MODELLING OF FORCED CONVECTION USING A STABILIZED MESHLESS METHOD

Adam Kulawik

Institute of Computer and Information Science, Czestochowa University of Technology, Poland
adam.kulawik@icis.pcz.pl

Abstract. Modeling of coolant flow is increasingly important in the problems of heat treatment. Variation of the coolant characteristics is not possible to present, as a boundary condition at the appropriate level of detail. This paper presents a solution of the heat transfer equation with a convective term and a Navier-Stokes equation for forced flow, which is characteristic for a hardening process. These equations are solved by means of a meshless method. The emerging models for solving thermal processes in coolants require a stable numerical method. In this paper a stabilization of the numerical models for the generalized finite difference method for both temperature and fluid flow models are proposed. The results obtained with software implementations of the proposed models are compared with the an analytical solution and with the numerical benchmarks.

Introduction

Modeling of cooling for a number of technological processes (heat treatment) requires the consideration of forced motion of the coolant. Modeling of a fluid flow is associated with the solution of differential equations of heat transport with convective term and the Navier-Stokes equations. The solution of these equations presents a number of numerical problems. One is the instability of the algorithms associated with high velocity and large *Peclet* numbers. In this paper the numerical model, which does not require regular grids, based on a generalized finite element method with stabilization is presented [1-3].

1. Mathematical model

Considered region consists of two areas - a steel element and a liquid coolant. It usually assumed that the influence of the coolant on a steel element is modeled by adequate boundary conditions. Fluid motion in the coolant caused by external forced is taken into account. Liquid movement caused by a vertical gradient of the temperature is neglected. With such high rates of forced, the convective motions caused by the impact of temperature are negligible.

The mathematical model consists of the heat transport equation with a convective term, the Navier-Stokes equation and the continuity one [3-6].
Heat transfer equation in following form is assumed

$$(\lambda T,_{\alpha}),_{\alpha} - \rho C T,_{t} - \rho C V_{\alpha} T,_{\alpha} = q_{v} \tag{1}$$

where T [K] is the temperature, t [s] is the time, λ [W/mK] is the thermal conductivity, ρ [kg/m^3] is the density, C [J/kgK] is the specific heat, V [m/s] is the velocity, and q_v [J/m^{3}s] is the volumetric heat source.
The Navier-Stokes equation is defined as follows

$$\left(\frac{\mu}{\rho} V_{\alpha,\beta}\right),_{l} - V_{\beta} V_{\alpha,\beta} - \frac{1}{\rho} p,_{\alpha} = \dot{V}_{\alpha} \tag{2}$$

and the continuity equation takes the form

$$V_{\alpha},_{\alpha} = 0 \tag{3}$$

where V_α [m/s] is the velocity component in the α-direction, μ [kg/ms] is the dynamic viscosity, p [kg/ms^2] is the pressure.

Equations (1) are (2) are supplemented by appropriate boundary and initial conditions.

2. Numerical model

The temperature in nodes was determined by the solution of the heat transfer equation with a convective term based on a generalized finite difference scheme GFDM in a nonlinear implicit time scheme written in the matrix form as

$$\mathbf{Am} \cdot \mathbf{T} = \mathbf{Dw} \tag{4}$$

where matrix **Am** is defined as

$$Am_{i,i} = \beta \left(\begin{aligned} & \sum_{j=1}^{n} \left(z_j^{xx} + z_j^{yy}\right)\lambda + \\ & + \sum_{\alpha} \left(\sum_{j=1}^{n} z_j^{\alpha} \lambda_j - \lambda_i \sum_{j=1}^{n} z_j^{\alpha} \right) \left(- \sum_{j=1}^{n} z_j^{\alpha} \right) + \rho c \left(\sum_{j=1}^{n} z_j^{\alpha} V_i^{\alpha} \right) \end{aligned} \right) - \frac{\rho c}{\Delta t} \tag{5}$$

$$Am_{i,j} = \beta \left(\left(z_j^{xx} + z_j^{yy}\right)\lambda + \sum_{\alpha} \left(\sum_{j=1}^{n} z_j^{\alpha} \lambda_j - \lambda_i \sum_{j=1}^{n} z_j^{\alpha} \right) z_j^{\alpha} - \rho c z_j^{\alpha} V_i^{\alpha} \right)$$

and vector **Dw** is written as

$$Dw_i = -\frac{\rho c}{\Delta t} T_i^{s-1} - Q_i - (1-\beta)$$
$$\left(\begin{array}{l} -\sum_{j=1}^{n} \left(z_j^{xx} + z_j^{yy} \right) \lambda + \\ + \sum_{\alpha} \left(\sum_{j=1}^{n} z_j^{\alpha} \lambda_j - \lambda_i \sum_{j=1}^{n} z_j^{\alpha} \right) \left(-\sum_{j=1}^{n} z_j^{\alpha} \right) + \rho c \left(\sum_{j=1}^{n} z_j^{\alpha} V_i^{\alpha} \right) \end{array} \right) T_i^{s-1} - \tag{6}$$
$$+ (1-\beta) \sum_{j=1}^{n} \left(\left(z_j^{xx} + z_j^{yy} \right) \lambda + \sum_{\alpha} \left(\sum_{k=1}^{n} z_k^{\alpha} \lambda_k - \lambda_i \sum_{k=1}^{n} z_k^{\alpha} \right) \left(z_j^{\alpha} \right) - \rho c z_j^{\alpha} V_i^{\alpha} \right) T_j^{s-1}$$

where z are the coefficients of approximation of derivatives for GDFM [3].

The Navier-Stokes equation (2) is solved only in a region filled with coolant by means of a characteristic based split (CBS) scheme. The CBS scheme is based on the projection method was developed by Chorin [7] and described by Zienkiewicz and Codina [7]. In this method an auxiliary velocity field V* is introduced [3, 7] to uncouple equations (2) and (3)

$$V_{\alpha}^{*} = \Delta t \left(\frac{\mu}{\rho} V_{\alpha},_{\beta\beta} - V_{\beta} V_{\alpha},_{\beta} \right) + V_{\alpha}^{s-1} \tag{7}$$

The momentum equation was solved by GDFM using an implicit time scheme for ***i***-th node of the grid. This solution in the matrix form is written as

$$\mathbf{Am}_{\alpha} \cdot \mathbf{V}_{\alpha}^{*} = \mathbf{Dw}_{\alpha} \tag{8}$$

where matrix $\mathbf{Am}_{\alpha}$ is described as

$$Am_{i,i}^{\alpha} = 1 + \Delta t \left(\frac{\mu}{\rho} \sum_{j=1}^{n} \left(z_j^{xx} + z_j^{yy} \right) \right)$$
$$Am_{i,j}^{\alpha} = -(z_j^{xx} + z_j^{yy}) \frac{\Delta t \mu}{\rho} \tag{9}$$

and vector $\mathbf{Dw}_{\alpha}$ is written as

$$Dw_i^{\alpha} = \left(V_{\alpha}^{s-1} \right)_i - \Delta t \left(V_x^{s-1} \right) \tag{10}$$

The pressure field is obtained by solving the following Poisson equation

$$p,_{\alpha\alpha} = \frac{\rho}{\Delta t} V_{\alpha}^{*},_{\alpha} \tag{11}$$

The above equation for *i*-th node in GDFM convention takes thr following form

$$\sum_{j=1}^{n}\left(\left(z_j^{xx}+z_j^{yy}\right)p_j\right)-p_i\sum_{j=1}^{n}\left(z_j^{xx}+z_j^{yy}\right)= \\ \frac{\rho}{\Delta t}\left(\left(\sum_{j=1}^{n}z_j^{x}V_{xj}^{*}-\sum_{j=1}^{n}z_j^{x}V_{xi}^{*}\right)+\left(\sum_{j=1}^{n}z_j^{y}V_{yj}^{*}-\sum_{j=1}^{n}z_j^{y}V_{yi}^{*}\right)\right) \tag{12}$$

The final velocity field is corrected by the pressure increment

$$\Delta V_\alpha^{*}=-\frac{\Delta t}{\rho}\left(p,_\alpha\right) \tag{13}$$

The solution of equation (11) in GDFM for *i*-th node is as follows

$$\Delta V_{\alpha i}^{*}=-\frac{\Delta t}{\rho}\left(\left(\sum_{j=1}^{n}z_j^{(\alpha)}p_{\alpha j}-\sum_{j=1}^{n}z_j^{(\alpha)}p_{\alpha i}\right)\right) \tag{14}$$

3. Stabilization method

Numerical modeling of the phenomena with high rates are causing problems with the stabilization of solutions. In the paper, the stabilization of the differential method using a combination of derivatives (determined on several nodal grids of nodes) was proposed.

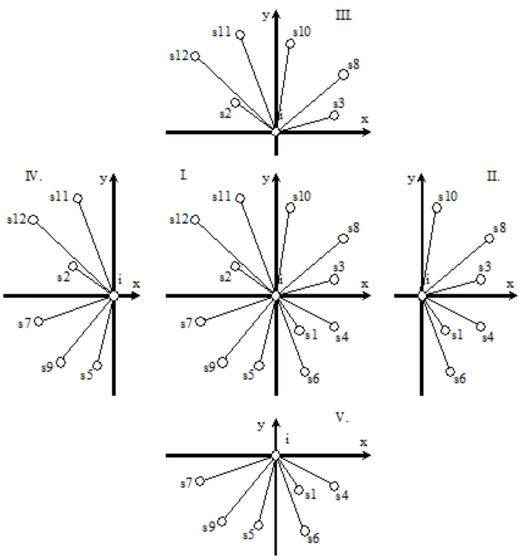

Fig. 1. Examples of grid nodes used for the calculation

The modification of the coefficients in the stabilization of GFDM is described by the following equation

$$z_\alpha = (1-\zeta_\alpha) z_\alpha^I + \zeta_\alpha z_\alpha^j,\ j = \text{II..V}, \qquad \zeta_\alpha = \left| \frac{1}{\tanh(Pe_\alpha / 2)} - \frac{2}{Pe_\alpha} \right| / 2 \tag{15}$$

where Pe is a local *Peclet* number defined as follows: in the heat transfer model - $Pe_\alpha = v_\alpha r_\alpha \rho C/\lambda$, in ithe flow model - $Pe_\alpha = \rho v_\alpha r_\alpha/\mu$, r_α is a characteristic size of the element of grid in the α direction.

4. Evaluations of models

The differential equation of heat transport using a generalized finite difference method in the control area **Ω** was solved. The rectangular geometry of dimensions 1×1 m, the initial and boundary conditions were adopted in such a way, that numerical calculations results of the approximate solution can be compared to an existing analytical solution.

As an initial condition we assumed, that the entire area $T_0(x_\alpha)$ = 300 K and the boundary conditions were follows:
on the left boundary (Γ, x = 0) the Dirichlet condition T_D(x = 0) = 1000 K
on the other edge the Neumann condition q = 0 W/m^2
For this case, the analytical solution is of the following form [7]

$$T(x,t) = \frac{1}{2}\left(T^D - T_0\right)\left(erfc\left(\frac{x-ut}{\sqrt{4Dt}}\right) + \exp\left(\frac{ux}{D}\right) erfc\left(\frac{x+ut}{\sqrt{4Dt}}\right)\right) + T_0 \tag{16}$$

where T [K] is the temperature, t [s] is the time, $D = \lambda/(\rho C)$.
The obtained results are shown in Figure 2.

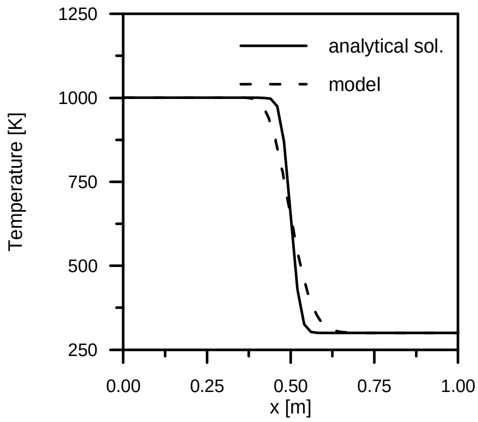

Fig. 2. Comparison of numerical model with the analytical solution, u = 1 m/s, t = 0.5 s, λ = 0.5 W/mK, ρ = 1 kg/m^3, C = 1000 J/kgK

Numerical model of coolant flow was verified using the numerical benchmarks involving a forced flow in a closed area with a movable top wall - Driven Cavity Test (Ghia et al.) [8]. The conditions of the test were as follows: area dimension d = 1 m, density ρ = 1 kg/m^3, dynamic viscosity μ [kg/ms] is depending on the *Reynolds* number.

a)

b)

c)

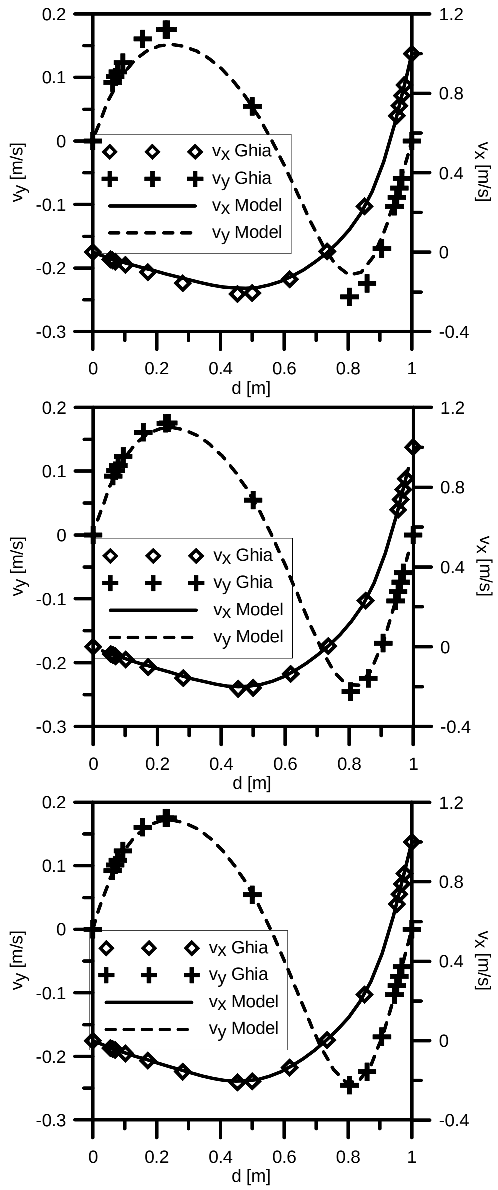

Fig. 3. Comparison of calculation results with the numerical benchmark, Re = 100, a) 50x50 nodes, b) 100x100 nodes, c) 150x150 nodes

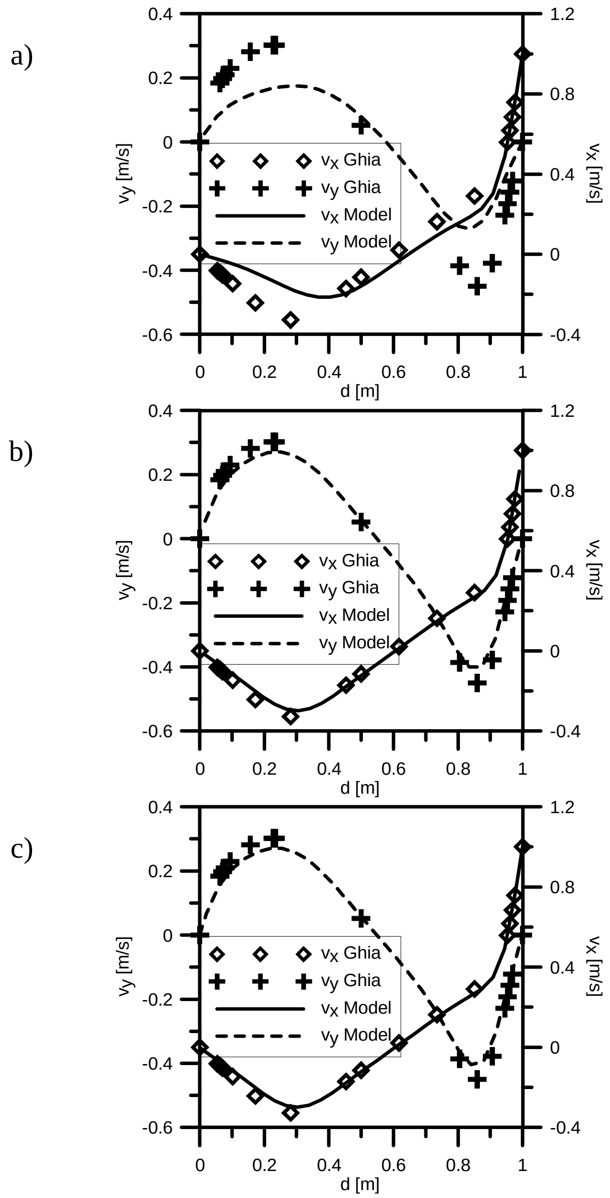

Fig. 4. Comparison of calculation results with the numerical benchmark, Re = 400, a) 50x50 nodes, b) 100x100 nodes, c) 150x150 nodes

Conclusions

The generalized finite difference method can be successfully applied to problems with irregular grids. GFDM is effective in the modeling processes for complex geometries. Stabilization of this method allows one to model the flow with large values of forced coolant velocity. The obtained results from the numerical model with stabilization suggest some "smoothing" of the results, but it is acceptable. Without the presented stabilization, this numerical model does not lead to accurate results for large *Peclet* numbers. This model may be used to estimate the temperature field during a cooling process for hardening steel tools.

References

[1] Benitoa J.J., Ureńaa F., Gaveteb L., Solving parabolic and hyperbolic equations by the generalized finite difference method, Journal of Computational and Applied Mathematics 2007, 209, 208-233.

[2] Liszka T., An interpolation method for an irregular net of nodes, Internat. J. Numer. Methods Eng. 1984, 20, 1599-1612.

[3] Pawlak E., Application of GFDM to model of linear and nonlinear heat diffusion problems, PhD Thesis, Częstochowa 1999 (in Polish).

[4] Węgrzyn-Skrzypczak E., Modelling of solidification with motion of the fluid in liquid and mushy zone, PhD Thesis, Częstochowa 2005 (in Polish).

[5] Zienkiewicz O.C., Codina R., A general algorithm for compressible and incompressible flow, Part I. The split characteristic based scheme, International Journal for Numerical Metods in Fluids 1995, 20, 869-885.

[6] Cardle J.A., A modification of the Petrov-Galerkin method for the transient convection-diffusion equation, International Journal for Numerical Methods in Engineering 1995, 38, 171-181.

[7] Chorin A.J., Numerical solution of the Navier-Stokes equation, Math. Comput. 1968, 23, 745--762.

[8] Ghia U., Ghia K.N., Shin C.T., High-Re solutions for incompressible flow using the Navier--Stokes equations and a multigrid method, J. of Computational Physics 1982, 48, 3, 387-411.