwinding. Moreover, it stabilizes the solution and reduces its dispersion error. The latter conclusion

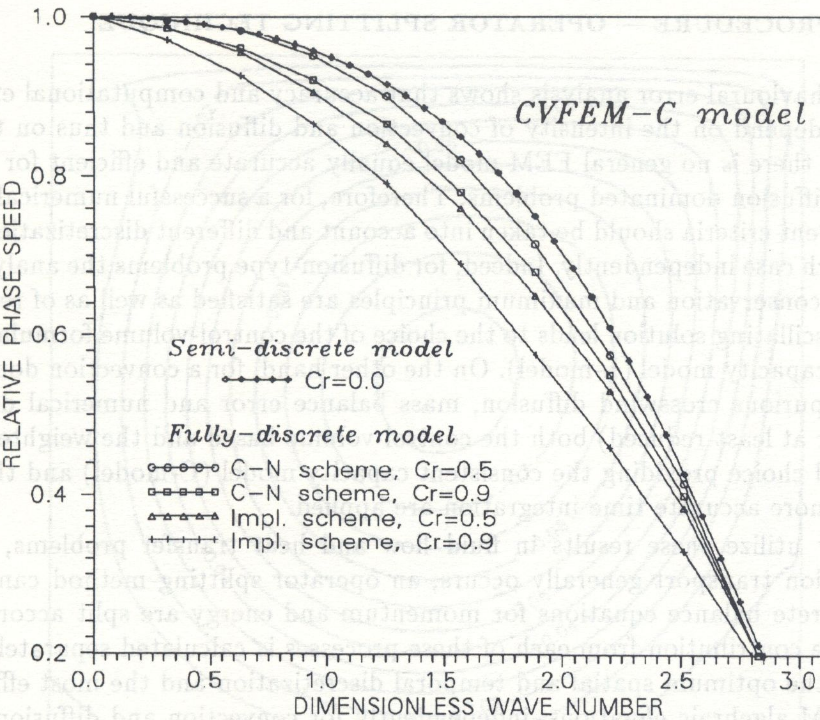

Fig. 3. Relative phase speed of fully-discrete CVFEM-C model for Crank-Nicolson (C-N) and Euler implicit (Impl.) schemes; Cr — grid Courant number

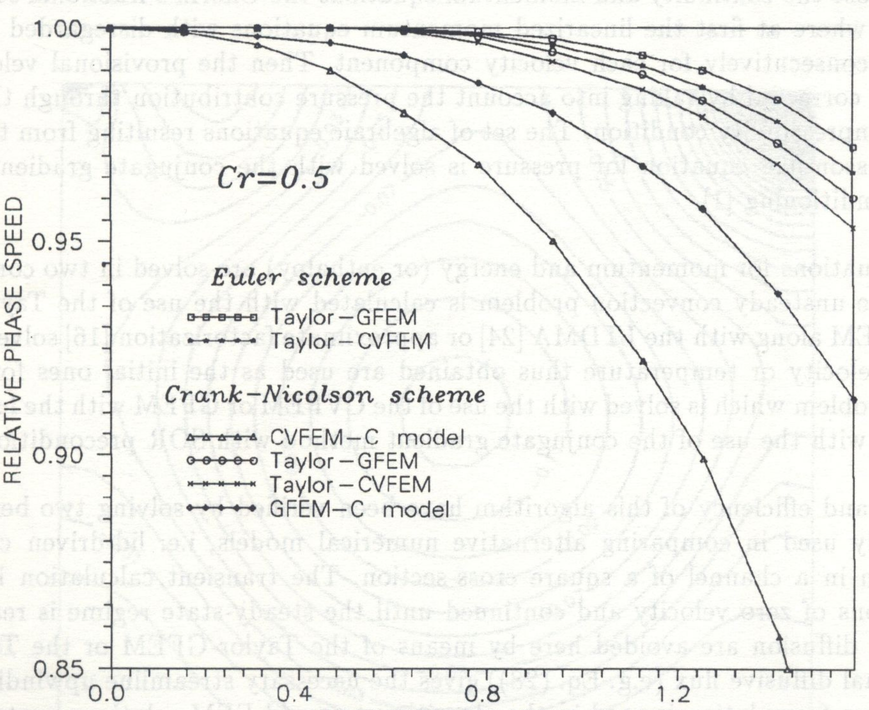

Fig. 4. Comparison of phase speed error for FEM analogues with consistent capacity matrix model

comes from the Fourier mode analysis, whose results are given in Fig. 4 in the form of a comparison of a relative phase speed $\bar{c}/c$ for various fully discrete FEM models. The competitive accuracy of Taylor-GFEM and Taylor-CVFEM solutions in comparison to the GFEM and CVFEM ones is confirmed in [6, 7], where the comparative analysis of the solutions obtained is given for a benchmark problem of pure advection of a cone in a rotating fluid.

4. SOLUTION PROCEDURE — OPERATOR SPLITTING TECHNIQUE

The complete behavioural error analysis shows that accuracy and computational efficiency of FEM models strongly depend on the intensity of convection and diffusion and thus on the type of PDE analysed. Hence, there is no general FEM model equally accurate and efficient for both convection dominated and diffusion dominated problems. Therefore, for a successful numerical simulation on a coarse grid, different criteria should be taken into account and different discretization models should be applied for each case independently. Indeed, for diffusion-type problems the analysis of conditions under which the conservation and maximum principles are satisfied as well as of requirements for a stable and non-oscillating solution leads to the choice of the control-volume formulation of the FEM and the lumped capacity model (L-model). On the other hand, for a convection dominated problem where wiggles, spurious crosswind diffusion, mass balance error and numerical dispersion should be eliminated (or at least reduced) both the control-volume based and the weighted residual based FEMs are a good choice providing the consistent capacity model (C-model) and the Lax-Wendroff method for the more accurate time integration are applied.

To practically utilize these results in fluid flow and heat transfer problems, where a couple convection-diffusion transport generally occurs, an operator splitting method can be used, where operators of discrete balance equations for momentum and energy are split according to physical processes and the contribution from each of these processes is calculated separately. This makes it possible to take the optimum spatial and temporal discretization and the most efficient solvers for a final set of FEM algebraic equations, independently for convection and diffusion. Therefore, the iterative process is constructed at each time step where:

- To decompose the continuity and momentum equations the Chorin's fractional step method [14] is applied, where at first the linearized momentum equations with disregarded pressure terms are solved consecutively for each velocity component. Then the provisional velocity field thus obtained is corrected by taking into account the pressure contribution through the enforcement of the incompressibility condition. The set of algebraic equations resulting from the FEM model for the Poisson-like equation for pressure is solved with the conjugate gradient method with SOR preconditioning [1].

- Balance equations for momentum and energy (or enthalpy) are solved in two consecutive steps. At first, the unsteady convection problem is calculated with the use of the Taylor-CVFEM or Taylor-GFEM along with the LTDMA [24] or approximate factorization [16] solvers. Next, nodal values of velocity or temperature thus obtained are used as the initial ones for the unsteady diffusion problem which is solved with the use of the CVFEM or GFEM with the lumped capacity model and with the use of the conjugate gradient method with SOR preconditioning [1].

The accuracy and efficiency of this algorithm have been verified by solving two benchmark problems commonly used in comparing alternative numerical models, i.e. lid-driven cavity flow and free convection in a channel of a square cross-section. The transient calculation is started from initial conditions of zero velocity and continued until the steady-state regime is reached. Wiggles and crosswind diffusion are avoided here by means of the Taylor-GFEM or the Taylor-CVFEM, where additional diffusive flux (e.g. Eq. (28)) gives the necessary streamline upwinding. The mixed velocity pressure formulation is used in the algorithm to avoid FEM solutions containing spurious pressure modes. The 9/4 element taken in calculations, in which the velocity is interpolated with the biquadratic Lagrange polynomial whereas the pressure field only by the bilinear one, satisfies the inf–sup condition of Brezzi and Babuska [18]. Recently, however, some authors (e.g. [25]) reported that the equal-order interpolation of the velocity and pressure fields also gives the correct FEM solutions in the time-splitting algorithm due to the strict enforcement of the continuity constraint at every stage of the iterative process [25]. Therefore, to compare both interpolation techniques in the presented algorithm, the benchmark problems have also been solved with the use of bilinear interpolation for both velocity and pressure.

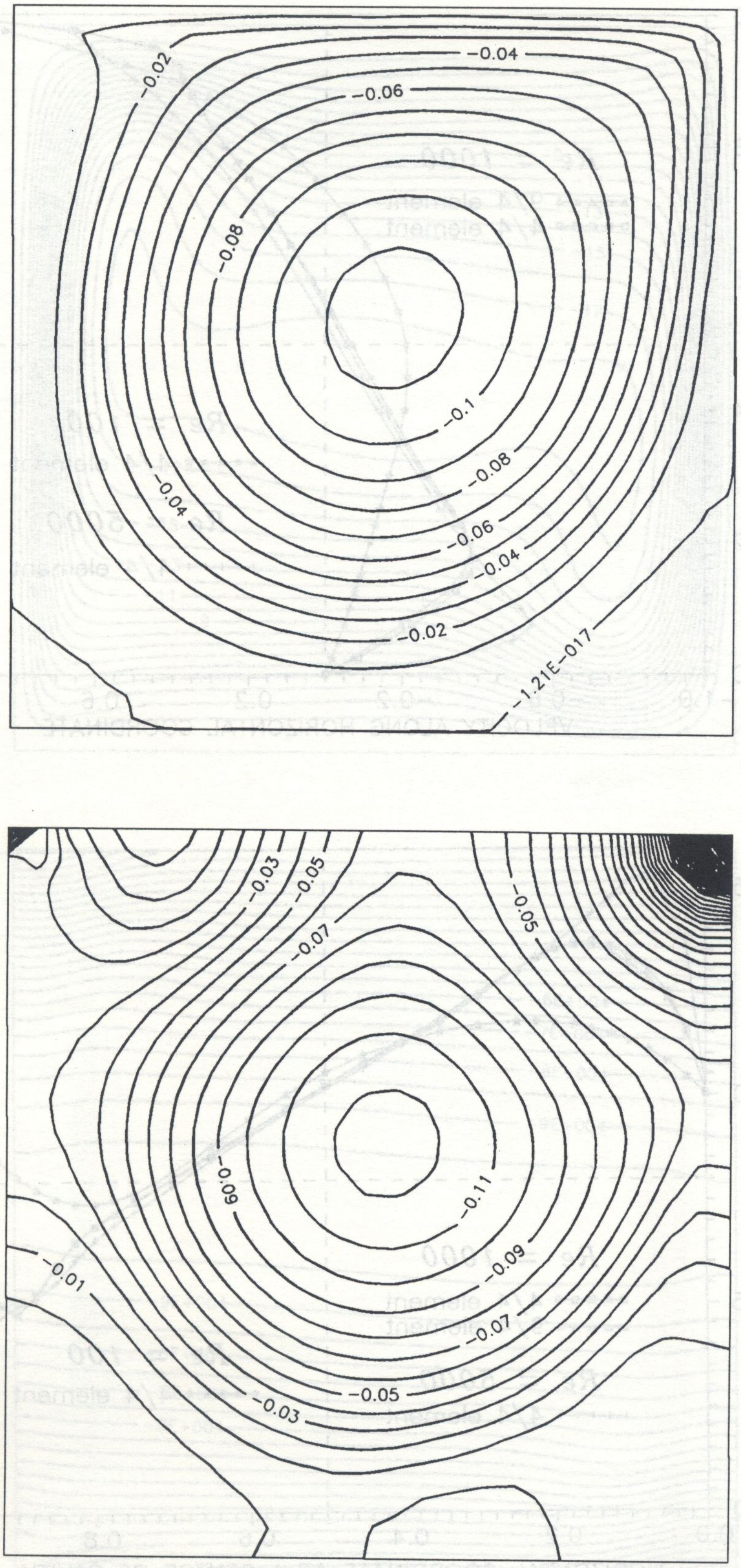

Fig. 5. Streamlines and pressure contours for driven cavity flow with $Re = 10^3$

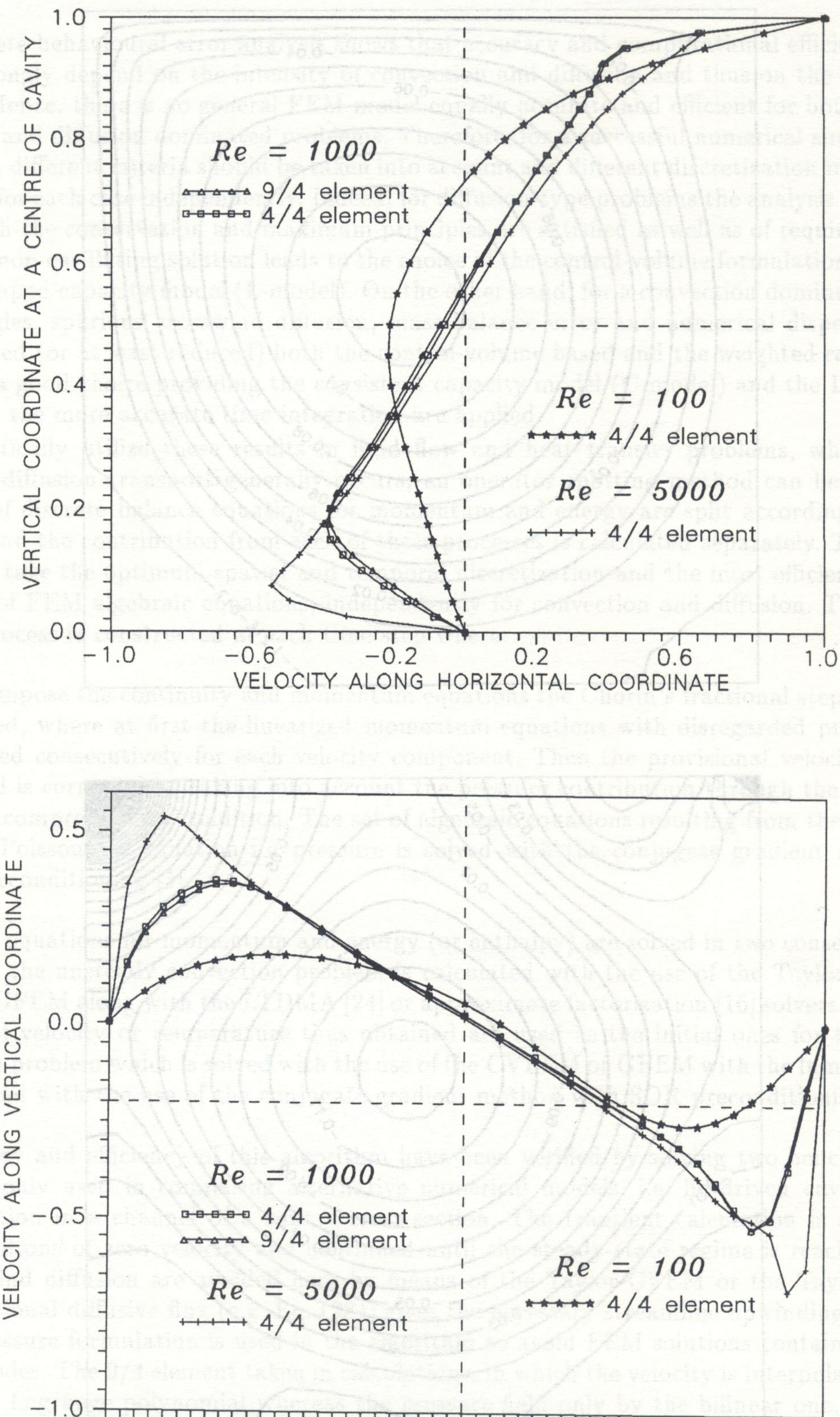

Fig. 6. Comparison of velocity components along vertical and horizontal centre lines of cavity for equal- and unequal-order interpolations

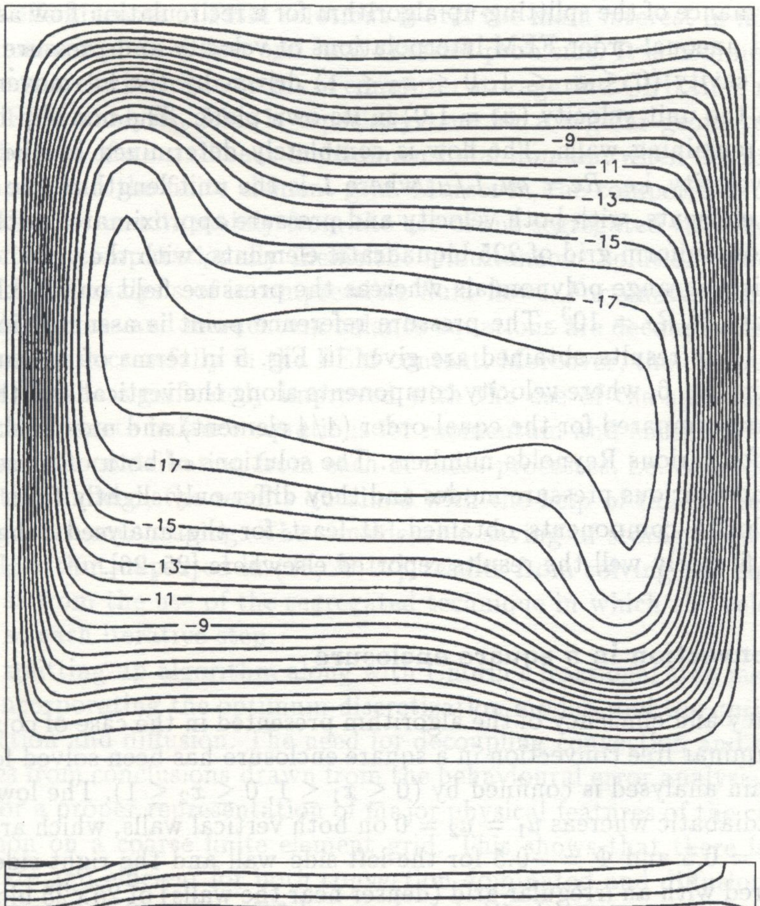

Fig. 7. Streamlines and pressure contours for free convection in a square enclosure with $Ra = 10^6$

Test 1 — Lid-driven cavity flow

To verify the performance of the splitting-up algorithm for a recirculation flow as well as to compare the equal-order and unequal-order FEM interpolations of velocity and pressure fields, the problem of flow in a closed cavity ($0 \leq x_1 \leq 1$, $0 \leq x_2 \leq 1$) driven by the lid movement is considered. The lid moves with the unit velocity ($u_1 = 1.0$) in its own plane. The non-slip boundary condition is assumed on the remaining walls. The flow is completely determined by the Reynolds number, defined for the lid velocity, i.e. $Re = \rho u_1 L/\mu$, where L is the unit length of the lid. A non-uniform grid of 900 bilinear elements, with both velocity and pressure approximated with bilinear Lagrange polynomials, or a non-uniform grid of 225 biquadratic elements, with the velocity field interpolated with the biquadratic Lagrange polynomials whereas the pressure field only with the bilinear ones, were used in the case of $Re = 10^3$. The pressure reference point is assumed in the middle of the cavity bottom wall. The results obtained are given in Fig. 5 in terms of streamlines and pressure contours as well as in Fig. 6, where velocity components along the vertical and the horizontal centre lines of the cavity are compared for the equal-order (4/4 element) and unequal-order (9/4 element) interpolations and for various Reynolds numbers. The solutions of both of these interpolations are free from wiggles and spurious pressure modes and they differ only slightly from each other (Fig. 6) in terms of the velocity components obtained, at least for the analysed range of the Reynolds number. They also fit fairly well the results reported elsewhere [25, 26].

Test 2 — Free convection in a square enclosure

To check the accuracy and efficiency of the algorithm presented in the case of coupled fluid flow and heat transfer, the laminar free convection in a square enclosure has been solved for various Rayleigh numbers. The domain analysed is confined by ($0 \leq x_1 \leq 1$, $0 \leq x_2 \leq 1$). The lower and upper walls are assumed to be adiabatic whereas $u_1 = u_2 = 0$ on both vertical walls, which are kept at a uniform temperature, i.e. $\Phi = 0.5$ and $\Phi = -0.5$ for the left-side wall and the right-side one, respectively. The domain is covered with an irregular grid (denser near the walls) of 25×25 biquadratic elements. Exemplifying results obtained are presented in Figs. 7 and 8, where steady-state distributions of streamlines, pressure contours and isothermal lines are given for the Rayleigh number $Ra = 10^6$. The computed solution is in agreement with the results published elsewhere [25, 26, 33].

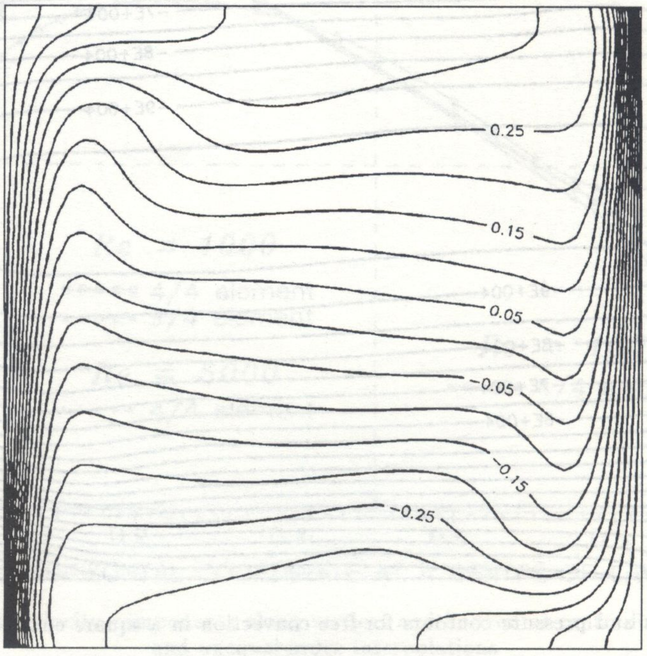

Fig. 8. Isothermal lines for free convection in a square enclosure with $Ra = 10^6$

5. Conclusions and final remarks

The question of cost effectiveness of FEM solutions is still the main interest in numerical modelling of practical, multidimensional fluid flow and heat transfer problems. It is commonly known that finite difference methods are superior in terms of computer storage and CPU time requirements when compared with the FEM analysis. This is especially true for a regular geometry domain but not necessarily true in the case of a curvilinear multidimensional domain, when the finite difference grid should be much denser than the finite element grid. Moreover, the competitive computational efficiency of FDM calculations results mainly from the use of the segregated velocity-pressure solution technique instead of the computationally inefficient simultaneous solution procedure, commonly applied in the early FEM analysis of incompressible fluid flows. Fortunately, this segregated solution technique, in which mass and momentum balance equations are decoupled in a discrete model, has recently been applied successfully in the FEM context. Moreover, the computational efficiency of FEM calculations can be significantly improved with the use of the splitting-up technique, in which operators of the discrete balance equations for momentum and energy are split according to physical processes and the contribution from each of these processes, i.e. convection and diffusion, is calculated separately. Indeed, the results obtained with the help of this technique that have recently been published are encouraging. The considerable saving of storage and execution time of the time splitting algorithm, reported in [25, 26, 27], results from solving the equations of motion sequentially as well as from the use of the segregated technique in which the velocity and pressure fields are separated at each iterative step.

In this study the splitting-up algorithm along with Chorin's fractional step method has been applied as a means of incorporating the optimum discretization models and the optimum solvers independently for convection and diffusion. The need for decoupling convection and diffusion operators in FEM models arises from conclusions drawn from the behavioural error analysis, in which the conditions are studied for a proper representation of major physical features of the convective-diffusive transport phenomenon on a coarse finite element grid. This shows that there is no general FEM model equally accurate and efficient for both convection dominated and diffusion dominated problems. Therefore, it seems that for a successful numerical simulation on a coarse grid different criteria should be taken into account and different discretization models applied for each case independently.

The accuracy and efficiency of the algorithm presented here have been verified by solving two pertinent benchmark problems. The results obtained for the lid driven cavity flow and free convection in a square enclosure show that solutions of both equal- and unequal-order FEM interpolations are free from wiggles and spurious pressure modes and they fit fairly well the results reported elsewhere.

Furthermore, it is also shown here that behavioural error analysis and Fourier mode analysis, which makes it possible to describe the behaviour of a discrete model on a coarse grid, provide a simple tool for setting up FEM models of acceptable accuracy on a coarse mesh and thus calculations are more cost-effective due to a much lesser number of nodal unknowns.

References

[1] O. Axelsson, V.A. Barker. *Finite Element Solution of Boundary Value Problems*. Academic Press, New York, 1984.

[2] B.R. Baliga, S.V. Patankar. A control-volume finite element method for two-dimensional fluid flow and heat transfer. *Numerical Heat Transfer*, **6**: 245–261, 1983.

[3] J. Banaszek. A conservative finite element method for heat conduction problems. *Int. J. Num. Meth. Engng.*, **20**: 2033–2050, 1984.

[4] J. Banaszek, B. Staniszewski. A conservative finite element method for convective diffusion problems. In: F. Pane, ed., *Integral Methods in Science and Engineering*, 435–454. Hemisphere Publishing Corporation, Washington, 1985.

[5] J. Banaszek. Comparison of control-volume and Galerkin finite element methods for diffusion-type problems. *Numerical Heat Transfer*, Part B, **16**: 59–78, 1989.

[6] J. Banaszek. Fourier mode analysis in error estimation for finite element solution of convection problems. *Archives of Thermodynamics*, **12**: 35–56, 1991.

[7] J. Banaszek. Analysis of Physical Correctness of Finite Element Models for Heat Transfer Problems. Wydawnictwa Politechniki Warszawskiej, Warszawa, 1991, (in Polish).

[8] J. Banaszek. Splitting-up technique in control-volume based FEM for incompressible fluid flow and heat transfer problems. In C.A. Brebbia, L.C. Wrobel, eds., *Advanced Computational Methods in Heat Transfer*, 637–656. Computational Mechanics Publications, Springer-Verlag, Berlin, 1992.

[9] J. Banaszek. Fourier mode analysis in error estimation for finite element solution to convective-diffusive phenomena. *Archives of Thermodynamics*, **14**: 67–92, 1993.

[10] J. Banaszek. A conservative property in finite element approximation of convective-diffusive transport phenomena. *Archives of Thermodynamics*, **14**:93–116, 1993.

[11] A.N. Brooks, T.J. Hughes. Streamline upwind Petrov-Galerkin formulation for convection dominated flows with particular emphasis on the incompressible Navier-Stokes equations. *Comp. Meth. Appl. Mech. Engng.*, **32**: 199–259, 1982.

[12] P.C. Ciarlet. Discrete maximum principle for finite difference operators. *AEQ Math.*, **4**: 338–352, 1970.

[13] C.N. Chen, L.C. Welford. Multi-level finite element solution algorithms based on multiplicative and additive correction procedures. *Int. J. Num. Meth. Engng.*, **28**: 28–41, 1989.

[14] A.J. Chorin. Numerical solution of the Navier-Stokes equations. *Math. Comput.*, **22**: 745–762, 1968.

[15] G. Comini, S. del Guidice. Finite-element solution of the incompressible Navier-Stokes equations. *Numerical Heat Transfer*, **5**: 463–478, 1982.

[16] J. Donea, S. Giuliani, H. Laval. Time-accurate solution of advection-diffusion problems by finite elements. *Comp. Meth. Appl. Mech. Engng.*, **45**: 123–145, 1984.

[17] I. Fried, D.S. Malkus. Finite element mass lumping by numerical integration with no convergence rate loss. *Int. J. Solid and Structures*, **14**, 1975.

[18] P.M. Gresho, R.L. Lee, R.L. Sani. On the time dependent solution of the incompressible Navier-Stokes equations in two and three dimensions. In: *Recent Advances in Numerical Methods in Fluids*, vol. 1. Pineridge Press, Swansea, UK, 1980.

[19] L. Hayes. Implementation of finite element alternating direction methods on nonrectangular regions. *Int. J. Num. Meth. Engng.*, **16**: 3–48, 1980.

[20] N.A. Hookey, B.R. Baliga, C. Prakash. Evaluation and enhancements of some control-volume finite element methods. *Numerical Heat Transfer*, **14**: 255–271, 1988.

[21] T. Ikeda. Maximum principle in finite element model for convection-diffusion phenomena. *Lecture Notes in Numerical and Applied Analysis*, **4**. North Holland Company, Amsterdam, 1976.

[22] P. Lax, B. Wendroff. Difference scheme for hyperbolic equations with high order accuracy. *Comm. Pure and Appl. Math.*, **17**, 1964.

[23] G.I. Marchuk. *Methods of Numerical Mathematics.* Springer-Verlag, Berlin ,1975.

[24] S.V. Patankar. *Numerical Heat Transfer and Fluid Flow.* Hemisphere Publishing Corporation, New York, 1980.

[25] B. Ramaswamy. Finite element solution for advection and natural convection flows. *Computers & Fluids*, **16**: 349–388, 1988.

[26] B. Ramaswamy. Efficient finite element method for two-dimensional fluid flow and heat transfer problems. *Numerical Heat Transfer*, Part B, **17**: 123–154, 1990.

[27] B. Ramaswamy, T.C. Jue, J.E. Akin. Semi-implicit and explicit finite element schemes for coupled fluid/thermal problems. *Int. J. Num. Meth. Engng.*, **34**: 675–692, 1992.

[28] E. Rank, C. Katz, H. Werner. On the importance of the discrete maximum principle in transient analysis using finite element methods. *Int. J. Num. Meth. Engng.*, **19**: 1771–1782, 1983.

[29] R. Richtmyer, K.W. Morton. *Difference Methods for Initial Value Problems.* Interscience Publishers, New York, 1967.

[30] S.P. Roache. *Computational Fluid Dynamics.* Hermosa Publishing, New Mexico, 1982.

[31] G.E. Schneider. Elliptic systems: Finite element method I. In W. Minkowycz, E.M. Sparrow, eds., *Handbook of Numerical Heat Transfer.* John Wiley & Sons, New York, 1988.

[32] R.J. Schnipke, J.G. Rice. A finite element method for free and forced convection heat transfer. *Int. J. Num. Meth. Engng.*, **24**: 117–128, 1987.

[33] M. Strada, J.C. Heinrich. Heat transfer rates in natural convection at high Rayleigh numbers in rectangular enclosers: A numerical study. *Numerical Heat Transfer*, **5**: 81–93, 1982.

[34] C. Taylor, T.J.R. Hughes. *Finite Element Programming of the Navier-Stokes Equations.* Pineridge Press, Swansea, 1981.

[35] J.M. Winget, T.J.R. Hughes. Solution algorithms for nonlinear transient heat conduction analysis employing element by element strategies. *Comp. Meth. Appl. Mech. Engng.* **52**: 711–815, 1985.

[36] O.C. Zienkiewicz, R.L. Taylor. *Finite Element Method*, Fourth Edition. McGraw-Hill Company, London, 1989.