UDK 532.595

M. Kolodziejczyk

ON A CERTAIN METHOD FOR NUMERICAL ANALYSIS OF THE NAVIER – STOKES EQUATIONS

The present paper describes a modification and development of the method presented in [18] *for the determination of unsteady plane flow of viscous incompressible fluid. The main feature of the method consists in such elimination of the pressure from the governing equations by means of integration that the order of resulting system is not increased in comparison with the original one. This operation leads to the initial problem for a system of the first order ordinary differential equations. In this paper the method was modified by application of the staggered grid for velocity components. Numerical results and their comparison with results obtained by other authors are presented in order to verify the method.*

1. Introduction. Many flows in nature and technological devices are viscous and incompressible. They are governed by the Navier – Stokes equations, describing both laminar and turbulent flow. The development of the methods for solving complete non-simplified Navier – Stokes equations is an important part of computational fluid dynamics. Even though in most cases these methods are too computationally demanding on today's computers, they can serve to study the physics of the flow, to predict and analyze turbulent flow or may provide tools for the averaging methods with turbulence modeling or reference databases for fitting parameterized models [1, 6, 8–10, 16, 17].

The numerical approximation of the Navier – Stokes equation is generally difficult due to the coupling between velocity and pressure fields and the presence of the non-linear convective term. The most popular numerical methods for decoupling of velocity and pressure fields and serving the solution to the Navier – Stokes equations are operator splitting methods, and they are the subjects of many papers [1, 2, 6, 8–11, 14, 16, 17, 19, 20]. They are based on discretizing first in time in order to get a set of simpler partial differential equations for which many efficient numerical methods exist. The specific feature of these approaches is deriving of the Poisson equation for the pressure that demands more boundary conditions than the original problem.

The method for numerical solution to the whole non-simplified Navier – Stokes equations, considered in this paper, is based on discretizing first in space. Hence, the implementation of correct boundary conditions is much more easier than in other splitting methods [16]. The method was first presented in [18]. This paper is a continuation and extension of earlier studies of this method [12, 13].

2. Statement of the problem and the method of solution. This paper deals with plane, unsteady flow of viscous, incompressible fluid of constant density and viscosity described by the known system of partial differential equations:

$$u_x + v_y = 0\,, \tag{1}$$

$$u_t + (u^2)_x + (uv)_y = -p_x + \frac{1}{\mathrm{Re}}\nabla^2 u\,, \tag{2}$$

$$v_t + (uv)_x + (v^2)_y = -p_y + \frac{1}{\mathrm{Re}}\nabla^2 v\,, \tag{3}$$

which refers to the rectangular system of coordinates x, y; the symbol t denotes time; the subscripts stand for partial derivatives with respect to the corresponding independent variables, whilst the symbol ∇ denotes the Hamilton operator. The system of equations consists of the dimensionless forms of continuity Eq. (1) and the Navier – Stokes Eqs (2), (3) with three unknown functions: fluid velocity components $u(x, y, t)$, $v(x, y, t)$ in the x, y-direction, respectively and pressure $p(x, y, t)$. The symbol $\mathrm{Re} = \frac{\rho V L}{\mu}$, including density ρ, dynamic viscosity μ together with velocity V and length L scales, denotes the Reynolds number. Many authors refer to the Eqs (1)−(3) as the Navier − Stokes equations.

The initial conditions consist of prescribing u and v. The boundary conditions can be of several types: prescribed velocity components, vanishing normal derivatives of velocity components, or prescribed stress vector components. The pressure can be determined by prescribing the value at one spatial point.

The method for determination of unsteady, plane flow of viscous incompressible fluids is based on some method of elimination of pressure from the system of the Navier − Stokes equations by means of integration. Consequently the order of resulting system of equations is not increased in comparison with the original one and there is no need for posing the additional, «artificial» boundary conditions, which do not exist in the original problem.

Both velocity components and pressure are univalent functions of the independent variables and any fixed contour integrals of total differentials of these functions must vanish at any instant. For pressure it can be expressed in the following form:

$$\int_{\Gamma} (p_x dx + p_y dy) = 0 \,, \tag{4}$$

where Γ denotes any closed contour under consideration.

The derivatives p_x and p_y can be obtained from (2) and (3) and after substitution into (4) this relation yields:

$$\int_{\Gamma} \left[\frac{1}{\mathrm{Re}} \Delta u - u_t - (u^2)_x - (vu)_y \right] dx + \\ + \left[\frac{1}{\mathrm{Re}} \Delta v - v_t - (uv)_x - (v^2)_y \right] dy = 0 \,. \tag{5}$$

The Eq. (5) and the continuity Eq. (1), partially differentiated with respect to time:

$$(u_t)_x + (v_t)_y = 0 \,, \tag{6}$$

do not contain pressure. These two equations can be applied to determine the velocity components $u(x, y, t)$ and $v(x, y, t)$. It can be done in a number of ways, depending on accepted contour of integration and the type of temporal and spatial discretization.

After determination of the velocity components in the nodes of the mesh at any instant, pressure can be obtained from the integral:

$$p(x, y, t) = p(x_0, y_0, t) + \int_{x_0, y_0}^{x, y} [\ldots] dx + [\ldots] dy \,, \tag{7}$$

where t denotes time and expressions in brackets are identical with those in (5). The symbol $p(x_{0,} y_{0,} t)$ denotes a known function of time at the fixed point (x_0, y_0).

The system of Eqs (5) and (6), serving the determination of velocity field, after discretization has a form of the system of the first order ordinary differential equations. It means that the problem of the determination of viscous incompressible flow described by the system of partial differential equations has been transformed to equivalent initial problem for a set of the first order ordinary equations. The initial problem so defined can be solved by means of different ways. The initial and boundary conditions are the same as in the original problem, because no differentiation has been used in described transformation of the problem.

3. Discretization of the problem. The final system of the first order ordinary differential equations depends on many factors. The main of them are: the shape of the domain of solution, the shape of the contour of integration Γ, the applied method of integration in (5), the applied type of discrete approximation of the Eq. (6) and the method of time discretization. Originally in [18] a finite difference scheme on non-staggered grid was applied to the driven cavity problem. There were checked different paths of integration in (5), and approximating formulas for the integral, the first and the second derivatives of velocity components. For the solution of the final system of ordinary differential equations the fourth order Runge – Kutta method was applied.

Later in [12] the method was developed to more complicated domains of solution and non-uniform, stretched and compressed in certain subdomains of solution, computational meshes. Then in [13] the method was testing on the flow around a given contour which exterior was transformed onto rectangular domain of solution.

In the present paper the method has been modified and developed on staggered grid, where velocity components are unknown at different spatial locations. The non-typical staggered grid, shown in Fig. 1, was applied in this case. The ($\rightarrow$) denotes u points, ($\uparrow$) denotes v points, whereas ($\bullet$) denotes (i, j) points of the «main» grid, around which the contour integral of the total differential of pressure was computed. Nodes of this main mesh are defined by the values

$$x_{i,j},\ y_{i,j}, \qquad i \in [0, M+1], \qquad j \in [0, N+1],$$

of the spatial variables in rectangular domain. Then the discretized unknown velocity components are functions of time only and are associated with points (i, j) in such a manner:

$$u(x_i, y_{j+1/2}; t) = u_{i,j+1/2}(t), \qquad v(x_{i+1/2}, y_j; t) = v_{i+1/2,j}(t).$$

The u and v points are located in the middle between the neighboring points of main grid as a rule. That is even though one of these points is a boundary point. The grid of points is generally non-uniform in both directions. Mesh spacings Δx_i and Δy_j are defined in Fig. 2.

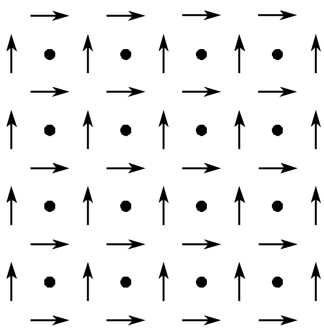

Fig. 1

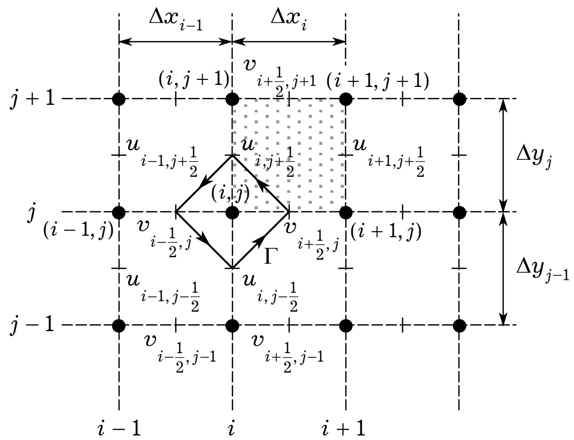

Fig. 2

Some cells in applied staggered grid are presented in Fig. 2. The (i, j) points are the points around which the integral of total differential of the pressure is computed, as shown in this figure. The path of integration has a shape of rhomb, and leads between u and v points surrounding the (i,j) point. Integration by means of the trapezoidal rule applied to (5), yields

$$-h\dot{u}_{i,j-1/2}+h\dot{u}_{i,j+1/2}+\dot{v}_{i-1/2,j}-\dot{v}_{i+1/2,j}=w_{i,j}\,, \tag{8}$$

where dots over velocity components indicate differentiation with respect to time, and

$$h=\frac{\Delta x_i+\Delta x_{i-1}}{\Delta y_j+\Delta y_{j-1}}\,, \tag{9}$$

$$w_{i,j}=h\left[G_{i,j+1/2}-G_{i,j-1/2}\right]+H_{i-1/2,j}-H_{i+1/2,j}\,, \tag{10}$$

and

$$G=\frac{1}{\mathrm{Re}}\nabla^2 u-(u^2)_x-(uv)_y\,, \tag{11}$$

$$H=\frac{1}{\mathrm{Re}}\nabla^2 v-(uv)_x-(v^2)_y\,. \tag{12}$$

The Eq. (8) represents the system of MN ordinary differential equations of the first order with unknown functions u and v, which number in rectangular domain equals $2MN+M+N$. The lacking $MN+M+N$ equations must be obtained from the relation (6) expressing the continuity equation partially differentiated with respect to time. This can be done by using the finite differences method as in previous works [12], but in this case more suitable is application of the continuity equation, partially differentiated with respect to time, in its integral form to the subdomain, illustrated in Fig. 2 as the dotted cell lying between (i,j), $(i+1,j)$, $(i+1,j+1)$ and $(i,j+1)$ nodes of the main grid. This operation yields the relation

$$\Delta y_j\left[\dot{u}_{i+1,j+1/2}-\dot{u}_{i,j+1/2}\right]+\Delta x_i\left[\dot{v}_{i+1/2,j+1}-\dot{v}_{i+1/2,j}\right]=0\,. \tag{13}$$

The subsystem (13) contains MN lacking equations written for all dotted cells associated with the node (i,j) as shown in Fig. 2. The remaining $M+N$ equations are obtained in the same way by applying the continuity equation (differentiated with respect to time) to the M subdomains (dotted in Fig. 3), lying near the boundary $j=0$, and to the N subdomains (dashed in Fig. 3), posed near the boundary $i=0$. The $M+N$ lacking equation are given by formulas:

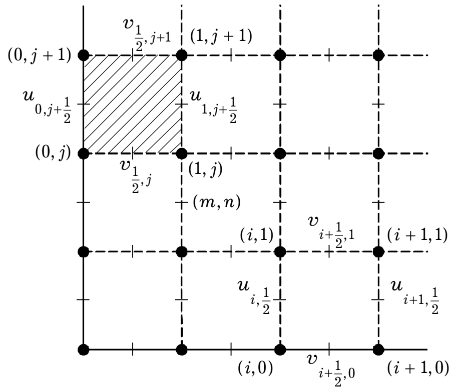

Fig. 3

$$\Delta y_0\left[\dot{u}_{i+1,1/2}-\dot{u}_{i,1/2}\right]+\Delta x_i\left[\dot{v}_{i+1/2,1}-\dot{v}_{i+1/2,0}\right]=0\,, \tag{14}$$

$$\Delta y_j\left[\dot{u}_{1,j+1/2}-\dot{u}_{0,j+1/2}\right]+\Delta x_0\left[\dot{v}_{1/2,j+1}-\dot{v}_{1/2,j}\right]=0\,. \tag{15}$$

Two cells being located at bottom-left and upper-left corners of the rectangular domain bring into Eqs (14) and (15) a slight modification in order to satisfy the mass conservation law in these subdomains.

The Eqs (8), (13), (14) and (15) form a linear system of $2MN+M+N$ equations serving the determination of the first derivatives of velocity components with respect to time.

The right-hand sides in (8), given by formula (10), contain the first and the second derivatives of velocity components. They may be approximated in different ways. In this paper they were obtained by application of the theory of cubic splines.

Let $L_{m,n} = \left.\frac{\partial^2 u}{\partial x^2}\right|_{m,n}$ and $l_{m,n} = \left.\frac{\partial u}{\partial x}\right|_{m,n}$ denote u derivatives with respect to x direction (n is kept constant) at its (m,n) location, shown in Fig. 3. Following the cubic spline theory [15] the derivatives are computed from equations for non-uniform grid:

$$\Delta x_{m-1} L_{m-1,n} + 2(\Delta x_{m-1} + \Delta x_m) L_{m,n} + \Delta x_m L_{m+1,n} = \\ = 6\left(\frac{u_{m+1,n} - u_{m,n}}{\Delta x_m} - \frac{u_{m,n} - u_{m-1,n}}{\Delta x_{m-1}}\right), \tag{16}$$

$$\frac{1}{\Delta x_{m-1}} l_{m-1,n} + 2\left(\frac{1}{\Delta x_{m-1}} + \frac{1}{\Delta x_m}\right) l_{m,n} + \frac{1}{\Delta x_m} l_{m+1,n} = \\ = 3\left(\frac{u_{m,n} - u_{m-1,n}}{\Delta x_{m-1}^2} + \frac{u_{m+1,n} - u_{m,n}}{\Delta x_m^2}\right), \tag{17}$$

where $m = 1,\ldots,M$ and $\Delta x_m = x_{m+1} - x_m$. The values of derivatives on the boundaries $m = 0$ and $m = M+1$ are computed by formulas stemming from the finite difference method for non-uniform grid. For example for the point $m = 0$ on the left boundary of the domain it is:

$$l_{0,n} = \left(\frac{\partial u}{\partial x}\right)_{0,n} = -\frac{2\Delta x_0 + \Delta x_1}{\Delta x_0(\Delta x_1 + \Delta x_1)} u_{0,n} + \\ + \frac{\Delta x_0 + \Delta x_1}{\Delta x_0 \Delta x_1} u_{1,n} - \frac{\Delta x_0}{\Delta x_1(\Delta x_0 + \Delta x_1)} u_{2,n}, \tag{18}$$

$$L_{0,n} = \left(\frac{\partial^2 u}{\partial x^2}\right)_{0,n} = \frac{2}{(\Delta x_0 + \Delta x_1)^2}\left[u_{2,n} - u_{0,n} - \right. \\ \left. - \frac{\Delta x_0}{\Delta x_1}(u_{0,n} - u_{2,n})\right] - 2\frac{u_{1,n} - u_{0,n}}{\Delta x_0 \Delta x_1}. \tag{19}$$

The derivatives for the right-hand side boundary can be computed from analogous formulas. The sets of formulas for the derivatives of u component with respect to y direction and for the derivatives of v component with respect to both directions can be obtained from (16)–(19) by proper replacement of symbols in all the relations.

The derivatives of the product of velocity components uv with respect to x and y directions were determined in the same way, but after appropriate averaging used in order to obtain the lacking values. It can be easily seen in Fig. 2 that in order to get the value of one component (u or v) in location of the second (v or u) the four surrounding values have to be used. The assumption of linear interpolation between nodes leads to the formulas presented below.

Resulting u component at $(i-1/2, j)$ point of v can be computed according following formula:

$$u_{i-1/2,j} = \frac{1}{\Delta y_{j-1} + \Delta y_j}(\Delta y_{j-1}\, u_{i-1/2,j+1/2} + \Delta y_j\, u_{i-1/2,j-1/2}), \tag{20}$$

where

$$u_{i-1/2,j+1/2} = \frac{1}{2}(u_{i-1,j+1/2} + u_{i,j+1/2}), \tag{21}$$

$$u_{i-1/2,j-1/2} = \frac{1}{2}(u_{i-1,j-1/2} + u_{i,j-1/2}). \tag{22}$$

The set of formulas for obtaining v component at $(i, j-1/2)$ node of u is quite analogues to (20)–(22):

$$v_{i,j-1/2} = \frac{1}{2}(v_{i,j} + v_{i,j-1}),$$

where

$$v_{i,j} = \frac{1}{\Delta x_{i-1} + \Delta x_i}(\Delta x_{i-1} v_{i+1/2,j} + \Delta x_i v_{i-1/2,j}),$$

$$v_{i,j-1} = \frac{1}{\Delta x_{i-1} + \Delta x_i}(\Delta x_{i-1}\, v_{i+1/2,j-1} + \Delta x_i\, v_{i-1/2,j-1}).$$

The Eqs (8), (13), (14) and (15) serving determination of the velocity components have to be completed by the appropriate initial and boundary conditions which follow from the initial and boundary conditions for original problem described by Eqs (1)–(3) and there is no need for creation of additional boundary conditions.

In the present paper the initial conditions express assumption that the motion of the fluid starts from rest. The boundary conditions express impermeability of the solid walls of the domain and the no-slip property of the fluid. It means that the velocity components at all nodes lying on the boundary are known and equal to the values of the solid walls.

The system of Eqs (8), (13), (14) and (15) can be rewritten in a matrix form:

$$A\dot{Y} = W. \tag{23}$$

The vector $\dot{Y}$ as well as W denote:

$$Y_k = u_{ij}(t), \qquad k := 1,\dots, MN,$$

$$Y_k = v_{ij}(t), \qquad k := MN+1,\dots,2MN,$$

$$Y_k = u_{ij}(t), \qquad k := 2MN+1,\dots,2MN+M,$$

$$Y_k = v_{ij}(t), \qquad k := 2MN+M+1,\dots,2MN+M+N,$$

$$W_k = w_{ij}(t), \qquad k := 1,\dots, MN,$$

$$W_k = 0, \qquad k := MN+1,\dots,2MN+M+N.$$

By means of the inverse matrix A^{-1} the system (23) can be presented in the form:

$$\dot{Y} = A^{-1}W. \tag{24}$$

The inversion of the matrix A can be performed only once at the beginning of the computation, because its elements are constant and the solution for time derivatives of velocity components can be obtained directly from (24). However, this simplest approach could be possible only on supercomputers.

In this paper a memory saving iterative method had to be applied. The matrix A of the equation (23) was transformed by multiplying by the transpose $A^\top$ in order to yield a symmetric and positive definite matrix of coefficients B:

$$B\dot{Y} = A^\top W, \tag{25}$$

where

$$B = A^\top A,$$

and a method of conjugate gradients has been applied in order to solve the Eq. (25) with respect to the derivatives of velocity components. Then the Runge – Kutta method of the fourth order was applied for integration of the system (25). This is a fully explicit method and right-hand sides in (25) were obtained from values computed at the previous time level.

After determination of velocity components from the system of Eqs (25), pressure can be computed at any point by means of the integral (7). The method and the path of integration should follow from the applied type of discretization of Eq. (5). In this paper the integration of the total differential of the pressure was realized by means of trapezoidal rule along the sides of rhombus as shown in Fig. 2. It means that in this case the path of integration ought to lead from the location of u to v by turns like along the sides of rhombus. An example of two possible paths of integration, whilst determining pressure at the point $(i+1/2, j)$, is shown in Fig. 4. Pressure at the point (x_0, y_0) is assumed to be known. Consequently the grid for obtaining pressure is twice dense as each grid for velocity components.

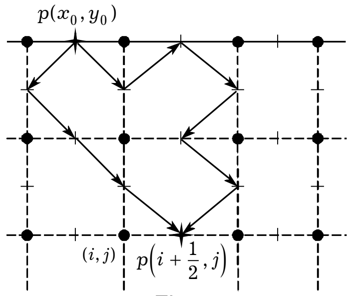

Fig. 4

4. Numerical results and conclusions. The viscous incompressible flow in a square cavity in which one wall moves with the known velocity is the well-known model problem for testing and evaluating numerical techniques. The domain of solution, considered in the present paper, is shown in Fig. 5. The fluid flow has been generated by the motion of the «bottom» wall.

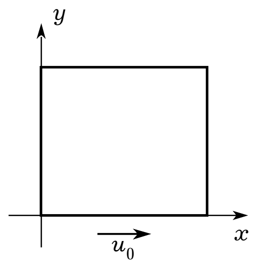

Fig. 5

The system of ordinary differential equations (25), governing the flow in the driven cavity, has to be completed by the proper initial and boundary conditions.

At the initial time:

$$t = 0: \qquad u(x, y, 0) = v(x, y, 0) = 0,$$

and the «bottom» wall suddenly begins to move with the known velocity:

$$u_0 = 1, \qquad t \geq 0.$$

The bottom left and right corners of the cavity, where moving wall is in contact with being at rest, are the singular points. In this paper the singularity has been omitted by assuming zero velocity at these points.

The computational mesh has M points in x direction and N in y. The boundary conditions on the non-moving walls of the cavity can be formulated as

$$u_{i,N+1} = v_{i,N+1} = 0, \qquad i = 0, \ldots, M+1,$$
$$u_{0,j} = v_{0,j} = u_{M+1,j} = 0, \qquad j = 0, \ldots, N+1,$$

and on the bottom wall

$$u_{i,0} = u_0(t) = 1, \qquad j = 1, \ldots, M.$$

Pressure was assumed to be zero in the middle of the upper wall of the cavity and at any instant and at all nodes of the grid was computed with respect to this point.

The viscous incompressible flow in the driven cavity was computed following the algorithm described in the previous section for three Reynolds numbers $\mathrm{Re} = 10, 100$ and 400 and compared with the known results [3–5, 7]. The Reynolds number was related to the side of the cavity and velocity of the moving wall.

The direct results of calculations concern the fluid velocity components in nodes of their meshes and can be presented in graphical form only as their distributions along sections of the cavity.

The indirect results obtained after appropriate calculations of the direct results concern:

− velocity field vectors − they can be calculated and shown graphically after averaging of the velocity components; in the present paper averaging was performed at (i, j) points of the main grid (Fig. 2) in the following way:

$$u_{i,j} = \frac{1}{\Delta y_{j-1} + \Delta y_j} (\Delta y_{j-1}\, u_{i,j+1/2} + \Delta y_j\, u_{i,j-1/2}),$$

$$v_{i,j} = \frac{1}{\Delta x_{i-1} + \Delta x_i} (\Delta x_{i-1} v_{i+1/2,j} + \Delta x_i v_{i-1/2,j});$$

– isobars – pressure is treated as an indirect result, because it can be computed (in the way described above) after obtaining velocity components;

– sets of lines of constant vorticity – vorticity Ω was computed following its definition:

$$\Omega = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y},$$

where derivatives of the velocity components were approximated by formulas analogous to (16)–(19), gained from the cubic spline-theory;

– streamlines – the values of stream function Ψ were obtained from Poisson equation:

$$\nabla^2 \Psi = -\Omega$$

by means of the Liebmann method;

– position of the «center of the vortex», appearing in the cavity.

The computations were performed for the following data:

$\mathrm{Re} = 10$ and the uniform mesh $M = N = 30$,

$\mathrm{Re} = 100$ and the uniform mesh $M = N = 50$,

$\mathrm{Re} = 400$ and the uniform mesh $M = N = 50$,

and the constant velocity of the «bottom» wall $u_0(t) = 1$, for comparison with the results obtained in [3–5, 7].

Selected results are shown in Figs 6–12. The calculations were terminated when two succeeding results of integration differed less then by 10^{-6} and the steady state conditions of the flow in the driven cavity were attained.

Velocity profiles along vertical and horizontal lines through the center of a driven cavity at $\mathrm{Re} = 10$ and $t = 5.0$, and at $\mathrm{Re} = 100$ and $t = 5.0$ and at $\mathrm{Re} = 400$ and $t = 10.0$ are given in Figs 6–8 respectively.

Figs 6–8 show distributions of velocity components along midlines of the domain of solution, indicated by solid lines, and their comparisons with known results [4, 5, 7], given by discrete points. The distribution of the u component is done along vertical line through geometric center of the cavity, whilst the distribution of v-velocity is given along horizontal line through geometric center. The results obtained agree quite close with those obtained by other investigators considering the same problem.

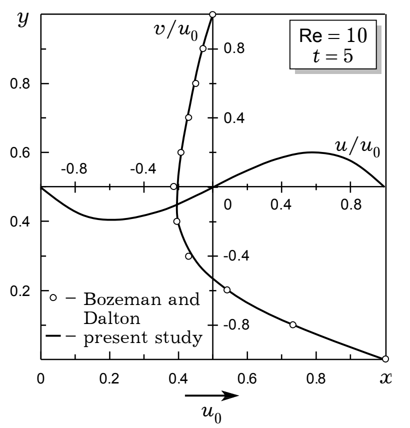

Fig. 6

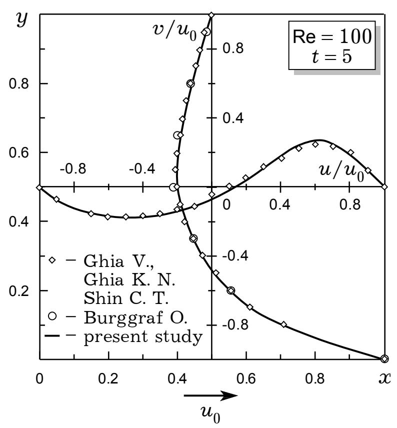

Fig. 7

An often compared quantity is the vorticity at the midpoint of the moving wall for $\mathrm{Re} = 100$. The present calculation gives the value $\Omega = 6.7394$. The values given in literature are: 6.57451 in [7] and 6.5641 in [3]. These results differ from the present value by 2.5% and 2.7% respectively.

The velocity extrema along the centerlines of the cavity at $\mathrm{Re} = 100$ yield results as follows: the minimum of the u-component equals $u_{\min} = -0.21116039$ at $y = 0.5784$, the minimum of the v component equals $v_{\min} = -0.1771529$ (its location $x = 0.2647$) and maximum is given by the value $v_{\max} = 0.26974899$ at the point $x = 0.81137$. The data borrowed from literature [3] are: $u_{\min} = -0.2140424$ at location $y = 0.5419$; $v_{\min} = -0.1795728$ at $x = 0.237$ and $v_{\max} = 0.253803$ at $x = 0.8104$. The obtained $u_{\max}$ differs from given in [3] by 1.3% in value and 6.7% in location. The $v_{\min}$ differs from the value in [3] by 1.3% and 11.8% in location. The $v_{\max}$ computed in the present paper differs from the value in [3] by 6.3% and 0.4% in location. These quantitative comparisons may indicate the fact that steady state conditions of the flow may not be attained and there must be involved a stronger condition for terminations of the calculations based on the value of the v component.

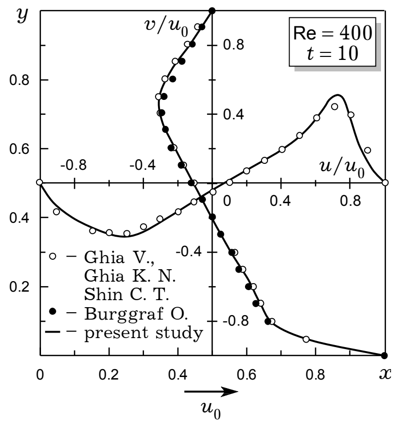

Fig. 8

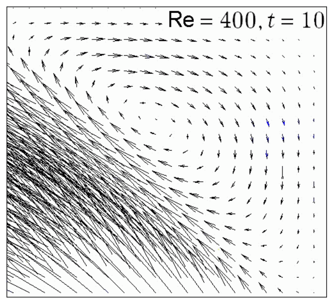

Fig. 9

The velocity fields are not presented in this paper in the sake of their difficult visualization except of one fragment (shown in Fig. 9) of the upper-right corner of the cavity in order to demonstrate the recirculating region and formation of the secondary vortex at $\mathrm{Re} = 400$ and $t = 10.0$.

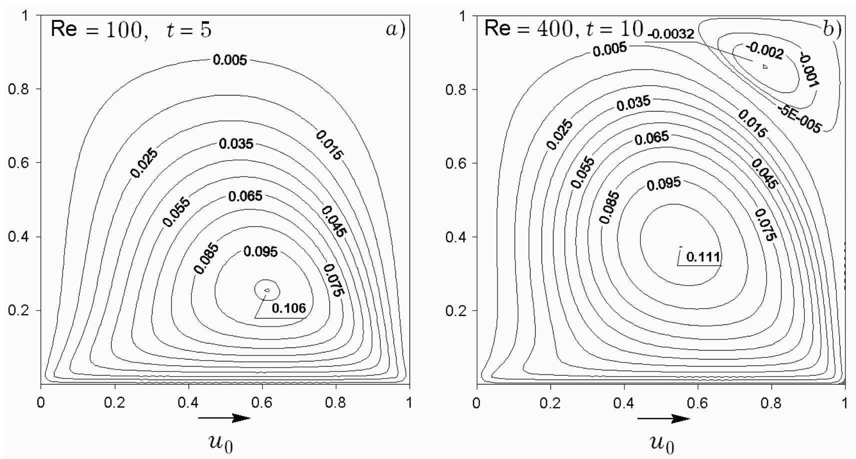

Fig. 10

Streamline patterns and stream function values for flow in a driven cavity for Reynolds numbers $\mathrm{Re} = 100$ and $\mathrm{Re} = 400$ are presented in Fig. 10. They show the development of the flow depending on the Re-number. With the grow of the Reynolds number the primary vortex moves in the direction of the trajectory of the bottom wall. Fig. 11 contains vorticity contours and vorticity values for flow in a driven cavity for $\mathrm{Re} = 100$ and $\mathrm{Re} = 400$, corresponding to the previous shown streamline patterns, whereas isobars for these flows are shown in Fig. 12. Streamlines, vorticity contours and isobars, presented in this paper, coincide closely with those published by other investigators [4, 5, 7].

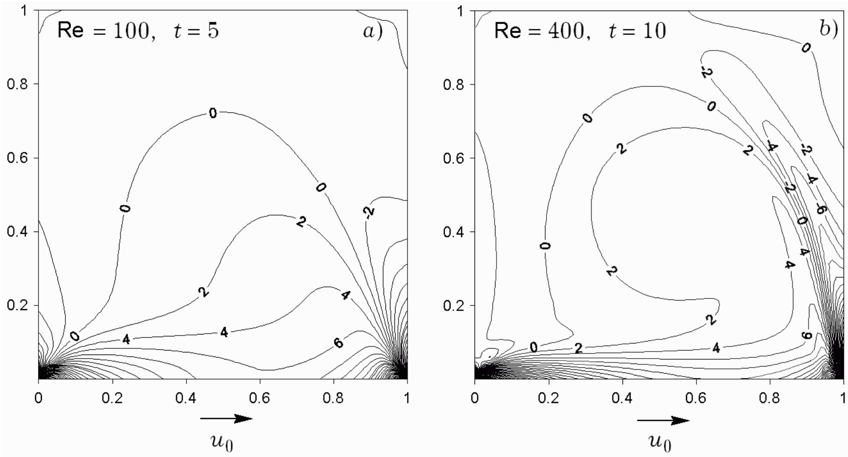

Fig. 11

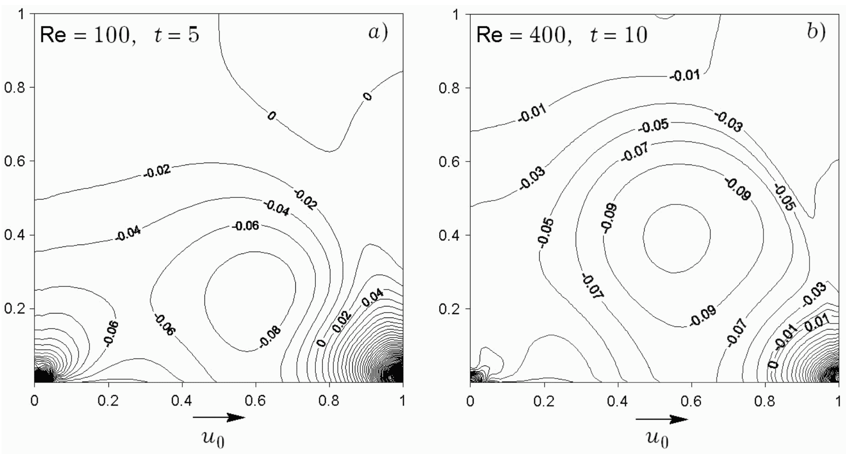

Fig. 12

Fig. 13 demonstrates an effect of Reynolds number on location of the primary vortex center appearing in the cavity and its comparison with result borrowed from [4].

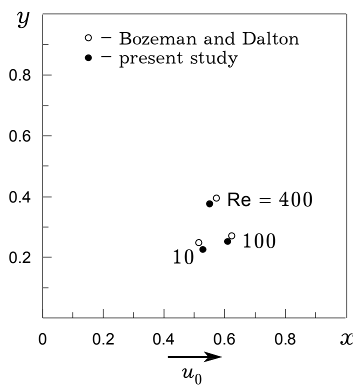

Fig. 13

The main feature of the method developed in this paper consists in the manner of elimination of pressure from the Navier – Stokes equations by means of integration. Consequently the order of the resulting system is not increased in comparison with the original one and there is no need to create the «artificial» boundary conditions, not existing in physical problem. The absence of any additional assumptions in the governing equations is an advantage in direct simulations. The second advantage is that the elements of the matrix of resulting system are constant. They can be computed only once, before integration.

The results gained for the flow in the driven cavity by application of presented method were compared with results obtained by other authors in order to verify the method. These comparisons indicate that the development of the method was performed properly.

The first computations of the flow in more complicated irregular domains reveal the great advantage of application of staggered grid. This will be the subject of further investigations.

1. *Ashrafian A., Andersson H. I., Manhart M.* DNS of turbulent flow in a rod-roughened channel // Int. J. of Heat and Fluid Flow. – 2004. – **25.** – P. 373–383.
2. *Auteri F., Parolini M.* A mixed-basis spectral projection method // J. Comput. Phys. – 2002. – **175.** – P. 1–23.
3. *Botella O., Peyret R.* Benchmark spectral results on the lid-driven cavity flow // Computers & Fluids. –1998. – **27,** No. 4. – P. 421–433.
4. *Bozeman J. D., Dalton C.* Numerical study of viscous flow in a cavity // J. Comput. Phys. – 1973. – **12.** – P. 348–363
5. *Burggraf O. R.* Analytical and numerical studies of the structure of steady separated flows // J. Fluid Mech. – 1966. – **24**. – P. 113–151.
6. *Friedrich R., Hüttl T. J., Manhart M., Wagner C.* Direct numerical simulation of incompressible turbulent flows // Computers & Fluids. – 2001. – **30.** – P. 555–579.
7. *Ghia V., Ghia K. N., Shin C. T.* High - Re solutions for incompressible flow using the Navier – Stokes equations and a multigrid method // J. Comput. Phys. – 1982. – **48.** – P. 387–411.
8. *Hoffman J., Johnson C.* A new approach to computational turbulence modeling // Comput. Methods Appl. Mech. Engng. – 2006. – **195.** – P. 2865–2880.
9. *Karniadakis G.* Simulating turbulence in complex geometries // Fluid Dynamics Research. – 1999. – **24**. – P. 343–362.
10. *Kasagi N.* Progress in direct numerical simulation of turbulent transport and its control // Int. J. of Heat and Fluid Flow. – 1998. – **19.** – P. 125–134.
11. *Kiris C., Kwak D.* Aspects of unsteady incompressible flow simulations // Computers & Fluids. – 2002. – **31. –** P. 627–638.
12. *Klonowska-Prosnak M. E., Kolodziejczyk M.* Tests on a new method for numerical solution to the Navier – Stokes equations. Part I: Plane unsteady flow in finite, rectangular domains // Int. J. Appl. Mech. and Eng. – 1999. – **4,** No. 2. – P. 311–340.
13. *Klonowska-Prosnak M. E., Kolodziejczyk M.* Tests on a new method for numerical solution to the Navier – Stokes equations. Part II: Plane unsteady flow around cur vilinear contours // Int. J. Appl. Mech. and Eng. – 2002. – **7,** No. 2. – P. 619–652.
14. *Koh Y.-M.* On boundary conditions and numerical methods for the unsteady incompressible Navier – Stokes equations // Computers & Fluids. – 2000. – **29.** – P. 791–812.
15. *Kosma Z.* Solution of the flow problems by means of splines / Zakład Narodowy im. Ossolińskich, Wyd-wo Polskiej Akademii Nauk. – 1993. – 253 p. (in Polish).
16. *Langtangen H. P., Mardal K. A., Winther R.* Numerical methods for incompressible viscous flow // Advances in Water Resources. – 2002. – **25.** – P. 1125–1146.
17. *Moin P., Mahesh K.* Direct numerical simulation: A tool in turbulence research // Annu. Rev. Fluid Mech. – 1998. – **30.** – P. 539–578.
18. *Prosnak W. J., Kosma Z. J.* On a new method for numerical solution of the Navier – Stokes equations // Acta Mech. – 1991. – **89.** – P. 45–63.
19. *Sengupta T. K., Dipankar A.* A comparative study of time advancement methods for solving Navier – Stokes equations // J. Sci. Comput. – 2004. – **21,** No. 2. – P. 225–250.
20. *Wesseling P., Segal A., Kassels C. G. M.* Computing flows on general three-dimentional non-smooth staggered grids // J. Comput. Physics. – 1998. – **149.** – P. 333–362.

ПРО ОДИН МЕТОД ЧИСЛОВОГО АНАЛІЗУ РІВНЯНЬ НАВ'Є – СТОКСА

Описано модифікацію і дальший розвиток методу, запропонованого в [18] *для визначення неусталеного плоского течіння в'язкої нестисливої рідини. Головною властивістю методу є виключення тиску з ключових рівнянь за допомогою інтегрування таким чином, щоб порядок отриманої системи не збільшувався порівняно з вихідною системою. Вказана операція приводить до задачі з початковими умовами для системи звичайних диференціальних рівнянь першого порядку. У статті метод модифіковано шляхом застосування зсунутої сітки для компонент швидкості. З метою перевірки методу наведено числові результати та їх порівняння з результатами, отриманими іншими авторами.*

ОБ ОДНОМ МЕТОДЕ ЧИСЛЕННОГО АНАЛИЗА УРАВНЕНИЙ НАВЬЕ – СТОКСА

Описана модификация и дальнейшее развитие метода, предложенного в [18] *для определения неустановившегося плоского течения вязкой несжимаемой жидкости. Главным свойством метода является исключение давления из ключевых уравнений путем интегрирования таким образом, чтобы порядок полученной системы не увеличивался по сравнению с исходной системой. Указанная операция приводит к задаче с начальными условиями для системы обыкновенных дифференциальных уравнений первого порядка. В статье метод модифицируется путем использования сдвинутой сетки для компонент скорости. Для проверки метода приводятся численные результаты и их сравнение с результатами, полученными другими авторами.*

Bialystok Univ. of Technology, Bialystok, Poland

Received
14.07.06