Jerzy Banaszek

Institute of Heat Engineering
Warsaw University of Technology

SOME THEORETICAL AND PRACTICAL ASPECTS OF EFFICIENT MODELLING OF HEAT TRANSFER PHENOMENA ON A FINITE-ELEMENT GRID

A review of the contemporary numerical techniques, commonly used in a computer simulation of practical engineering problems, involving the fluid flow and coupled heat transfer, is given in terms of comparison of their accuracy and computational economy. Particularly, some aspects of the utilization of two different formulations of the Finite Element Method (FEM), i.e.: the one based on the weighted residual approach (PGFEM) and the one where the control--volume method is exploited (CVFEM), are discussed to find the answer to the question of how to improve the economy of the FEM calculations. In this context, the accuracy of both methods is considered through verifying a physical correctness of the solution obtained. Some simple and no-time consuming means are discussed to reduce inaccuracy resulting from a violation of the physically meaningful conservation and maximum principles as well as from the improper modelling of convection over a sparse grid. Furthermore, to speed up the FEM calculations it is shown here how to adopt the time-split and factorization techniques, which take their origin from the finite difference methods, in the Control-Volume based FEM. The encouraging preliminary results are reported here for three pertinent test problems, involving the recirculated flow and coupled heat transfer.

NOMENCLATURE

$\mathbf{C}$ – the capacity matrix
$C_{k,m}$ – component of the capacity matrix
F – integrand function
Gr – Grashoff number
f_n – given normal stresses on the domain boundary
f_τ – given share stresses on the domain boundary
h – enthalpy
h_b – boundary value of enthalpy
h^* – intermediate value of enthalpy at time $t + \Delta t$
$\mathbf{K}$ – diffusion matrix

$\mathbf{K}_u$ – convection matrix
M_p – interpolation function associated with geometrical node p
N_m – interpolation function associated with node m used to approximate the scalar quantity sought
N_g – number of geometrical nodes
N_u – number of nodes used in the velocity interpolation
NE – number of elements
NP – number of nodes used in an interpolation of a scalar Φ
n_i – components of an outward unit normal
Pe – Peclet number
P_l – velocity interpolation function associated with the node l
Pr – Prandl number
p – pressure
Q_k – pressure interpolation function associated with the node k
$\boldsymbol{R}$ – right-hand side vector of the matrix equation
Ra – Raileigh number
Re – Reynolds number
$REZ(...)$ – residuum (error) of a partial differential equation
S – rate of a volumetric source
S_i – rate of a volumetric force in the momentum equation in i-direction
S_h – rate of a volumetric source in the energy equation
t – time
u_{i_*} – component of the velocity vector
$u_i, \hat{u}_i$ – components of the intermediate velocity vectors
$\bar{x}$ – global Cartesian coordinate system
x_i – component of Cartesian coordinate system
W_k – weighing function associated with the node k
Γ – boundary surface of the domain Ω
Γ_e – boundary surface of the element domain Ω_e
Γ_k – boundary surface of the control volume Ω_k
Γ_u, Γ_τ – external surface of domain Ω with given velocity or boundary stresses, respectively
Γ_h, Γ_q – external surface of Ω with the given enthalpy (temperature) or heat flux, respectively
$\lambda_{i,j}$ – component of the conductivity tensor to the specific heat
μ – dynamic viscosity
Δt – time step
θ – parameter of the one-step implicit finite difference time marching scheme
$\bar{\zeta}$ – local curvilinear coordinate system
ζ_i – component of the local curvilinear coordinate system
ρ – product of density and specific heat

Φ – scalar quantity sought
Ω – volume of geometrical domain
Ω_e – finite element volume
Ω_k – control volume associated with the node k
$\Omega_{k,e}$ – element fraction of the control volume Ω_k

Subscripts

e – pertaining to the element e
i, j – coordinate directions
k, l, m – associated with the node k, l or m, respectively

Superscripts

e – pertaining to the element e
n – refers to time $t = n\Delta t$
0 – refers to initial values of u_i, p and h

Abbreviations

C – Consistent Capacity Matrix Model
CVFDM – Control-Volume (based) Finite Difference Method
CVFEM – Control-Volume based FEM
CVFEM-TM – CVFEM with high order approximation of the temporal term of the balance equation
FDM – Finite Difference Method
FE – Finite Element
FEM – Finite Element Method
GFEM – FEM based on the Galerkin Method
GFEM-D – GFEM model based on the conservative form of the partial differential equation
GFEM-T – Taylor-Galerkin FEM
L – Lumped Capacity Matrix Model
LTDMA – Line-Tridiagonal Solver
PDE – Partial Differential Equation
PGFEM – FEM based on the Petrov-Galerkin Method
SOR – Successive Over-Relaxation iteration technique

INTRODUCTION

There is no doubt that the most reliable information about physical phenomenon can be obtained by direct observation and measuring the reality itself. Full

scale experimental investigations are, however, in most cases prohibitively expensive and often impossible. Therefore, they are replaced by experiments on small scale models. Unfortunately, they cannot always simulate all important features of the subject in its full scale and there are not general rules of extrapolation of the resulting information to the full scale. Moreover, experiments are often cumbersome and time-consuming. It is, therefore, a need for searching a new way of quantitative analysis of the problem considered.

Owing to a tremendous progress in both the computer technology and the computer simulation techniques, a numerical analysis of physical phenomena and practical engineering problems, which is based on a discrete representation of a classical continuous model, has been becoming more and more popular among scientists and designers for more than last thirty years.

Indeed, a numerical simulation of physical phenomena is often a reasonable alternative and always a desirable supplement to the experimental study. Low cost and remarkable speed in achieving comprehensive information about the subject analyzed are commonly appreciated advantages of numerical calculations. In most applications the cost of computer run is many orders lower in its magnitude than the cost of corresponding experiments. Moreover, numerical analysis provides, with a remarkable speed, an answer to the question of how an individual parameter influences behaviour of the system considered or to the question of what are optimal data for a design.

Despite these merits, a proper numerical model can be defined only if adequate physical and mathematical models of the phenomenon exist. It is seen, therefore, that in a comprehensive analysis (or in a design program) an effective balance in a use of both an experimental approach and a discrete simulation must be struck. The experimental analysis server to verify the physical model and its mathematical counterpart utilized in the investigation, whereas numerical methods serve to provide a quick and cheap tool for examining the influence of various parameters on behaviour of the system or design considered.

This paper deals explicitly with contemporary numerical techniques used to simulate fluid flow and heat transfer problems on a computer and its main purpose is to give some comments on the utilization of the Finite Element Methods to the convective-diffusive transport phenomenon.

1. GENERAL COMPARISON OF CONTEMPORARY NUMERICAL METHODS

Concise review of contemporary numerical techniques, commonly used in computer simulation of practical engineering problems, involving the heat transfer and fluid analysis, is given in Fig. 1.

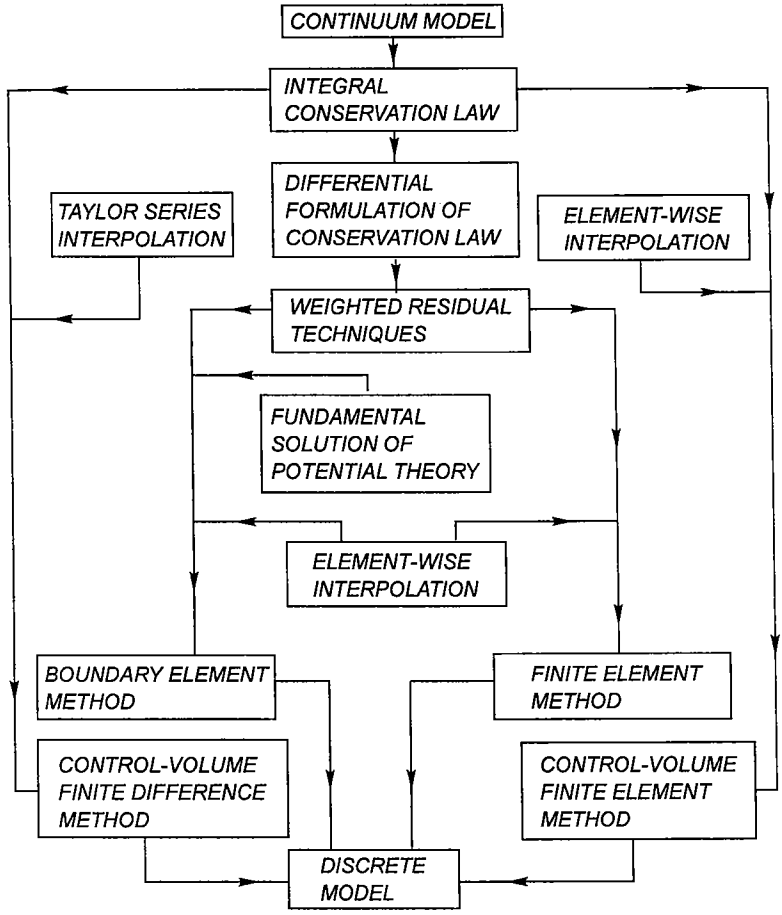

Fig. 1. Contemporary numerical methods based on integral formulation of heat transfer problems

They differ from each other in both the local interpolation technique used to approximate the unknown field quantity and the way in which the final set of algebraic equations of the discrete model is established.

When the truncated Taylor series expansion is applied to approximate derivatives of an unknown between two adjacent grid points one obtains the finite difference interpolation, typical for any finite difference analogue (FDM) of the continuous model. On the other hand, a more general approximation of the field quantity can be established on the basis of an interpolation with the local subdomain support [1,2]. The latter discretization procedure, inherent in any Finite Element Method (FEM) is further called the element-wise interpolation and described in the subsequent chapter.

When the local form of the conservation principle, given by partial differential equations, is taken as the starting point of the solution procedure, the final set of discrete model equations can be obtained by means of sophisti-

cated mathematical techniques, indispensable to formulate an integral equation of the problem. Namely, the classical FEM is based on the Petrov-Galerkin weighted residual approach (PGFEM) [1,3], whereas the integral equations of the Boundary Element Method (BEM) are established by exploiting the Galerkin weighted residual technique and utilizing the fundamental solutions of the potential theory [4,5]. The success of both these methods in solid mechanics provided an initial impetus for their use in the field theory problems, governed by equation of mass, momentum and energy transfer. Soon it has been found, however, that to avoid numerical instability and a violation of fundamental physical constraints of the transport phenomena, special techniques are needed for proper discrete modelling of convection. Moreover, BEM, which seems to be very attractive due to a reduction of problem dimensionality, is still in its infancy and its applicability to fluid flow problems is very limited. On the other hand, although PGFEM is well established in this case, much should be done to improve its poor computational economy.

On the other hand, a direct imposition of the physical conservation principle over small but finite control volumes, defined beforehand around nodal points distributed in the region considered, leads to the so-called Control Volume formulation, which is simple, clear and convincing for those who are not deeply involved in mathematics. This approach permits a direct physical interpretation of the final algebraic equations of a computational model through an enforcement of the fundamental physical law of conservation of mass, momentum and energy. In consequence, it assures an accurate accounting of flows of these entities through each control volume and through the whole domain, irrespective of the pattern and density of the geometrical and time discretization applied.

The Control Volume Finite Difference Method (CVFDM), based on this approach and on the finite difference approximation, is well established [6,7] and it has widely been used in the Computational Fluid Dynamics for more than twenty years. The main reasons of that lie in:

- simple and physically justified way of founding CVFDM equations;
- simple and economic algorithm of the method in general curvilinear orthogonal coordinate system, provided that its metrics are given analytically;
- regular and sparse structure of the global matrix for which the sophisticated and very quick solvers of a set of algebraic eqs have been elaborated;
- availability of several very economic CVFDM software codes.

There are, however, two serious drawbacks of the method. Namely, the poor accuracy in a coarse grid, due to the assumption of a constant gradient of the field quantity along each segment of a control-volume boundary, and difficulties in a good approximation of irregular non-orthogonal domains. Both of them lead to need for a use of much denser grid than the one possible in the FEM discretization model.

While a more general element-wise interpolation procedure is used in local integral balances one obtains the discrete model – further called Control-Volume Finite Element Method (CVFEM in abbreviation) [8,9,10,11]. This – a quite new – numerical technique, which can be viewed as a generalization of the CVFDM [10], seems to be very attractive as it shares advantages of the physically justified control-volume formulation for the conservation principle and of the FEM discretization procedure.

Nowadays the most efficient and flexible numerical simulation of nonlinear fluid flow and heat transfer in geometrically complex domains is based on the CVFDM or the FEM technique. However, the answer to the question which of them is superior is rather difficult, as their performance depends on many features of the problem considered. Nevertheless, some general conclusions, drawn from the author's analysis as well as taken from the literature on the subject, are collected in Table 1.

Table 1

General Comparison of Finite Difference and Finite Element Models for Fluid Flow and Heat Transfer Problems

METHOD / ATTRIBUTE	Control-Volume Finite Difference Method (CVFDM)	Petrov-Galerkin (Galerkin) Finite Element Method (PGFEM or GFEM)	Control-Volume Finite Element Method (CVFEM)
Ease of coding	very easy	more complicated	more complicated
Flexibility	good	much better	much better
Accuracy per unknown	fair	better	better
Computational Efficiency	good	good	good
Mains strengths	physical background ease of coding economy	mathematical background close approximation of curvilinear geometry flexibility	physical background close approximation of curvilinear geometry flexibility
Main weaknesses	poor approximation of curvilinear domain boundaries extension to higher order interpolation cumbersome modelling of boundary conditions	economy violation of local conservation property	economy

Ease of coding is defined here as a programming effort and algebraic manipulations required to make the problem ready for calculation. It can be approximately quantified by using Paterson's formula:

$$E = 5.2(L/1000)^{0.91}$$

$$D = 49L^{1.01}$$

where: L – number of source code lines
E – total effort in programmer months
D – number of pages of documentation

Flexibility means an inverse measure of a number of changes needed to adopt a written computer program to a new problem, whereas Computational Efficiency (CE) can be estimated by:

$$\mathrm{CE} = \frac{k}{\varepsilon \tau}$$

where: ε – the error solution in some appropriate norm
τ – the CPU time or operation count
k – proportionality constant

The economy of a computer simulation is measured by the computational time and the memory size required.

The above comparison shows that the most encouraging advantage of the FEM analysis of practical fluid flow and heat transfer problems is its simplicity in approximation of curvilinear geometry on a coarse grid. This feature cannot be overestimated if one takes into account the fact that a solution of most field theory problems is very sensitive to even minor changes of the domain boundaries.

On the other hand, it is also visible from Table 1 that the main draw-back of FEMs is their poor economy in comparison with the one inherent in the CVFDM. This conclusion is reported in many papers but only for a regular rectangular domain, which can be exactly reflected in a discrete model on a sparse difference grid. In this case there is no reason, except for a comparative one, to use the FEM discretization. Moreover, this conclusion is not necessarily valid in the case of complex geometry, where the *stair-case-like* finite difference approximation of curved boundaries needs much denser grid than the FEM approximation does.

Nevertheless, the poor computational economy still remains a major impediment in a widespread use of FEM in modelling of practical fluid flow and heat transfer problems. Therefore, its improvement is a crucial task in theoretical and practical development of the method nowadays.

It seems that there are two possible ways to change the FEM computational economy for the better. The first one lies in searching for such modifications of the classical FEM formulations which provide a required accuracy of the approximate solution on coarser space-time discretization grids. Thus, a computational time and computer storage requirements can be significantly reduced. The second way consists in a use of sophisticated acceleration techniques for the solving process, like the time-split and the approximate factorization algorithms. They take their origin from the FDM analysis and the work is still in progress to utilize them in the way which enables to retain the versatility of the FE discretization procedure.

In this context, the available means for the improvement of the FEM computational economy are further discussed in the paper.

2. FEM EQUATIONS FOR CONVECTIVE-DIFFUSIVE TRANSPORT PROBLEM

The physical conservation principle for a scalar field quantity Φ, 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:

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

which states that a time rate of change of the quantity Φ can take place due to both a convective-diffusive flux through the boundary Γ_k and sources occurring within the domain Ω_k.

For an infinitesimally small control-volume the balance eq. (1) assumes a local differential form:

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

In the above equation and all others in the paper Einstein summation convection is used.

The integral eq. (1) gives a basis for setting up the discrete nodal equations of the Control-Volume FEM, whereas eq. (2) is used in the weighted residual Petrov-Galerkin FEM. In the latter case the integral form of the governing equation is obtained by weighing the error of eq. (2), caused by the approximate solution, through the assumed weighing functions W_i within the whole domain Ω [1,3], i.e.:

$$\int_{\Omega} W_k REZ(eq.(2)) d\Omega = 0 \quad \text{for } k = 1,2,...,NP \tag{3}$$

Eq. (1) and eq. (3) are the model integral equations for convective-diffusive transport phenomenon. They express, in two different ways from mathematical point of view, the conservation of such quantities like mass, momentum components or enthalpy. This means that when the fluid flow and coupled heat transfer problems are calculated, they are governed by the set of eqs (1) or eqs (3), each for different variables, i.e. for velocity components, for pressure or for temperature. Nevertheless, the form of any equation in both these sets is similar to the form of eq. (1) or eq. (3). Therefore, a proper discrete approximation of the single convective-diffusive transport eq. (1) or eq. (2) is a crucial step in an accurate and efficient numerical analysis of the whole problem, governed by the continuity, Navier-Stokes and energy equations.

To obtain the discrete form of these equations two subsequent discretization steps should be performed, i.e.: the spatial discretization and then the integration over time. The former is based on, the above mentioned, element--wise interpolation procedure, whereas the latter can be done by using any of, the commonly utilized in finite differences, one- or two-step marching-in-time schemes [12].

The spatial finite-element approximation is based on a division of the domain considered into a set of the subdomains called finite elements. Over each of them an independent interpolation of the field quantity sought is assumed in terms of a sum of products of its nodal values and relevant interpolation polynomials [1], i.e.:

$$\Phi(\bar{\zeta},t) = N_m(\bar{\zeta})\Phi_m(t) \quad \text{for } m = 1,2,...,N_\Phi \tag{4}$$

$$u_i(\bar{\zeta},t) = P_l(\bar{\zeta})u_{i,l}(t) \quad \text{for } i = 1,2,3; \ \ l = 1,2,...,N_u \tag{5}$$

$$p(\bar{\zeta},t) = Q_k(\bar{\zeta})p_k(t) \quad \text{for } k = 1,2,...,N_p \tag{6}$$

for temperature, velocity components and pressure, respectively.

The shape of the element in the Cartesian coordinates, $\bar{x} = (x_1, x_2, x_3)$, is given by the one-to-one correspondence (Fig. 2):

$$x_i(\bar{\zeta}) = M_p(\bar{\zeta})x_{i,p} \quad \text{for } i = 1,2, \text{ or } 3, \ p = 1,2,...,N_g \tag{7}$$

between the triplets of these coordinates and the local curvilinear coordinates $\bar{\zeta} = (\zeta_1, \zeta_2, \zeta_3)$.

Weighing and interpolation functions are usually defined in the way which ensures the C^0-interelement continuity and thus each global integral (over the whole domain Ω) can be calculated as a sum of the corresponding element shares (integrals over the element domain Ω_e) [1]:

$$\int_{\Omega} F(\bar{x})d\Omega = \sum_{e} \int_{\Omega_e} F^e(\bar{x})d\Omega \tag{8}$$

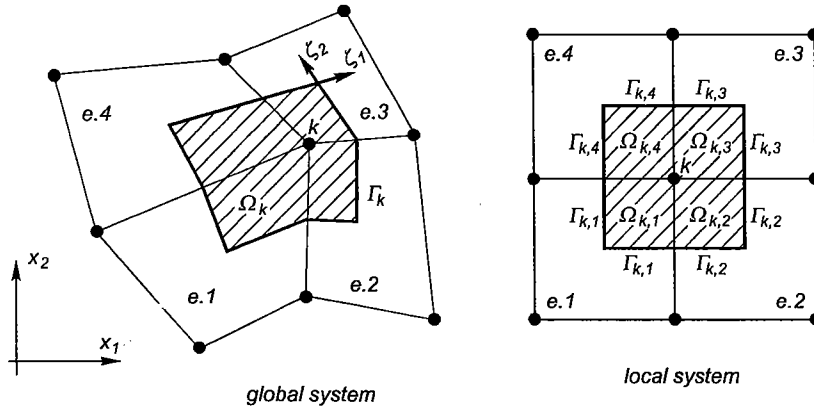

Fig. 2. Control-Volume division pattern for the bilinear element

This discretization procedure possesses some appealing features. First of all, an independent approximation of geometry and each field quantity can be used by selecting different numbers of nodal points and different form of the relevant interpolation functions for coordinates, velocity, pressure and temperature. Moreover, the possibility of using high-order polynomials M_m enables the local boundary fitting of the element side to a small segment of the real boundary of the domain Ω. Thus, external curvilinear surfaces can be satisfactorily approximated on a coarse grid. Finally, the procedure provides a tool for a use of non-structural gird, i.e. the grid with irregular shapes of subdomains and with a different density in the different parts of the domain.

On putting the element-wise interpolations, given by eq. (4) to eq. (8), into eq. (2), weighing thus obtained *residuum*, through the functions W_k, in the whole domain Ω (eq. (3)) and taking into account the boundary conditions (described in details elsewhere [1,2,3]), one obtains a set of ordinary differential equations in the following matrix form:

$$\mathbf{C}\frac{d\Phi}{dt} + \left(\mathbf{K} + \mathbf{K}_u\right)\Phi = \mathbf{R} \tag{9}$$

where: $\mathbf{C}$, $\mathbf{K}$, $\mathbf{K}_u$ and $\mathbf{R}$ denote, respectively, the capacity matrix, the diffusion and convection matrices and the right-hand side vector resulting from boundary conditions and internal sources, for the finite element method based on the Petrov-Galerkin technique [1,2,3]. The Galerkin FEM (GFEM), commonly exploited in the diffusive-type problems [1], is a particular case of the PGFEM where each W_k coincides with the appropriate interpolation function N_k.

On the other hand, if the FE discretization algorithm (eq. (4) to eq. (8)) is used to calculate the integrals in the balance eq. (1) over the control volume Ω_k, defined beforehand for each nodal point k as a sum of the volume fractions $\Omega_{k,e}$ of all elements which share the given node (Fig. 2), the nodal equation for the Control-Volume based FEM (CVFEM) is obtained [8,9,10,11]. The set of all these equations takes the matrix form given by eq. (9).

The way chosen to spatially approximate the time derivative $\partial\Phi/\partial t$ in eq. (1) or eq. (2) determines a form of the capacity matrix $\mathbf{C}$ and significantly influences the accuracy and stability of the FE solution to transient convection-diffusion problems [1,10,11]. When the spatial approximation of $\partial\Phi/\partial t$ is assumed in accordance with eq. (4):

$$\frac{\partial \Phi^e}{\partial t}(\bar{\zeta}, t) = N_i(\bar{\zeta}) \frac{d\Phi_i}{dt}(t)$$

the so-called Consistent Capacity Matrix model (C-model) is obtained where elements of the C-matrix are given as:

$$C_{k,m} = \sum_{e=1}^{NE} \int_{\Omega_e} \rho N_m \, d\Omega \quad \text{or} \quad C_{k,m} = \sum_{e=1}^{NE} \int_{\Omega_e} \rho W_k N_m \, d\Omega \qquad \text{for } k, m = 1, 2, ..., NP \tag{10}$$

for the CVFEM or PGFEM, respectively.

On the other hand, neglecting the spatial changes of $\partial\Phi/\partial t$ over a control--volume Ω_k or over an element leads to the diagonal capacity matrix of the following form:

$$C_{k,k} = \sum_{e=1}^{NE} \int_{\Omega_{k,e}} \rho \, d\Omega \quad \text{or} \quad C_{k,k} = \sum_{e=1}^{NE} \int_{\Omega_{k,e}} \rho W_k \, d\Omega \qquad \text{and } C_{k,m} = 0 \quad \text{for } k \neq m \tag{11}$$

for the CVFEM and PGFEM, respectively. The model is further labelled as L-model (Lumped Capacity Matrix).

To integrate eq. (9) in time, the one-step implicit finite difference scheme:

$$\Phi_k^{n+1} = \Phi_k^n + \theta \Delta t \left(\frac{d\Phi}{dt}\right)^{n+1} + (1-\theta)\Delta t \left(\frac{d\Phi_k}{dt}\right)^n \tag{12}$$

where: $0 \leq \theta \leq 1$ is applied and thus the final set of algebraic equations of the FEM model takes a matrix form:

$$\left(\mathbf{C} + \theta \Delta t (\mathbf{K} + \mathbf{K}_u)\right) \Phi^{n+1} = \\ = \left(\mathbf{C} - (1-\theta)\Delta t (\mathbf{K} + \mathbf{K}_u)\right) \Phi^n + \theta \Delta t \mathbf{R}^{n+1} + (1-\theta)\Delta t \mathbf{R}^n \tag{13}$$

3. BEHAVIOURAL ERROR ANALYSIS FOR THE FEM MODELS

3.1. Preliminary comments

The commonly used error analysis for a discrete model gives an estimation of the truncation error, referring to inaccuracy caused by ignoring high order terms in an infinite Taylor series expansion of the unknown. This error, which is due to finite sizes of steps of both spatial and temporal discretizations, vanishes when the grid increments Δx_i and Δt tend to zero. This is, however, a dubious consolation as the complex mathematical description of fluid flow and heat transfer problems, given by a set of coupled nonlinear partial differential equations, usually forces a use of a grid with a finite density, due to still limited power and efficiency of contemporary computers. Moreover, to retain the computational economy of a numerical analysis one should deal with rather moderate grid densities.

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 behavioral error analysis, developed by R.P.Roache [13] in the FDM context, is used here. It 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 the FE grid [14,15]. Really, for engineers and physicists such appraisal seems to be the most convincing way for verifying a correctness of a numerical model employed.

3.2. Conservation property

The *conservation property*, which directly arises 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, it means that the field quantity sought if correctly balanced within the domain considered and/or within its subdomains, irrespective of the discretization pattern and the grid density used [13,15,16].

It is shown above that the FEM nodal equations can be established by two different ways, i.e.: by using 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 the reason to distinguish two different forms of the conservation property for the FE analogues, namely: the global and the local ones [15,16].

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

The weighted residual technique consists in a global minimalization of the PDE residuum within the whole domain Ω (eq. (3)). Therefore, its individual nodal equation cannot, in general, be considered as the local balance equation. Therefore, only the global form of the conservation property can be associated with this model [15,16]. The sufficient condition for the GCP requires that a sum of all weighing functions W_k should be equal to unity at any point of the domain [16].

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.

The LCP, although not indispensable, is desirable as it frequently offers a better accuracy and less stringent stability requirements of the FEM solution in comparison with the ones obtained from the only globally conservative weighted residual FEM models. This is shown in [10,11,16,17] where the accuracy of the GFEM and CVFEM solutions is compared for the selected steady-state and transient diffusion test problems.

3.3 Mass balance error

In the FEM analysis of a convection-dominated problem a correctness of both mass and convected scalar quantity balances should be simultaneously taken into account [16]. A poor interpolation of a solenoidal velocity field over a coarse low-order FE grid can cause the mass balance error which might significantly influence the accuracy of the FEM solution for the scalar quantity sought. To reduce this inaccuracy, one can establish the PGFEM (GFEM) nodal equations on the basis of non-conservative form of eq. (2) [15,16]. 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 [15,16]. The FEM models thus obtained are likely not to satisfy the GCP but they provide more accurate results for the problems tested ([16]). This is because the impact of inaccuracy in an approximation of the continuity condition (which is in force for an incompressible fluid) over the FE grid on the calculated field quantity is significantly reduced [15,16]. In consequence, an acceptable accuracy of the FEM solution can be often achieved on a coarser mesh.

To show this, a practical problem of the free convection in a channel of a square cross-section has been solved [16]. Two different weighted residual FEM models, based on the Galerkin method, are compared on the nine-node FE grid, where velocity components and temperature are interpolated by the biquadratic polynomials whereas pressure is interpolated by the bilinear polynomial within each element domain ([18]). The first model (further named with the abbreviation GFEM-D) is based on *conservative* form of the PDE for the energy balance (eq. (2) where Φ stands for temperature). The second one (GFEM) is developed using the *non-conservative* form of eq. (2), which through the continuity condition is differentially equivalent to the previous form.

Geometry and boundary condition for velocity components and for temperature are given in Fig. 3. The mixed FEM model [18] is used along with the technique of simultaneous solution of a coupled set of the continuity, momentum and energy equations. It means that the intermediate mass balance correction at each iteration step, inherent in the SIMPLE-like algorithms commonly used in the FDM calculation (e.g. [6]), is not applied here.

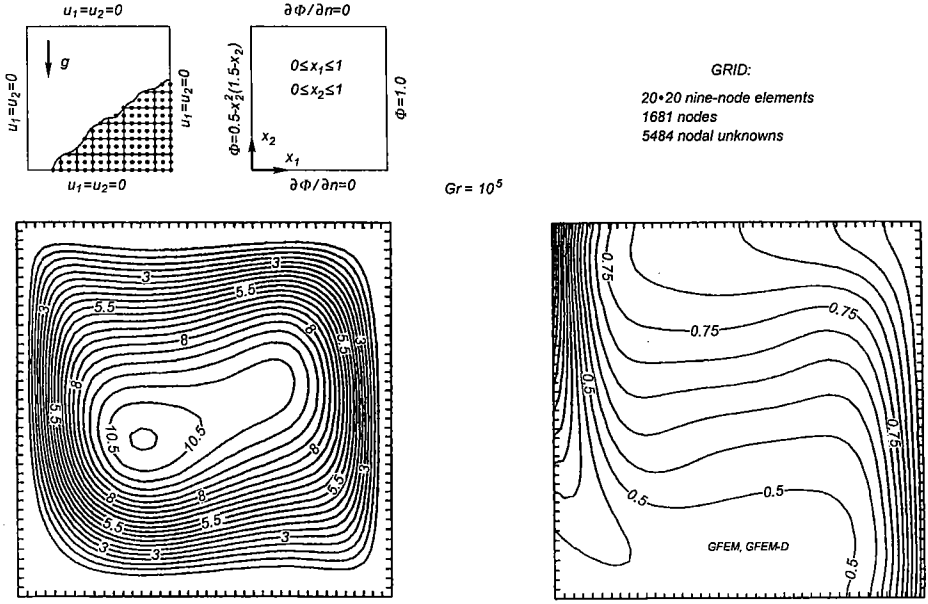

Fig. 3. Free convection in a square channel – geometry, velocity and temperature boundary conditions, stream- and isothermal-lines obtained

The results obtained for the Grashoff number $Gr = 10^5$ in the dense grid, consisted of 20·20 nine-node elements, are given in Fig. 3 in terms of both stream- and isothermal-lines. The GFEM and GFEM-D solutions are here graphically indistinguishable. In the 2×2 time coarser grid, however, the GFEM-D solution exhibits incorrect run of the isothermal lines (Fig. 4). Fotru-

nately, this is not observed for the GFEM solution (Fig. 4) which is close to the one obtained in the dense grid (Fig. 3). In the GFEM and GFEM-D calculations the same discretization pattern, the same velocity, pressure and temperature interpolations and finally the same FEM procedure for the continuity and momentum equations have been applied. The only difference between these two models lies in the form of the energy PDE used to obtain its weighted residual FEM analogue. Therefore, one can conclude that the different accuracy of each of them results mainly from the impact which the mass balance error has on the accuracy of the calculated temperature.

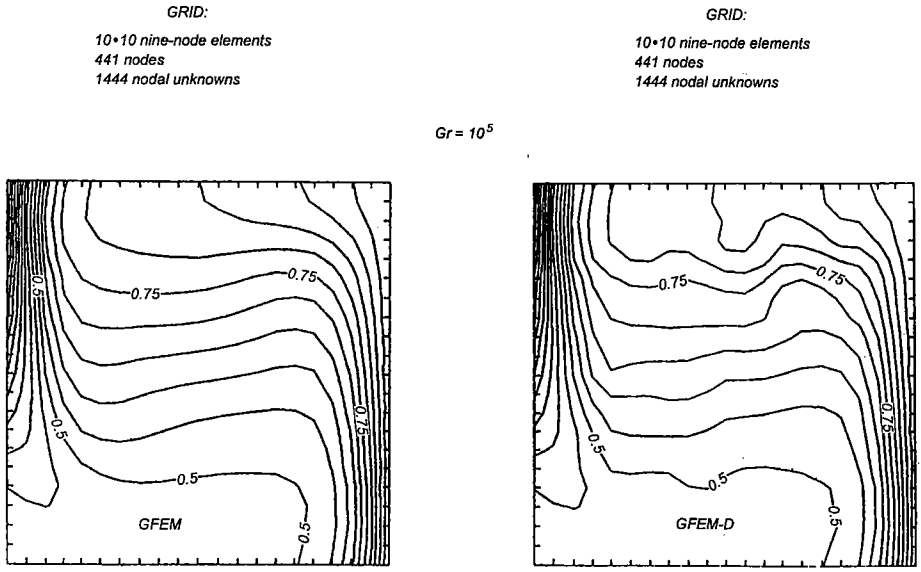

Fig. 4. Comparison of GFEM and GFEM-D solutions for temperature in the free convection problem calculated on the coarse grid

Good agreement of the GFEM solution on the coarse grid with the one obtained on four times denser grid (Fig. 3 and Fig. 4) enables to accept the former disctertization as a sufficient one for the FEM analogue in which the inaccuracy of the velocity interpolation does not significantly influence the temperature obtained. In consequence, the number of nodal unknowns can be diminished 3.8 times (from 5484 to 1444) and thus considerable savings of both the computer storage and the computational time are achieved.

3.4. Discrete maximum principle

One of the most important properties of the convective-diffusive transport phenomenon is the *maximum principle* which provides the physical limits on extremum values of the unknown. For instance, when the heat conduction

with no heat source is considered, then the principle states that both the maxi mum and the minimum values of temperature can only occur on the domain boundary or at an initial time.

If a numerical model does not preserve this principle within a division grid (i.e. the discrete form of the maximum principle [19]), its solution may exhibit strong spatial and temporal oscillations and it may even take values which are outside the physically justified range [11,15,17]. To avoid this error in the FEM analysis one should select the space-time discretization pattern in the way which ensures the fulfillment of Ciarlet matrix criterion [19], which imposes inequality relations for individual terms of the capacity and diffusion matrices [17]. They provide restrictions on the grid Fourier number, on a ratio of the spatial division steps and on the θ parameter of the time marching scheme (given by eq. (12)). 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 the lower and the upper limits [17,20].

3.5. Upwind technique in FEM models

Convection is a *one-way* transport phenomenon, i.e. the convected scalar quantity is travelling only along 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 provide considerable spatial oscillations of a numerical solution, called *wiggles* [3,11,14], particularly in the case where a strong gradient of the transported variable occurs along the streamlines.

To give an image of a size of the error, the GFEM solution is presented in Fig. 5 for the energy equation of the forced convection developing flow in the channel of a square cross-section. The problem has been solved [14] using the mixed FEM model ([18]) on the biquadratic nine-node element grid ([1]) with the Lagrange polynomial interpolation and weighing functions. To force a strong gradient of the dimensionless temperature Φ, its unit value is assumed on the outlet of the channel.

The GFEM solution exhibits strong spatial oscillations and its value exceeds physically meaningful limits. Moreover, the CVFEM provides essentially the same incorrect solution when Lagrange polynomials are taken for the interpolation functions [14].

To suppress wiggles, either a very dense FE grid should be used or special *upwind* techniques, similar to the ones commonly incorporated in the FDM models [6,13], should be developed. In the PGFEM, the convective movement upwind to the flow is eliminated by using high order unsymmetric polynomials W_i for weighing process given by eq. (3) [1,3]. This way cannot be used, however, in the CVFEM where the weighing technique is not exploited.

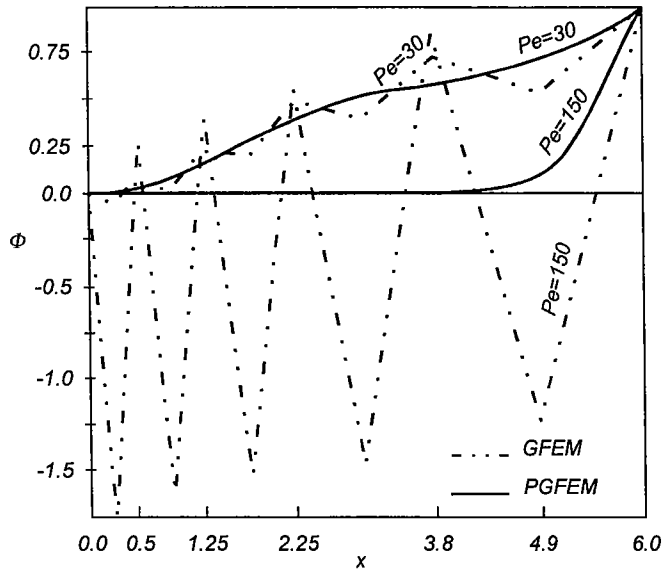

Fig. 5. FEM solution of temperature along the channel axis for the forced developing convection

The effect of upwinding can be obtained here by the development of the flow--oriented form of the interpolation function for the field quantity Φ, i.e.: the form which depends on both the direction and intensity of convection [8,11,15,21,22]. Although several such techniques have been proposed in the literature for the triangular ([8]) and rectangular ([11,15,21,22]) FE grids, there is no a general way to obtain the consistent approximations of all terms in the balance equation (eq. (1)). Therefore, a choice of any individual technique should take into account some characteristic features of the problem considered, like the scale of expected flow recirculations, the importance of the source term in the balance eq. (1) and the direction of the convective transport ([15]).

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

3.6. Numerical dispersion on FE grid

The transient dominated convection is of a great practical importance, as in a fluid flow far away from a wall the diffusion is negligible, and this leads to

the problem of a proper modelling of the almost pure convection transport on the FE grid. Indeed, the special care is needed here because the commonly used discretization models may cause the substantial dispersion error of the CVFEM and GFEM solutions [23,24,25].

For example, the lumped capacity matrix model (L-model), inherent in any FDM approximation and highly desirable in FEMs for a diffusive-type problem, cannot be accepted here, unless a very dense grid is used, due to its considerable dispersion error.

This is visible in Fig. 6, where the CVFEM and GFEM solutions are given to the benchmark problem of a pure convection of the scalar quantity Φ in a rotation velocity field ([24]). As the physical diffusion does not occur, the initial shape of Φ (given as a cone) should not change when Φ is convected in the domain. The numerical results, obtained after a complete revolution, are presented in Fig. 6 for both capacity matrix models. They reveal that:

- the L-model provides the solution of an unacceptable accuracy. It exhibits a considerable deformation of the initial shape and a significant reduction of the cone height. The latter error is further called the numerical dissipation;
- numerical dissipation is much smaller for the consistent capacity matrix model but the spurious waves behind the cone (Fig. 6) cannot be physically justified.

A usual way to examine the dispersion error, consisted of both the numerical dissipation and phase-shift errors, is via Fourier mode analysis of a discrete solution to the constant velocity pure convection problem [23, 24]. Upon the assumption that the continuous model (governed by the PDE for convection) properly describes physics of the phenomenon considered, Fourier series representation of the continuous model solution is compared with the relevant representation of its numerical counterpart. In the papers [24] and [25] Fourier mode analysis is used to compare errors of the CVFEM and GFEM solutions on a uniform rectangular grid and then to provide simple means to improve the performance of the control-volume based FEM.

It reveals that the potential inaccuracy of the FEM solution comes from the spatial approximation of the temporal and convection terms of the transient convection equation (eq. (1) or eq. (2) with $\lambda_{i,j} = 0$) and that the one-step implicit time-stepping scheme (eq. (12)) hardly influences the dissipation and the phase-shift errors.

Moreover, the comparison between the GFEM and CVFEM models shows that the latter model is visibly inferior in the convection dominated problems owing to its substantial dispersion error. Therefore, simple and no-time consuming means have been proposed [24,25] to improve the performance of the CVFEM for the transient pure convection transport problem. They consist in the spatial integration of the temporal term of the nodal balance equation in the way which ensures higher accuracy of the CVFEM model and in a use of

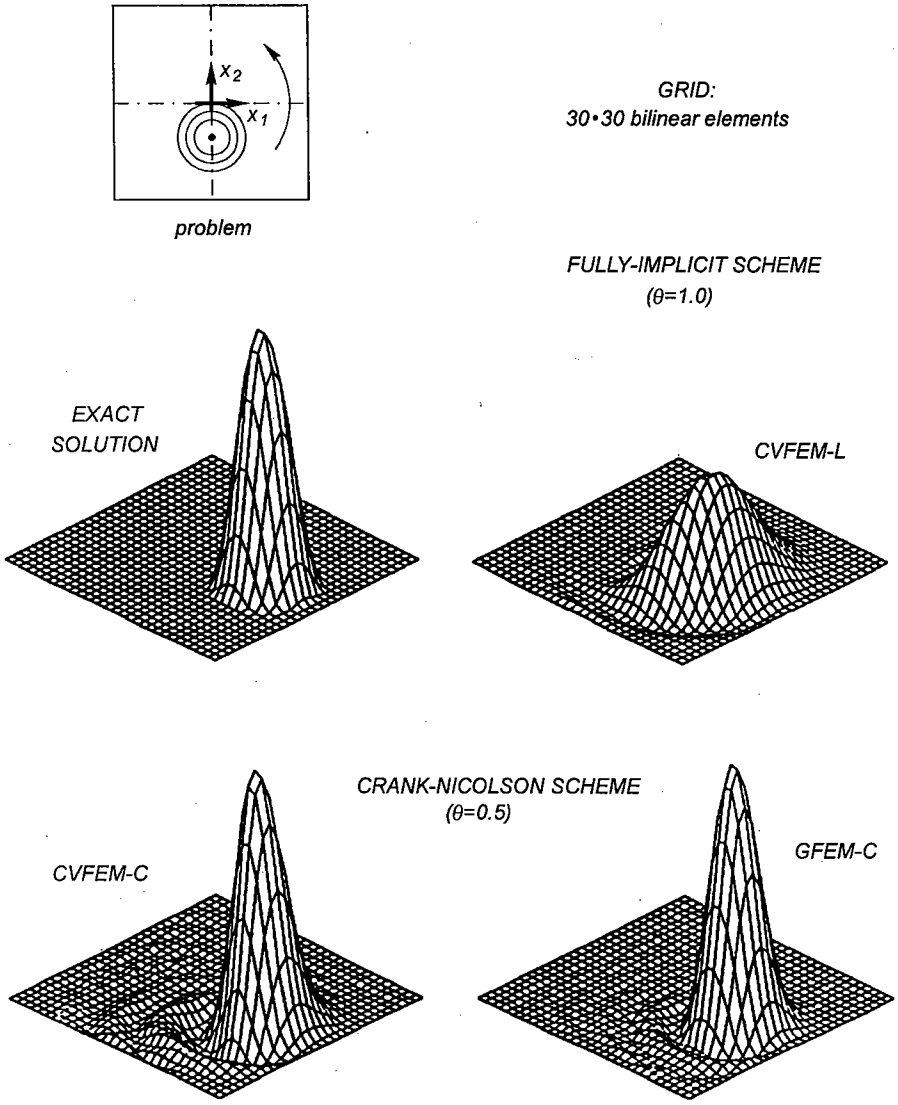

Fig. 6. GFEM and CVFEM solutions for convection of a cone in a rotating fluid

the Taylor series time representation of Φ including the second- and the third-order time derivatives. This significantly reduces both the numerical dissipation and dispersion and the model thus obtained (CVFEM-TM) offers the accuracy comparable to the one of the Taylor-Galerkin FEM (GFEM-T in Fig. 7) proposed in the paper [23]. The use of this special means in a development of the CVFEM model for the benchmark problem of the pure convection in the rotating fluid gives the solution which is not spoiled by visible numerical dispersion (compare Fig. 6 with Fig. 7).

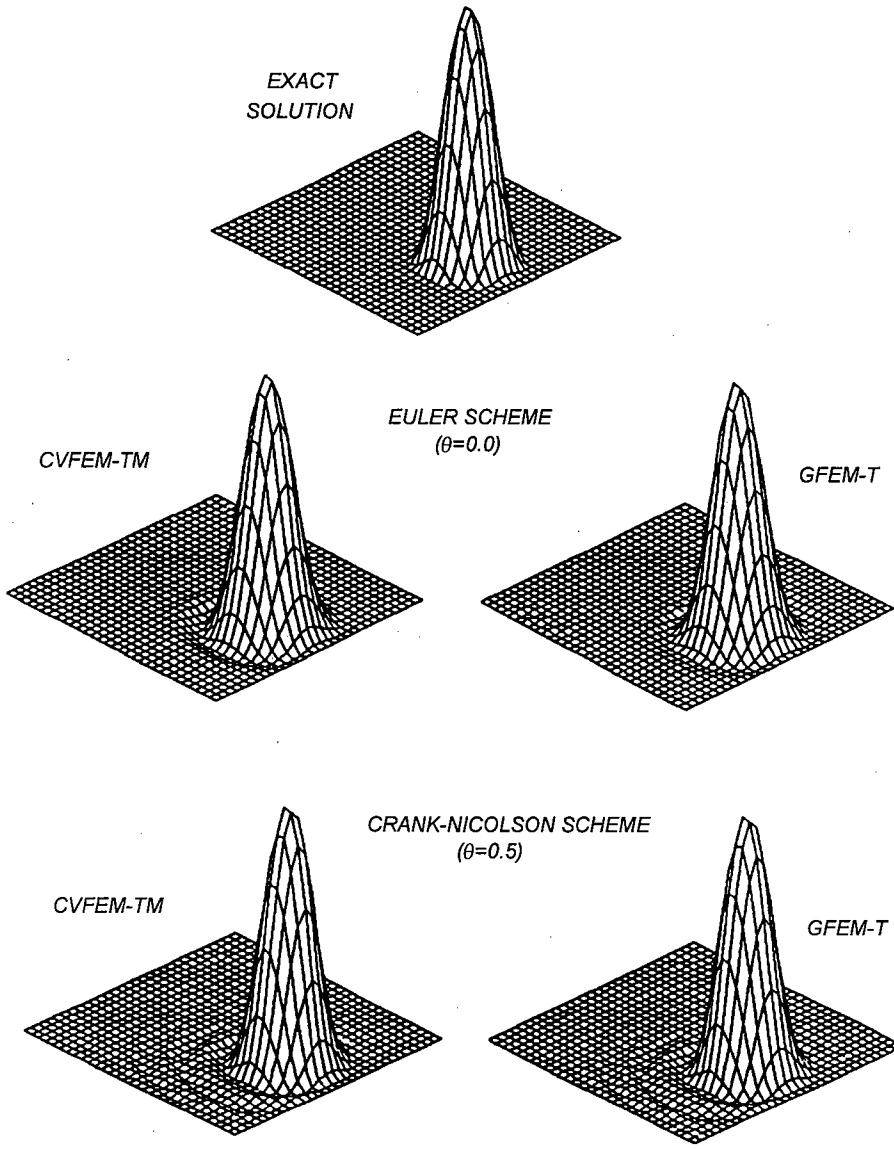

Fig. 7. CVFEM-TM and GFEM-T solutions for convection of a cone in a rotating fluid

Moreover, the better approximation of the temporal term of eq. (1), obtained by a retention of the higher order derivatives in Taylor series time expansion of Φ, provides the additional diffusive flux which operates only in the streamline direction. This simply forms the streamline upwinding in the CVFEM for transient convection dominated problems, which is similar to the *balancing tensor diffusivity* technique developed by P.M.Gresho for the GFEM [26].

3.7. Conclusions

The inaccuracy of the FEM solution caused by the wiggles, mass balance error, numerical dissipation and a violation of the conservation and maximum principles can be significantly reduced by the use of a very dense space-time discretization grid. This is, however, not reasonable from the computational economy point of view. Fortunately, simple and no-time consuming modifications of the classical FEM formulation, discussed above, give the same result but over a coarse FE grid. This, in turn, provides considerable savings in computer time and storage required for the FEM calculations of the connective-diffusion problems.

Moreover, it is shown above that the accuracy and computational efficiency of the FEM approximation of the convection-diffusion transport of the field quantity Φ depend on a proper choice of the discrete model, where the direction and intensity of convection and the intensity of diffusion should be taken into account. For example, for the diffusion-type problem the lumped capacity matrix model is a reasonable choice from the stability and early-time solution accuracy points of view. On the other hand, in the case when convection dominates (moderate and high Peclet numbers), the consistent capacity matrix model significantly reduces numerical dissipation and dispersion.

Hence, there is not a general FEM model, equally accurate and efficient for both the convection dominated problem and the diffusion dominated one. Each of them needs, therefore, individual approach. For each of them different optimal discrete models should be used to obtain efficient numerical technique.

In the fluid flow and heat transfer problems where, in general, a coupled convection-diffusion transport occurs, this can be established through a use of the component-by-component splitting-up method [27]. The convection and diffusion are treated there in two distinct phases of the time integration procedure. The method has been used by J. Donea [23] and B. Ramaswany [28] in the context of the Galerkin FEM. This paper presents its utilization to the Control-Volume based FEM [29].

4. EFFICIENT TIME INTEGRATION ALGORITHM FOR CONTROL--VOLUME FEM

4.1. Preliminary comments

The splitting-up technique enables to take the most optimal spatial and temporal discretization models as well as the most efficient solvers of a set of algebraic equations independently for convection and diffusion.

Hence, the unsteady diffusive problem is solved here with the lumped capacity matrix model and by a use of the successive over-relaxation iteration method (SOR) [30] or the conjugate gradient method [31] for the final set of algebraic equations obtained. On the other hand, to avoid significant dispersion and dissipation errors of the approximate solution as well as to perform the streamline upwinding the Taylor series expansion of the temporal term, including the high-order time derivatives is applied in the CVFEM formulation of the pure convection problem [24]. The set of linear algebraic equations is solved by LTDMA solver [6] or by the approximate factorization technique [23].

To decouple the continuity and momentum balance equations in the CVFEM, the fractional step method (also called the velocity correction algorithm) is used. This technique was originated by Chorin [32] in the FDM and it was successfully applied recently in the GFEM [28,33]. At first, the momentum balance equations are solved, disregarding the pressure gradient terms. Then, the provisional velocity field thus obtained is corrected by taking into account pressure contributions through an enforcement of the incompressibility requirement. The CVFEM algebraic equations, resulting from the Poisson-like equation for the pressure, are solved by the SOR or the conjugate gradient methods [29].

Strict enforcement of the continuity constraint, at every stage of the iterative process in the fractional step method, ensures that no spurious pressure modes appear in the FEM solution and thus equal-order pressure and velocity interpolations can be successfully used.

The question of cost effectiveness of the FEM solutions is still of the main interest in numerical modelling of practical, multidimensional fluid flow and heat transfer problems. It is commonly known that the FDM is superior in terms of the computer storage and execution time requirements. This mainly results from a use of the segregated velocity-pressure solution technique [6] where velocity and pressure are solved sequentially at each iteration or time step. On the other hand, most of the published FEM solutions to fluid flow have involved direct simultaneous solution of the continuity and momentum discrete equations [18], which is much more time consuming and requires large amount of computer storage. In recent years, however, a few papers have appeared, where the successful use of the segregated solution technique in the Galerkin FEM is reported (e.g. [28,33]). In the present study, this technique is applied in the CVFEM formulation.

The linearized momentum equations for each velocity components are solved sequentially and independently and they are followed by solving the pressure equation and by the velocity correction procedure at each iteration step.

The dependent variables, which are of interest in the fluid flow and heat transfer problems, i.e.: mass, components of momentum and enthalpy, obey, respectively, the following integral balance eqs within the control-volume Ω_k surrounding the node k (i.e.: eq. (1)):

$$\int_{\Gamma_k} \rho\, u_i n_i d\Gamma = 0 \tag{14}$$

$$\int_{\Omega_k} \rho \frac{\partial u_i}{\partial t} d\Omega + \int_{\Gamma_k} \rho\, u_i u_j n_j d\Gamma = \int_{\Gamma_k} \mu \frac{\partial u_i}{\partial x_j} n_j d\Gamma + \int_{\Omega_k} \left(S_i - \frac{\partial p}{\partial x_i} \right) d\Omega \tag{15}$$

$$\int_{\Omega_k} \frac{\partial}{\partial t} (\rho h) d\Omega + \int_{\Gamma_k} \left(\rho\, u_i h - \lambda \frac{\partial h}{\partial x_i} \right) n_i d\Gamma = \int_{\Omega_k} S_h d\Omega \tag{16}$$

for $i,j = 1,2,3;\ \ k = 1,2,...,NP$.

They are supplemented with boundary and initial conditions:

$$\begin{aligned} u_i &= u_{ib} && \text{on } \Gamma_u \\ -p + \mu \frac{\partial u_n}{\partial n} &= f_n \\ \mu \frac{\partial u_\tau}{\partial n} &= f_\tau && \text{on } \Gamma_\tau \end{aligned} \tag{17}$$

$$\begin{aligned} h &= h_b && \text{on } \Gamma_h \\ \frac{\partial h}{\partial n_i} n_i &= q && \text{on } \Gamma_q \end{aligned} \tag{18}$$

and

$$\left.\begin{aligned} u_i &= u_i^0 \\ p &= p^0 \\ \frac{\partial u_i^0}{\partial x_i} &= 0 \end{aligned}\right\} \text{ in } \Omega \text{ at } t = 0 \tag{19}$$

$$h = h^0 \quad \text{in } \Omega \text{ at } t = 0 \tag{20}$$

where in eq. (16) the viscous dissipation of an incompressible fluid is neglected.

The problem governed by eq. (14) to eq. (20) is decomposed into three simpler problems, i.e. convection, diffusion and continuity ones, being consecutively solved more effectively with a computer.

At first, disregarding the pressure contribution, the transient convective transport of momentum is solved for the intermediate velocity components u_i^*

$$\left.\begin{aligned} &\int_{\Omega_k} \rho \frac{\partial u_i^*}{\partial t}\, d\Omega + \int_{\Gamma_k} \rho\, u_i^* u_j^n n_j\, d\Gamma = 0 \\ &\text{with} \\ &u_i^* = u_{ib} \quad \text{on} \quad \Gamma_u \\ &u_i^* = u_i^n \quad \text{in} \quad \Omega_k \text{ at } t = n\Delta t \end{aligned}\right\} \quad \text{for } i,j = 1,2,3 \qquad (21)$$

To avoid the significant numerical dissipation and dispersion inherent in the classical FEM formulation for the pure convection problem, Taylor series expansion of the local derivative $\partial\Phi/\partial t$ of any field quantity Φ, including the high-order time derivatives, is applied [24]. This leads to the following balance equation referred to the control-volume Ω_k:

$$\int_{\Omega_k} \rho \frac{\Phi^{n+1} - \Phi^n}{\Delta t}\, d\Omega + \frac{\Delta t}{2} \int_{\Gamma_k} \rho\, u_i u_j \frac{\partial}{\partial x_j} \left\{ \theta \Phi^{n+1} - (1-\theta)\, \Phi^n \right\} n_i\, d\Gamma -$$

$$- \frac{\Delta t^2}{6} \int_{\Gamma_k} \rho\, u_i u_j \frac{\partial}{\partial x_j} \left(\frac{\Phi^{n+1} - \Phi^n}{\Delta t} \right) n_i\, d\Gamma = \qquad (22)$$

$$= -\theta \left(\int_{\Gamma_k} \rho\, u_i \Phi\, n_i\, d\Gamma \right)^{n+1} - (1-\theta) \left(\int_{\Gamma_k} \rho\, u_i \Phi\, n_i\, d\Gamma \right)^n$$

for $\Phi = u_l^*$, where $l = 1, 2, 3$. The CVFEM approximation of the problem governed by eq. (21), with the velocity field u_i^n known from the previous time step (previous iteration), is obtained from eq. (22) with $\theta = 0$ for each components of the intermediate velocity field $\Phi \equiv u_i^*$ interpolated by eq. (5). The sets of the algebraic equations thus obtained are solved separately for each components u_i^* by using the efficient LTDMA solver [6] or by the approximate factorization technique [23].

Next, the nodal values of u_i^* at t^{n+1} are used as the initial ones for the velocity components $\hat{u}_i$ obtained at the end of the time step by taking into account the diffusive transport of momentum governed by:

$$\left.\begin{aligned} &\int_{\Omega_k} \rho \frac{\partial \hat{u}_i}{\partial t}\, d\Omega = \int_{\Gamma_k} \mu \frac{\partial \hat{u}_i}{\partial x_j}\, n_j d\Gamma + \int_{\Omega_k} S_i d\Omega \\ &\text{with} \\ &\hat{u}_i = u_{ib} \qquad \text{on } \Gamma_u \\ &\hat{u}_i = u_i^{*,n+1} \qquad \text{in } \Omega_k \text{ at } t = n\Delta t \end{aligned}\right\} \quad \text{for } i,j = 1,2,3 \tag{23}$$

The lumped capacity matrix model (eq. (11)) is taken here and the above equations are solved separately for each momentum components. They are integrated in time using the second-order accurate, unconditionally stable Crank-Nicolson scheme (eq. (12) with $\theta = 0.5$). The final set of algebraic equations is then solved by the SOR or the conjugate gradient techniques.

Thus computed velocity field does not generally satisfy the incompressibility condition. Hence, it is further corrected by taking into account the pressure contribution to the solenoidal velocity field through the continuity and momentum equations. The final velocity field u_i^{n+1} at the end of the time step should satisfy the mass conservation principle for incompressible fluid referred to the control-volume Ω_k confined by the boundary Γ_k:

$$\int_{\Gamma_k} \rho u_i^{n+1} n_i d\Gamma = \int_{\Omega_k} \frac{\partial u_i^{n+1}}{\partial x_i}\, d\Omega = 0 \tag{24}$$

The pressure gradient is a factor which forces a correction of the velocity at t^{n+1} in the way which ensures a fulfilment of the incompressibility condition, hence:

$$\int_{\Omega_k} \rho \frac{u_i^{n+1} - \hat{u}_i}{\Delta t}\, d\Omega = -\int_{\Omega_k} \frac{\partial p^{n+1}}{\partial x_i}\, d\Omega \tag{25}$$

Taking the divergence of eq. (25) and eliminating u_i^{n+1} through eq. (24) one obtains:

$$\left.\begin{aligned} &\int_{\Omega_k} \frac{\partial}{\partial x_i}\left(\frac{\partial p^{n+1}}{\partial x_i}\right) d\Omega = \frac{1}{\Delta t} \int_{\Omega_k} \rho \frac{\partial \hat{u}_i}{\partial x_i}\, d\Omega \\ &p = p_b \quad \text{on } \Gamma_\tau \\ &\frac{\partial p}{\partial n} = \frac{1}{\Delta t}\left(\hat{u}_j - u_j^{n+1}\right) \quad \text{on } \Gamma_u \end{aligned}\right\} \tag{26}$$

The pressure is interpolated within each element in accordance with eq. (6) but for $Q_k(\zeta) \equiv P_l(\zeta)$ and $N_u = N_p$. It means that the equal-order pressure and velocity formulation is taken in the algorithm presented. The algebraic set

of eqs (26) is solved by means of the SOR or conjugate gradient methods. Once the pressure nodal values have been thus determined the velocity components u_i^{n+1} are calculated from eq. (25).

Eventually, the velocity field, satisfying the incompressibility condition at the end of the time step, is used to calculate enthalpy (or temperature) through the energy equation (16). At first, the pure convection transport of energy is analyzed

$$\left.\begin{aligned} &\int_{\Omega_k} \rho \frac{\partial h^*}{\partial t}\, d\Omega + \int_{\Gamma_k} \rho\, h^*\, u_i^n\, n_i\, d\Gamma = 0 \\ &\text{with} \\ &h^* = h_b \quad \text{on} \quad \Gamma_h \\ &h^* = h^n \quad \text{in} \quad \Omega_k \quad \text{at } t = n\Delta t \end{aligned}\right\} \tag{27}$$

by using the equation (22) with $\Phi = h^*$. The set of algebraic equations thus obtained is again solved by the LTDMA method or by the approximate factorization technique. The nodal values of the intermediate enthalpy h_i^* at time t^{n+1} are then used as the initial ones for the final h_i^{n+1}, obtained at the end of time step by taking into account the diffusive transport of energy:

$$\left.\begin{aligned} &\int_{\Omega_k} \rho \frac{\partial h}{\partial t}\, d\Omega - \int_{\Gamma_k} \lambda \frac{\partial h}{\partial x_i}\, n_i d\Gamma = \int_{\Omega_k} S_h d\Omega \\ &\text{with} \\ &h = h_b \qquad \text{on} \quad \Gamma_h \\ &h = h^{*,n+1} \quad \text{in } \Omega_k \quad \text{at } t = n\Delta t \end{aligned}\right\} \tag{28}$$

The lumped capacity matrix model (L-model) is taken here and the above equations are integrated in time using the unconditionally stable Crank-Nicolson scheme (eq. (12) with $\theta = 0.5$). The final set of algebraic equations is then solved by the SOR or the conjugate gradient techniques.

4.3. Numerical examples and conclusions

The accuracy and efficiency of the algorithm presented have been verified by solving a few specific test problems ([29]), commonly regarded in literature (e.g.: [28,34]) as the bench-mark ones for the comparison of alternative numerical models. The details of geometry, boundary conditions and the finite element grid used are given in Fig. 8, Fig. 9 and Fig. 10 along with the calculated velocity, pressure and temperature fields for three pertinent examples chosen.

The sudden expansion problem is commonly used as the benchmark test to verify if no spurious pressure modes appear in the approximate solution. It is applied here to find whether the equal-order velocity-pressure formulation can be taken in the fractional step approach with the segregated solution technique for the Control-Volume FEM. The geometry of the flow domain ($0 \le x_1 \le 6$, $0 \le x_2 \le 2$ with upward facing step of expansion 1:2) is depicted in Fig. 8. The boundary conditions are assumed as follows. At the top and bottom walls $u_1 = 0$ and $u_2 = 0$, whereas at the inlet $u_1 = 4x_2(1 - x_2)$, $u_2 = 0$ and at the outlet $\partial u_1/\partial x_1 = 0$, $u_2 = 0$, $p = 0$. The regular, rectangular grid ($\Delta x_1 = 0.25$, $\Delta x_2 = 0.1$) of 400 bilinear elements is shown in Fig. 8 along with the steady state distribution of streamlines and pressure contours, calculated with $\Delta t = 0.1$ in the case where $Re = 50$ ($\mu/\rho = 0.02$) at the inlet.

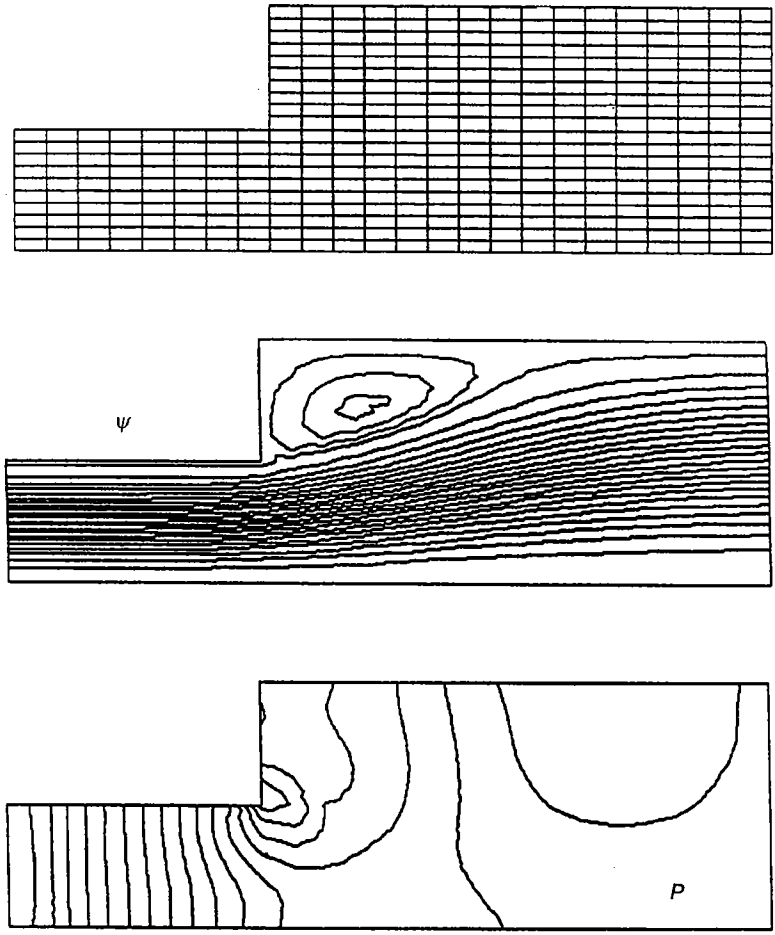

Fig. 8. Geometry, FE grid, streamlines (ψ) and pressure contours (P) for flow through a sudden enlargement

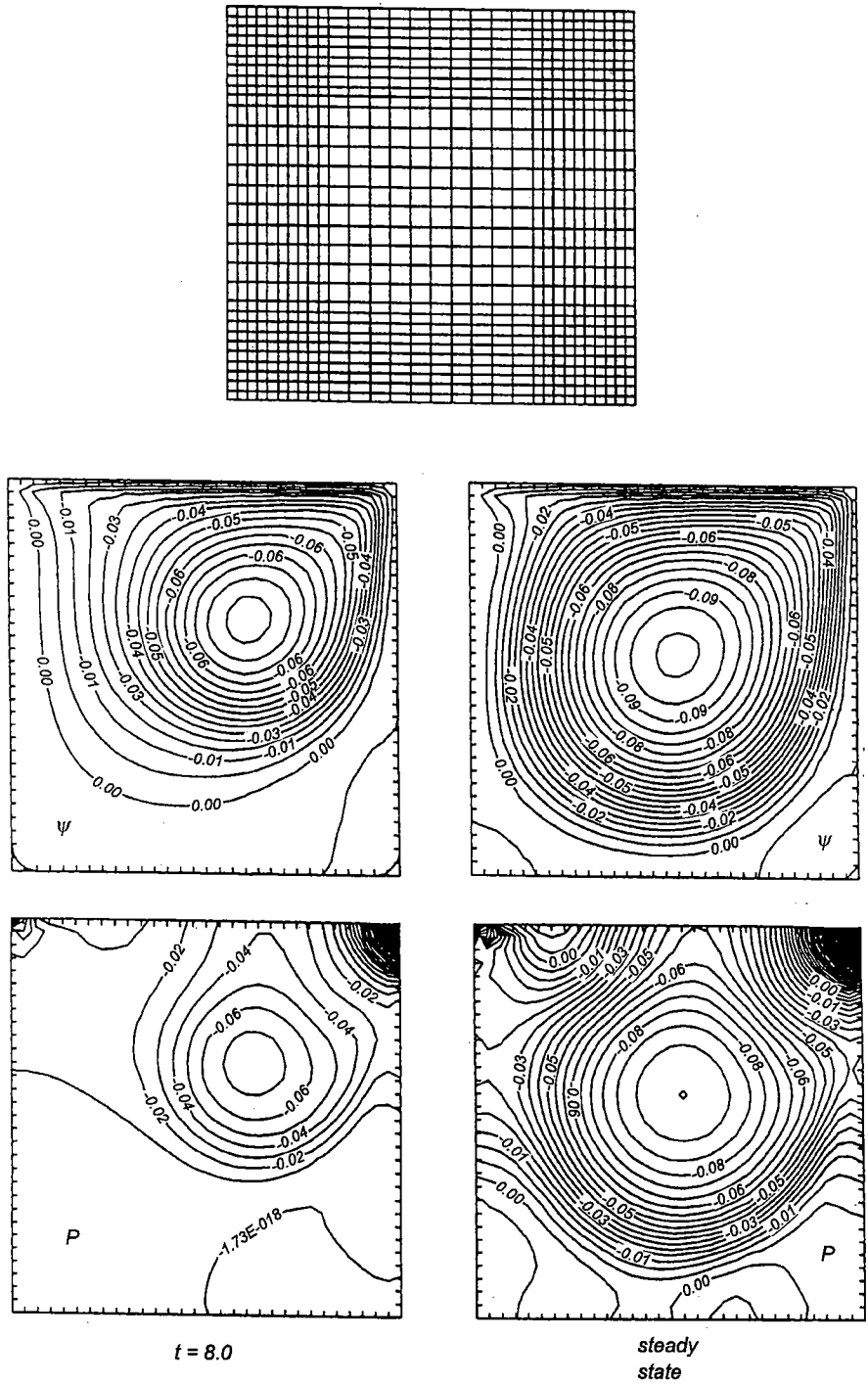

Fig. 9. Geometry, FE grid, streamlines (ψ) and pressure contours (P) for lid driven cavity flow

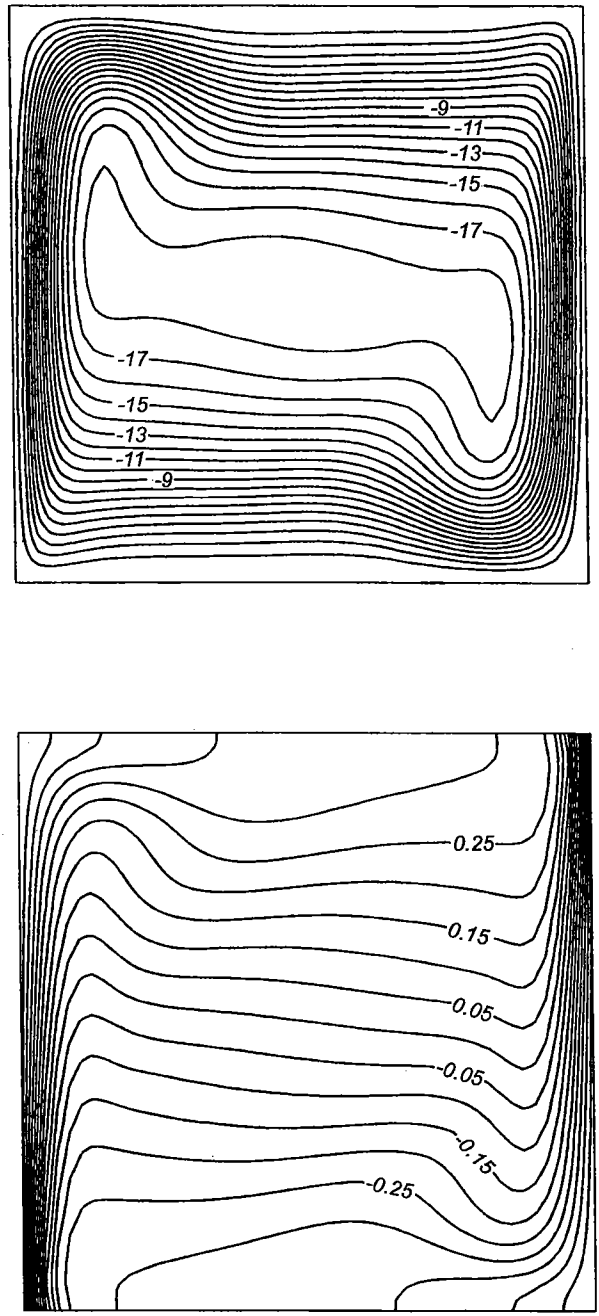

Fig. 10. Streamlines and isothermal lines for free convection in a square enclosure with $Gr = 10^6$

Next, to examine the performance of the CVFEM algorithm for recirculated flows the problem of flow in a closed cavity ($0 \le x_1 \le 1$, $0 \le x_2 \le 1$) driven by the lid movement has been considered. The lid moves with a unit velocity ($u_1 = 1.0$) in its own plane. The no-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.: $Re = \rho u_l / \mu$. The nonuniform grid of 900 bilinear finite elements (Fig. 9) and $\Delta t = 0.04$ were used in the case of $Re = 1000$ (i.e. $\mu/\rho = 0.001$). The results obtained are given in Fig. 9 in terms of streamlines and pressure contours at $t = 8.0$ and for the steady-state case.

Eventually, to verify the algorithm accuracy in the coupled fluid flow and heat transfer, the laminar free convection in the square enclosure has been analyzed for different angles of inclination. The undimensional domain is confined by $0 \le x_1 \le 1$ and $0 \le x_2 \le 1$. The lower and upper walls are assumed to be adiabatic ones ($u_1 = u_2 = 0$, $\partial\Phi/\partial x_2 = 0$, Φ – undimensional temperature) More-over, $u_1 = u_2 = 0$ at the vertical walls. The left one is maintained at constant undimensional temperature $\Phi = 0.5$ and $\Phi = -0.5$ is taken at the right wall. Exemplifying results obtained are given in Fig. 10 for Grashoff number $Gr = 10^6$ in terms of steady-state distribution of streamlines and isotherms, obtained on the nonuniform grid of 2500 bilinear elements.

An effective computing technique has been presented for calculating the transient incompressible viscous flow and coupled heat transfer by means of the Control-Volume FEM. This technique is based on the velocity-pressure formulation, splitting up and fractional step methods. The preliminary results for the test problems presented in Fig. 8 to Fig. 10 show that the solutions obtained are free from the wiggles and spurious pressure modes and that they fit fairly well to the results presented by B.Ramaswany [28] and M.Strada [33]. Moreover, due to a use of the segregated solution technique the better computational efficiency has been obtained in comparison with the one of the simultaneous solution algorithm [18] and thus the computational economy is much closer to the one reported elsewhere (e.g.: [6]) for the CVFDM.

REFERENCES

[1] Zienkiewicz O.C.: Finite Element Method. Fourth Edition, McGraw-Hill Company, London 1989.

[2] Oden J.T., Reddy J.N.: An Introduction to the Mathematical Theory of Finite Elements. John Wiley & Sons, New York 1976.

[3] Brooks A.N., Hughes T.J.: Streamline Upwind Petrov-Galerkin Formulation for Convection Dominated Flows with Particular Emphasis on the Incompressible Navier-Stokes Equations. *Comp.Meth.Appl.Mech.Eng.* 1982, Vol.32, pp.199-259.

[4] Brebbia C.A., Telles J.C., Wróbel L.C.: Boundary Element Techniques. Theory and Applications in Engineering. Springer-Verlag, Berlin 1984.

[5] Wróbel L.C., Brebbia C.A., Nowak A.: *Proceedings of the First International Conference on Advanced Computational Methods in Heat Transfer, Computational Mechanics Publications*. Springer-Verlag, Berlin 1990.

[6] Patankar S.V.: Numerical Heat Transfer and Fluid Flow. Hemishere Publishing Corporation, New York 1980.

[7] Raithby G.D., Schneider G.E.: Elliptic System: Finite Difference Method II. **In**: Handbook of Numerical Heat Transfer. (Ed. W.Minkowycz) John Wiley & Sons, New York 1988.

[8] Baliga B.R., Patankar S.V.: A Control-Volume Finite Element Method for Two-Dimensional Fluid Flow and Heat Transfer. *Numerical Heat Transfer*. 1983, Vol.6, pp.245-261.

[9] Schneider G.E., Raw H.J.: A Skewed Possitive-Influence Coefficient Upwinding Procedure for Control-Volume Based Finite Convection-Diffusion Computation. *Numerical Heat Transfer*. 1986, Vol.9, pp.1-26.

[10] Banaszek J.: A Conservative Finite Element Method for Heat Conduction Problems. *Int.J.Num.Meth.Eng.* 1984, Vol.20, pp.2033-2050.

[11] Banaszek J. and Staniszewski B.: A Conservative Finite Element Method for Convective-Diffusion Problems. **In**: Integral Methods in Science and Engineering. (Ed. F.Pane) Hemishere Publishing Corporation, Washington 1985, pp.435-454.

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

[13] Roache S.P.: Computational Fluid Dynamics. Hermosa Publishing, New Mexico 1982.

[14] Banaszek J.: Behavioral Error of the FEM Solution for Convection-Diffusion Problems. *Zeszyty Naukowe Politechniki Łódzkiej, Seria Maszyny Przepływowe*, 1988 z.97, pp.41-71 (in Polish).

[15] Banaszek J.: Analysis of Physical Correctness of Finite Element Models for Heat Transfer Problems. WPW Warszawa, 1991 (in Polish).

[16] Banaszek J.: A Conservation Property in Finite Element Approximation of Convective--Diffusive Transport Phenomena. *Archives of Thermodynamics*. 1993, Vol.14, pp.93-116.

[17] Banaszek J.: Comparison of Control-Volume and Galerkin Finite Element Methods for Diffusion-Type Problems. *Numerical Heat Transfer*. 1989, Part B, Vol.16, pp.59-78.

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

[19] Ciarlet P.C.: Discrete Maximum Principle for Finite Difference Operators. *AEQ Math.* 1970, Vol.4, pp.338-352.

[20] Rank E.,Katz C.,Werner H.: On the Importance of the Discrete Maximum Principle in Transient Analysis Using Finite Element Methods. *Int.J.Num.Meth.Eng.*, 1984, Vol.19, pp.1771-1782.

[21] Schneider G.E.: Elliptic Systems: Finite Element Method I. **In**: Handbook of Numerical Heat Transfer. (Ed. W.Minkowycz) John Wiley & Sons, New York 1988.

[22] Hookey N.A., Baliga B.R., Prakash C.: Evaluation and Enhancements of Some Control--Volume Finite Element Methods. *Numerical Heat Transfer.* 1988, Vol.14, pp. 255-271.

[23] Donea J., Giuliani S., Laval H. and Quartapelle L.: Time-accurate Solution of Advection-Diffusion Problems by Finite Elements. *Comp.Meth. Appl.Mech.Eng.* 1984, Vol.45, pp.123-145.

[24] Banaszek J.: Fourier Mode Analysis in Error Estimation for Finite Element Solution of Convection Problems. *Archives of Thermodynamics.* 1991, Vol.12, pp.35-56.

[25] Banaszek J.: Fourier Mode Analysis in Error Estimation for Finite Element Solution to Convective-Diffusive Phenomena. *Archives of Thermodynamics.* 1993, Vol.14, pp.67-92.

[26] Gresho, P.M., Lee,R.L. and Sani,R.L.: On the Time-Dependent Solution of the Incompressible Navier-Stokes Equations in Two and Three Dimensions. **In**: Recent Advances in Numerical Methods in Fluids. (Eds. C.Taylor, K.Morgan) Pineridge, Swansea 1980, pp.27-79.

[27] Marchuk G.I.: Methods of Numerical Mathematics. Springer-Verlag, Berlin and New York 1975.

[28] Ramaswany B.: Efficient Finite Element Method for Two-Dimensional Fluid Flow and Heat Transfer Problems. *Numerical Heat Transfer.* 1990, Part B, Vol.17, pp.123-154.

[29] Banaszek J.: Splitting-up Technique in Control-Volume Based FEM for Incompressible Fluid Flow and Heat Transfer Problems. *Proceedings of the Second International Conference on Advanced Computational Methods in Heat Transfer, Computational Mechanics Publications.* Springer-Verlag, Berlin 1992.

[30] Hirsh C.: Numerical Computation of Internal and External Flows. Vol.1, John Wiley & Sons, Chichester 1988.

[31] Axelsson O., Barker V.A.: Finite Element Solution of Boundary Value Problems. Accademic Press, New York 1984.

[32] Chorin A.J.: Numerical Solution of the Navier-Stokes Equations. *Math. Comput.* 1968, Vol.22, pp.745-762.

[33] Comini G., Del Giudice S.: Finite-Element Solution of the Incompressible Navier-Stokes Equations. *Numerical Heat Transfer.* 1982, Vol.5, pp.463-478.

[34] Strada M. and Heinrich J.C.: Heat Transfer Rates in Natural Convection at High Rayleigh Numbers in Rectangular Enclosers: A Numerical Study. *Numerical Heat Transfer.* 1982, Vol.5, pp.81-93.

Acknowledgment

The research has been supported by the KBN, Grant Number 993209203.

O PEWNYCH TEORETYCZNYCH I PRAKTYCZNYCH ASPEKTACH EFEKTYWNEGO MODELOWANIA ZJAWISK WYMIANY CIEPŁA NA SIATKACH ELEMENTÓW SKOŃCZONYCH

Streszczenie

W pracy przedstawiono porównanie współczesnych technik numerycznych stosowanych w symulacji komputerowej ruchu płynu nieściśliwego i wymiany ciepła z uwzględnieniem ich dokładności i efektywności obliczeniowej.

W szczególności, rozważono sposoby przyspieszenia obliczeń realizowanych metodą elementów skończonych, opartą na technice reszt ważonych Galerkina lub na bilansach masy, pędu i energii w objętościach kontrolnych.

Weryfikacja warunków poprawności fizykalnej rozwiązania numerycznego, obejmującej spełnienie zasad zachowania i maksimum oraz poprawne modelowanie dyskretne zjawiska przenoszenia konwekcyjnego, została wykorzystana w tworzeniu modeli elementów skończonych o zadowalającej dokładności na rzadkich siatkach podziału.

Ponadto, budowa algorytmu obliczeniowego na podstawie techniki rozdziału w czasie, w której udział poszczególnych form przenoszenia, tj. konwekcji i dyfuzji, modeluje się niezależnie, pozwoliła na dobór najbardziej optymalnych i efektywnych postaci przestrzenno-czasowej dyskretyzacji dla każdej z tych form transportu wielkości polowej.

О НЕКОТОРЫХ ТЕОРЕТИЧЕСКИХ И ПРАКТИЧЕСКИХ АСПЕКТАХ ЭФФЕКТИВНОГО МОДЕЛИРОВАНИЯ ЯВЛЕНИЙ ТЕПЛООБМЕНА НА СЕТКАХ КОНЕЧНЫХ ЭЛЕМЕНТОВ

Краткое содержание

В работе дано сравнение современных численных техник, применяемых при моделировании на вычислительных машинах движения несжимаемой жидкости и теплообмена с учетом их расчетной точности и эффективности.

В особенности рассматривались способы ускорения расчетов, осуществляемых методом конечных элементов, основанным на технике взвешиванных погрузок Галеркина или на балансах массы, энергии и количества движения в контрольных обьменах.

Верификация условий физической правильности численного решения, охватывающей выполнение закона сохранения и максимума, а также правильное дискретное моделирование конвекционной передачи, была использована при создавании достаточно точных моделей конечных элементов в редких сетках деления.

Кроме того, структура вычислительного алгоритма на основе техники разделения во времени, при которой участие отдельных форм передачи, то есть конвекции и диффузии, моделируется независимо, позволяет выбрать наиболее оптимальную и эффективную форму пространственно-временной дискретизации для каждой из этих форм передачи полевой величины.