Simulation of the flapping–foil effects 
by vortex particle method

Henryk Kudela* and Tomasz Kozlowski †

Wrocław University of Technology, +48 90-370 Wrocław, Poland

It was presented the 2D numerical simulation of the various effect caused by the flapping motion of the foil. That effect concerned to the changing of the topology of the Karman vortex street for flow over the profile, caused by the pitching and plunging motion. It was also depicted results related to the hovering of profile. As a numerical method was used Vortex-In-Cell method. In order to fit the numerical grid to the solid body of the airfoil the conformal mapping was used. Description of the method and the results of tests was shown.

I. Introduction

The great interests in the low Reynolds number unsteady aerodynamics stems from the increasing interest in the micro air vehicles and from the will to understand the aerodynamics of insects and birds. The basic question birds flight is how does the birds generate enough lift and the thrust force to be able to perform remarkable maneuvers with rapid accelerations and decelerations. Flapping motion is a basic mode of locomotion in birds, insects and fishes. From the point of the fluid mechanics it is believed that all the phenomena that are related to the generation of lift and thrust forces are ruled by the dynamics of the vorticity. In incompressible flow the only place were the new vorticity can by generated is the solid boundary. Especially the behavior of the boundary layer creation and separation from the solid boundary are extremely important. To be able the study of the dynamics of evolution of the vorticity, the creation and the interaction with the solid wall of vortex structures for numerical study we choose the Vortex–In–Cell method (VIC). In the paper we presented the numerical results that are related to generation of the vorticity on moving objects and relations of that to the production of the thrust and lift forces on flapping and pitching ellipse. The importance of the vortex particle method lies in the possibility of the analyzing more easily and directly the vorticity field due to fact that in computation are used the particles that carry the information about the vorticity field. Attractive feature of the method is also the elimination of the difficulties connected with modeling of the inertial terms of Navier–Stokes equations. To check the relevance of two dimensional simplification and code validation we compare our computational results with both numerical data and experimental data obtained on robotic fly.

In present paper we demonstrate the unsteady effects of the lift force and propulsion production for the basic two dimensional flapping kinematics at Reynolds number $Re < 150$. Although flight in nature is naturally three-dimensional the two-dimensional simplification is widely used and seems to be appropriate to capture the essence of flapping flight hydrodynamics.8,22,23

*Professor, Department of Fluid Flow Modeling, Wąszyńskie 27, 50-370 Wrocław
†PhD candidate, Department of Fluid Flow Modeling, Wybrzeże Wyspiańskiego 27, 50-370 Wrocław
II. Vortex–In–Cell method

A. Governing equations

The Navier-Stokes equation in primitive variables with the coordinates fixed to the moving body has the form\(^{21}\)

\[
\frac{\partial \mathbf{u}}{\partial t} + \nabla \mathbf{u} \cdot \mathbf{u} = -\nabla p + \nu \Delta \mathbf{u} - \frac{d U_0}{dt} - \frac{d \Omega_0}{dt} \times \mathbf{r} + 2 \Omega_0 \times \mathbf{u} + \Omega_0 \times (\mathbf{u} \times \mathbf{r}),
\]

\[
\nabla \cdot \mathbf{u} = 0,
\]

where \(\Omega_0\) is is the angular and \(U_0\) is the translation velocity vector of the wing. The last three terms in equation (1) arise from non inertial coordinate system and denote non inertial force due to rotational acceleration, the Coriolis force and the centrifugal force respectively. Taking the curl, the equations (1), can be transform to the form

\[
\frac{\partial (\omega + 2 \Omega_0)}{\partial t} + (\nabla \omega) \cdot \mathbf{u} = \nu \Delta \omega,
\]

\[
\Delta \psi = -\omega,
\]

\[
\mathbf{u} = \nabla \times (0, 0, \psi).
\]

The equation (3) and (4) represent the vorticity transport equation in moving non inertial reference frame (for observer moving with the body). The stream function far from the body due to translational velocity \(U_0\), and the angular rotation \(\Omega_0 = \frac{d \alpha}{dt}\), can be write as

\[
\psi_\infty = U_0(y \cos(\alpha) - x \sin(\alpha)) - \frac{\Omega_0}{2} (x^2 + y^2).
\]

The vorticity field observed in the laboratory frame, differs only by a constant from the vorticity in the non inertial frame (see equation (3) and (4)), therefore, introduce the following change of variables

\[
\omega^* = \omega + 2 \Omega_0 \quad \text{and} \quad \psi^* = \psi - \psi_\infty.
\]

The detailed description of solution of the Helmholtz equations in moving reference frame can be found in.\(^8\)

B. Vortex in Cell method for conformal geometry

In order to better fit of numerical grid to solid boundary, we transform non-rectangular physical region \(x, y\) – variables to the rectangular one \((\xi, \eta)\), the conformal transformation was applied

\[
x + iy = \cosh(\xi + i \eta).
\]

In new variables \((\xi, \eta)\), taking into account (7), the equations (3), (4) have the form

\[
\frac{\partial \omega}{\partial t} + (\nabla \omega) \cdot \mathbf{u} = \frac{\nu}{J} \Delta \omega,
\]

\[
\Delta \Psi = -J \omega,
\]

where \(J\) denotes Jacobian of the conformal transformation

\[
J = \det \begin{vmatrix} x_{\xi} & x_{\eta} \\ y_{\xi} & y_{\eta} \end{vmatrix}.
\]

We have already omitted the star index. The velocity field \(u(u, v)\) is expressed by the formulas

\[
u = \frac{1}{J} \frac{\partial \psi}{\partial \eta}, \quad v = \frac{1}{J} \frac{\partial \psi}{\partial \xi}.
\]

The nullifying of the normal velocity component is obtained by setting \(\psi = \text{const.}\) on the wall. The no-slip condition is realized by introducing a proper portion of vorticity that ensure the \(\mathbf{u} \cdot \mathbf{s}^0 = 0\) were \(\mathbf{s}^0\) is tangential unit vector,\(^{30} \)\(^{35}\)
In the VIC method the continuous vorticity field is approximated with the discrete particles distribution. The flow region is covered with the numerical grid \( h = \Delta \eta = \Delta \xi \). In every grid node, the particles with circulation \( \Gamma_j = \int_A \omega d\xi d\eta \) are placed, where \( A = h^2 \), and

\[
\omega(\xi, \eta) = \sum_p \Gamma_p \delta(\xi - \xi_p) \delta(\eta - \eta_p).
\] (13)

The viscous splitting algorithm\(^4\) was used for solution of (9), (10). At first, the inviscid fluid motion equation was solved

\[
\frac{\partial \omega}{\partial t} + (\nabla \omega) \cdot \mathbf{u} = 0.
\] (14)

From (14) stems that vorticity is constant along the trajectory of the fluid particles. According to Helmholtz theorem,\(^2\) vortex particles are moving like material fluid particles. The differential equation (14) is replaced by the set of ordinary equations

\[
\frac{d\xi}{dt} = u, \quad \frac{d\eta}{dt} = v, \quad \xi(0, \alpha_1) = \alpha_1, \quad \eta(0, \alpha_2) = \alpha_2,
\] (15)

where \( \alpha = (\alpha_1, \alpha_2) \) means Lagrangian coordinate of fluid particles. The number of the particles are equal to the number of the grid nodes. The Lagrangian parameter \( \alpha \) takes in each time step the value \( \alpha_i, \alpha_j = (\xi_i, \eta_j) \).

The finite set of equations (15) was solved by fourth order Runge-Kutta method. The velocity field was obtained by solving Poisson equation (10) on the numerical grid and utilizing (12). The velocities of the particles that are found between the grid nodes were calculated by the interpolation formula

\[
u(\xi_p, \eta_p) = \sum_j l_j(\xi_p, \eta_p) u_j,
\] (16)

where \( l_j \) denotes two dimensional bilinear interpolation Lagrange base.

At second step the viscosity was taken into account. It was done by solving the diffusion equation

\[
\frac{\partial \omega}{\partial t} = \nu \frac{\partial^2 \omega}{\partial \xi^2} + \frac{\partial^2 \omega}{\partial \eta^2},
\] (17)

\[
\omega(\xi, \eta, 0) = \omega_0, \quad \omega|_{wall} = \omega_s,
\] (18)

where \( \omega_s \) was calculated on the basis of the Poisson equation (10). The no–slip condition \( u = 0 \), gives the vorticity on the wall \( \omega(0, j)_s = -\Psi_{\eta j}/J \). The value of \( \Psi_{\eta j} \) was calculated by Briley formula\(^25\)

\[
\omega (0, j)_s = \frac{1}{J} \frac{108 \Psi_{1,j} + 27 \Psi_{2,j} + 4 \Psi_{3,j}}{18 h^2} + O(h^4),
\] (19)

where \( h \) denotes the grid step, index 0 refers to the wall and index \( i, i = 1, 2, 3 \) to the distance \( ih \) from the wall.
Figure 2. Redistribution of the particle masses onto the neighboring grid nodes, a) for particles laying inside of the computational domain (at least one cell from the wall), b) for the particles in the vicinity of the wall.

After particles displacement according to the ordinary differential equations (15), the diffusion equation was solved on the grid. But before one have to pass the vorticity from the particles to the grid nodes, fig. 2. It was done, according to the formula

$$\omega_{ij} = \frac{1}{h^3} \sum_p \Gamma_p \varphi_h(\xi) \varphi_h(\eta),$$

where

$$\varphi_h(\xi) = \varphi \left( \frac{\xi - \xi_i}{h} \right), \quad \varphi_h(\eta) = \varphi \left( \frac{\eta - \eta_j}{h} \right).$$

(20)

Indexes p, i, j refer to vortex particles and grid nodes respectively and \(\varphi(\cdot)\) is the kernel of the interpolation function. Interpolation of particle masses onto grid nodes has the fundamental meaning for the precision of VIC method. In present work the redistribution process was done using Z-splines. The main advantage of these functions is easy constructions of high order symmetrical functions and also one-sided interpolation function to apply near the boundary. Four order interpolation kernel \(Z_4\) is identical with known in literature \(M4\) kernel, and has the form\(^5\)

$$\varphi(x) = \begin{cases} 
1 - \frac{5}{3}x^2 + \frac{3}{2}|x|^3 & \text{for } |x| < 1 \\
\frac{5}{3}(2 - |x|)^3(1 - |x|) & \text{for } 1 \leq |x| \leq 2. \\
0 & \text{for } |x| > 2
\end{cases}$$

(22)

For particles near the wall one-sided interpolation functions were used, derived according to the algorithm presented in\(^2\)

$$\varphi(x) = \begin{cases} 
1 + \frac{1}{2}x^2 - \frac{3}{2}|x| & \text{for } j = 0, |x| \leq 1 \\
-x^2 + \frac{3}{2}|x| & \text{for } j = 1, |x| \leq 1. \\
\frac{1}{2}x^2 - \frac{1}{2}|x| & \text{for } j = 2, |x| \leq 1
\end{cases}$$

(23)

Both interpolation kernels conserve three first moments\(^4\)

$$\sum_p x_p^\alpha \varphi \left( \frac{x_p - x}{h} \right) = x^\alpha, \quad \alpha = 0, 1, 2.$$  

(24)

After redistribution the diffusion equation (17) was solved on the numerical grid with alternating direction implicit (ADI) scheme\(^18\)

$$\omega^{n+\frac{1}{2}} = \omega^n + \frac{\Delta t}{2h^3} \left( \Lambda_{\xi \xi} \omega^n + \Lambda_{\eta \eta} \omega^{n+\frac{1}{2}} \right),$$

$$\omega^{n+1} = \omega^{n+\frac{1}{2}} + \frac{\Delta t}{2h^3} \left( \Lambda_{\xi \xi} \omega^{n+\frac{1}{2}} + \Lambda_{\eta \eta} \omega^{n+\frac{1}{2}} \right).$$

(25)

(26)

where \(\Lambda\) means of the three point central finite difference quotient, with respect to the variable that was put in lower index. The solution of the diffusion equation ends the calculations in the \(n\)-th time step of the Vortex–In–Cell method.
Table 1. Grid Parameters and time steps for flow past circular cylinder

<table>
<thead>
<tr>
<th>$Re$</th>
<th>$n_r$</th>
<th>$n_q$</th>
<th>$\Delta t$</th>
<th>$t_s$</th>
</tr>
</thead>
<tbody>
<tr>
<td>550</td>
<td>256</td>
<td>512</td>
<td>0.01s</td>
<td>0.2s</td>
</tr>
<tr>
<td>3000</td>
<td>512</td>
<td>1024</td>
<td>0.005s</td>
<td>0.3s</td>
</tr>
<tr>
<td>9500</td>
<td>2048</td>
<td>4096</td>
<td>0.002s</td>
<td>0.4s</td>
</tr>
</tbody>
</table>

C. Unbounded domain

Due to fact that we use the numerical grid, the domain of computation have to be finite. To establish far field boundary conditions for stream function, we used the method that was described in.\textsuperscript{1,20} That method takes advantage of the fact that the domain of non-zero vorticity around the solid body is limited to the small domain around the obstacle. In the far distance from the body where the vorticity is zero the asymptotic properties of Laplace equation and its representation by Fourier series is used. The detailed description of the obtaining the correct boundary value, applied in present work can be found in.\textsuperscript{11}

D. Tests of the method

1. Flow over circular cylinder

In order to check the VIC method and solution procedure we perform several comparison tests. At first, we check the above procedure for the far field boundary condition and the vortex particle ability to reproduce very fine vortex structures. The calculations were carried out for a well documented flow over a circular cylinder in wide regime of Reynolds number. The following conformal transformation was used

$$x + iy = \exp(r + i\theta).$$

With sudden, impulsive flow over the cylinder, one encounter the transition problem which has the consequences in oscillatory solutions in very initial time steps. To eliminate the transition problem a smooth start was performed. In the first calculation step, the vorticity equals zero and it was assumed a potential solution. The velocity at infinity of the flow, changes in the linear way from zero to $U_\infty$, in time interval $[0, t_s]$. Then we restarted the calculation using the obtained solution for $t = t_s$ as an initial condition for the vorticity field. In each time step that followed, on the external radius a proper boundary condition for stream function was obtain, as mentioned in section C.

Figure 3. Comparison of the streamlines evolution with the experimental data is presented, $Re = U D / \nu = 9500$; a) experimental data,\textsuperscript{19} b) streamlines from computations.

5 of 12

American Institute of Aeronautics and Astronautics
Calculations were carried out for a wide range of Reynolds number $Re = U_\infty 2R_1/\nu$. The parameter of the numerical grid, time steps and $t_s$ were presented in table 1. For comparison, drag force and drag coefficient were determined on the surface of the cylinder:

$$F_D = b\mu \int_0^{2\pi} \left( \omega - R \frac{\partial \theta}{\partial r} \right) \sin(\theta) Rd\theta,$$

$$C_D = \frac{F_D}{\rho u_\infty^2 R b},$$

where $\mu$ is a dynamical coefficient of fluid viscosity, $\mu = \rho \nu$, $\rho$ is a density of fluid, $b$ denotes unit length of cylinder and $R$ denotes radial of cylinder. We assume $R = 1, \rho = 1$.

![Figure 4. Comparison of time evolution of drag coefficient with work](image)

In figure 4 comparison of the calculated drag coefficient with the results of the work was presented. Some discrepancies were observed for times $t < 1$. It can be related to the problem how one managed the transition problem at the beginning of calculations. In the present work, the initial condition for vorticity was obtained by smooth start. The noted differences in the beginning do not influence the results in a significant way in the later time steps. For Reynolds numbers $Re = 9500$ the correct reproduction of vortex structures in the vicinity of the wall, requires a very dense numerical mesh (see table 1). The ability of the reproduction of the fine vortex structure on the cylinder is clearly visible. The numerical results for Reynolds number $Re = 9500$ were presented in the figures 3. Qualitatively, a very good agreements was obtained with the visualization data that were published in.

2. Flow past elliptic cylinder

The next test is related to the more complicated shape of the solid body. We investigated the flow past a thin ellipse perpendicular to the flow direction. The calculations were carried out in elliptic coordinates, which was transformed into a regular Cartesian mesh using conformal transformations

$$x + iy = \cosh(\xi + i\eta).$$

The calculations were carried out for the following parameters: ellipse thickness $e = 0.25$, ellipse length $L = 2$, Reynolds number $Re = 10000$. On the sharp ends of the ellipse, significant vorticity are generated. A dense grid in this region (figure 1), allows to reproduce the flow phenomena generated in this area, however a small time step is required, $\Delta t = 0.005$. At high Reynolds number, along the line that forms main vortex structure, smaller, secondary vortex structures appear (figure 5).

Numerical calculations carried out in reflect qualitatively the phenomenon discussed. It is considered, that the frequency of the small, secondary vortex is control by, the vortex which appears near the sharp edge of ellipse (corner vortex). After a sudden start, the corner vortex reaches a quasi-stationary state. The calculations were carried out for the same parameters as in. Vorticity generated at the end of the ellipse agrees very well with the results of. In figure 5 the experimental visualization from was presented. The similarity of the visualization data and numerical results is visible.

American Institute of Aeronautics and Astronautics
III. Dynamics of Karman vortex street behind the moving ellipse

It was found that the foil (the ellipse) that is subjected to the sinusoidal plunging and pitching (see eq. (31)-(33) below) in steady motion can inverse of topology in Karman vortex street.

\[ x(t) = Ut \]  \hspace{1cm} (31)
\[ y(t) = \frac{A_0}{2} \sin(2\pi ft) \]  \hspace{1cm} (32)
\[ \alpha(t) = \alpha_p + \alpha_0 \sin(2\pi ft) \]  \hspace{1cm} (33)

On the left side in figure 6 we see the classical vortex street and on the right the vortex street with inverse topology of vortices. It is due to plunging and pitching. Inversion of the topology of the vortices led to the production of propulsion force (compare the mean velocity profile behind of the ellipse). The increasing the frequency of the plunging may results on the deflection of the vortex street from the horizontal direction fig. 'i. This effect may be produced also by introduction of the pitching and plunging motion. Depending on the

Figure 6. Vortex street behind the foils that is in simple horizontal motion on the left; vortex street behind the foil that is subjected to the sinusoidal plunging on the right

Figure 7. Deflected wing wake; experiment\textsuperscript{14} - on the left, VIC simulation - on the right
amplitude and frequency of that motions \( A_0, \alpha_0, f \) (see equations (32)- (33)) the foil may experience the thrust or drag forces.

**IV. Simulation of normal hovering kinematics**

The main focus of this study is one of the simplest family of two-dimensional hovering motions. It was recognized the pattern of the tip wing motion of the bird and insect in hovering flight, recall the 8-figure shapes and may be described by the equations\(^{21}\)

\[
\begin{align*}
x(t) &= \frac{A_0}{2} \cos(2\pi ft) \\
\alpha(t) &= \alpha_0 + \alpha_m \sin(2\pi ft)
\end{align*}
\]

(34) (35)

where \( x(t) \) denote instantaneous position of the wing center, \( A_0 \) is the plunging amplitude, \( f \) is the frequency, \( \alpha(t) \) is the angle of attack measured relative to the horizontal line, \( \alpha_0 \) is the initial angle of attack and \( \alpha_m \) is the pitching amplitude. The translation and angular velocities are given by \( U_0 = \frac{dx}{dt} \) and \( \Omega_0 = \frac{d\alpha}{dt} \). The presented flapping kinematics is known in literature as normal hovering mode.\(^{16}\) On the right side in fig. 8, full stroke of the flapping wing according to equations (34), (35) is presented. The stroke is often divided on two sub-strokes: up-stroke and back-stroke. The time \( t \) was nondimensionalized by the frequency of the motion \( T = ft \) so that the \( T = 1 \) correspond to one full stroke of the wing motion.

The Reynolds number is related to maximum translational wing velocity, \( U_{\text{max}} = \pi f A_0 \) obtained from equation (34) and the chord of the profile \( c \)

\[
Re = \frac{U_{\text{max}} c}{\nu},
\]

(36)

where \( \nu \) denote kinematic viscosity coefficient of the fluid.

In the following computations flight parameter appropriate for fruit-flies\(^{16,21}\) were used: initial angle of attack \( \alpha_0 = \frac{\pi}{2} \), pitching amplitude \( \alpha_m = \frac{\pi}{4} \) and \( \frac{dc}{dt} = 2.8 \), frequency \( f = 0.25 \text{Hz} \), thickness of the elliptical wing \( c = 0.125 \), Reynolds number based of the maximum translational velocity \( Re = 75 \). We assumed homogeneous fluid with constant density \( \rho = 1 \). In fig 9 it was presented the sequence streamlines and

![Figure 8. Three dimensional hovering insect model with marked cross section of the wing, on the left. On the right simple two dimensional approximation of the hovering flight, known in literature as normal hovering mode](image)

field of the vorticity produced by the full up and back strokes of the profile. The evolution of the vorticity presents a typical pictures that one can find in many article.\(^{16,21,25}\) Resultants forces which act on the profiles depends on the distribution of the vorticity and its changing in time.\(^{27}\) In practice, the distribution of the vorticity strongly depend on the violent phenomena that take place in the boundary layer like eruption (burst separation) of boundary layer.\(^{13,17}\)
Figure 9. Vorticity contour plot and stream lines for third stroke of the flapping wing, $Re = 75$. The stream lines are plotted for coordinates fixed to the wing.

A. Computational details

The hydrodynamic forces acting on the wing, were calculated from the vorticity field by integration the pressure and viscous stress along the ellipse

$$\mathbf{F}_n = \nu \rho \int_C \frac{\partial \omega}{\partial \mathbf{n}} dC + \rho A_b \frac{dU_n}{dt},$$

$$\mathbf{F}_t = \nu \int_C \omega s^b dC,$$

where $\rho$ denote density of the fluid, $\mathbf{n}$ and $s^b$ is normal and tangential unit vector respectively and $A_b$ is the area of the body. The last term in the right side of equation (37) represent the inertial force of the fluid displaced by the profile. The forces obtained from equations (37), (38) were decomposed on to horizontal $F_x$ and vertical $F_y$ components, that correspond to lift and drag forces for case depicted in fig. 8. The force coefficient were calculated according to relations

$$C_x = \frac{F_x}{\frac{1}{2} \rho U_{\text{max}}^2 cb}, \quad C_y = \frac{F_y}{\frac{1}{2} \rho U_{\text{max}}^2 cb},$$

where $U_{\text{max}}$ is the maximum translational velocity, defined in the previous section and $b$ is the unit contractual length.
The calculations were carried on for flapping kinematics described by the equations (34), (35) for five complete strokes of the flapping motion, in nondimensionalized time range $T = (0, 5)$. For numerical solution, we use the elliptical mesh, given in fig. 1, with 296 grid nodes in radial direction and 128 grid nodes in azimuthal direction. In the following computations, the outer radius of the physical domain was $R_2 \approx 13$. In every time step, we perform the correction of the boundary condition for the stream function far from the body, as detailed in section C. The time step was set to $\Delta t = 0.005$.

In fig. 10 the comparison of the calculated lift force coefficient (green), with experimental (red) and numerical (blue) data published in\textsuperscript{20} is presented. In the experiment described in cited paper the lift force was measured at 70% of the wing span length. After two complete strokes, we achieve very good agreement with experimental measurements, which confirm the sense of the two dimensional approach for presented flapping kinematics.

V. Simulation of inclined hovering motion and free flapping flight

In the case of inclined hovering motion depicted on the left in fig. 11 we have\textsuperscript{21}

$$x(t) = \frac{A_0 \cos(\beta)}{2} \cos(2\pi ft),$$  \hspace{1cm} (40)

$$y(t) = \frac{A_0 \sin(\beta)}{2} \cos(2\pi ft).$$  \hspace{1cm} (41)

The angle of attack $\alpha$ of the wing is changed in time according to

$$\alpha(t) = \alpha_0 + \alpha_m \sin(2\pi f).$$  \hspace{1cm} (42)

The calculation parameters was the same as in work\textsuperscript{22}. We included this hovering model to the investigation of the free flapping flight. The free flapping motion was described by Newton’s second law $m(\ddot{x}, \ddot{y}) =$
\((0, -g) + \mathbf{F}_{\text{fluid}} + m\mathbf{a} = \int (-p\mathbf{n} + \tau)ds\), where \(\mathbf{F}_{\text{fluid}} = \int (p\mathbf{n} + \tau)ds\) and \(\mathbf{v}_0\) are calculated from equations (37) and (38). The numerical results are presented in the figures 12.

Figure 12. Free flapping flight; the trajectory - on the left, the vorticity contour plot - on the right

VI. Conclusion

Assumption of the two dimensionality about the flow simplified the investigation of the unsteady motion of the profiles. Despite of that simplification the 2D phenomena are strongly non-linear. This lead to interesting, an unexpected effects. All that phenomena, like changing of topology of the vortex behind of the profiles, the production of thrust and lift force are related to the dynamic of vorticity. Due to fact that vorticity plays so important role it seems that the vortex particle method is very good suited for study such phenomena. The importance of the vortex particle method lies in the possibility of the analyzing more easily and directly the vorticity field due to fact that in computation are used the particles that carry the information about the vorticity field. Attractive feature of the method is also the elimination of the difficulties connected with modeling of the inertial terms of Navier–Stokes equations.

References

19. Van Dake M. An Album of Fluid Motion, Stanford, California, 1982
24. Wang Z., J., Liu J. G., Childress S. Connection between corner vortices and shear layer instability in flow past an ellipse, Physics of Fluids 11, No 9, 1999