ON MESHLESS LAGRANGIAN VORTEX METHODS FOR TWO-DIMENSIONAL FLOW SIMULATION AT MODERATE AND HIGH REYNOLDS NUMBERS

YULIA IZMAILOVA & ILIA K. MARCHEVSKY
Bauman Moscow State Technical University, Russia

ABSTRACT

Vortex particle methods of computational fluid dynamics belong to the class of meshless Lagrangian particle methods, relying on integral representation of velocity field and boundary integral equations method. One of the main advantages of vortex particle methods is connected with the fact that they do not require to reconstruct or deform the mesh, which is important in coupled FSI problems. Vortex methods are widely used to estimate hydrodynamic loads acting on structural elements, especially for essentially unsteady non-potential flow regimes with intensive vortex generation. In a number of engineering applications involving modelling of long structures (bridges, pipelines, buildings), the only way to solve the problem is to use flat cross-section approach. In each cross-section, two-dimensional problem of flow simulation around an airfoil is considered. On one hand, modern models of vortex methods allow for a viscous medium simulating, and on the other hand, they are based on direct approximation of the diffusion term (the Laplacian operator in the Navier–Stokes equation). This makes it possible to correctly simulate the flow around airfoils at low Reynolds numbers. At moderate and high Reynolds numbers, simulation using vortex particle methods remains valid only for airfoils with sharp edges where flow separation takes place. For airfoils with smooth boundaries, the results of simulation do not correspond to experimental data. While the macroscopic flow remains two-dimensional, three-dimensional effects related to turbulence significantly influence the microscale behaviour. Numerical experiments have been performed; the scope of the viscous vortex domain method, implemented by the authors in original code (https://github.com/vortexmethods/VM2D), is estimated. A detailed study of the velocity field in a vortex wake behind a circular cylinder showed that the spatial and temporal spectra of turbulence kinetic energy align well with the Kolmogorov–Obukhov law, but with an exponent of (−3) instead of (−5/3). This observation is consistent with theoretical results for plane flows where three-dimensional effects do not appear at all. The possible ways for developing of two-dimensional models of vortex methods towards including of LES-type turbulence models are discussed.

Keywords: vortex particle methods, flow simulation around airfoil, high Reynolds numbers, turbulence spectrum, Kolmogorov–Obukhov law.

1 INTRODUCTION

One of the most important problems in aerohydrodynamics is the simulating the interaction between various structural elements and the flow. In engineering applications, the flow of the medium itself is typically not of independent interest. Instead, it is essential to calculate the loads acting on the streamlined airfoil(s), as the magnitude and nature of these loads determine the behaviour of movable or deformable structures. Such problems arise when simulating the flow around structural components, particularly in long-span bridges, various cable structures, overhead power lines, underwater pipelines and hoses, components of aircraft structures, heat exchanger tubes of power plants, etc.

Currently, there are many different approaches to solving computational fluid dynamics problems. The most common methods belong to the mesh-based class, including the finite difference method, the finite volume method, the finite element method and their various modifications. Note that in the flow simulation problems these methods allow for the accurate consideration of many factors relevant to the physical processes being studied. This results

WIT Transactions on Engineering Sciences, Vol 136,
www.witpress.com, ISSN 1743-3533 (on-line)
doi:10.2495/BE470101

in a broad range of applicability for mesh-based methods, although the computational cost can be rather high, especially when one deals with unsteady flow simulation around moving/deformable airfoils.

Another class, namely Lagrangian meshless methods, includes vortex particle methods [1]–[3], which scope is limited to incompressible flows, but this simplification seems acceptable for many technical applications. In vortex particle methods, vorticity is considered as the primary computational variable. Known vorticity distribution allows for reconstructing the velocity field at any point in the medium using the generalised Biot–Savart law [4], and the pressure field using analogues of the Bernoulli and Cauchy–Lagrange integrals [5]. Vorticity is generated on the airfoil contour line, thereby satisfying the no-slip boundary condition. Note, when simulating external flow using vortex particle methods, there is no need to bound the flow domain, since the boundary condition of perturbations decay at infinity is automatically and exactly satisfied.

Nowadays, vortex particle methods are actively developing and there are modifications that allow for considering three-dimensional and two-dimensional flows. The corresponding algorithms are incommensurate in numerical complexity, but when solving many practical problems, bodies in the flow have close to cylindrical shape with large elongation. Therefore, instead of simulating the entire spatial flow, it is often sufficient to consider one or several two-dimensional flow problems around individual cross-sections (flat cross-sections method [6]). Algorithms designed for problems that require simulation of the flow around flat airfoils remain highly relevant; their main 'competitive advantage' is low computational complexity, and therefore the ability to perform calculations rapidly.

Nevertheless, due to the specific features of vortex particle methods, there are currently only a few their software implementations that are freely available to researchers and engineers (optimally with open source code). One of them is the VM2D code [7], freely available from the github repository at https://github.com/vortexmethods/VM2D. This code is based on the Viscous vortex domains method [2], [8] and some original author's modifications, mainly related to the use of high-precision T-schemes [9] for solving boundary integral equations, as well as some other algorithms. The VM2D code can be used to simulate two-dimensional flows; it allows for solving a wide range of problems, including coupled FSI ones, using the capabilities of modern multiprocessor computers of various architectures (computations on CPU and GPU are available). The aim of this paper is to consider the scope of existing algorithms of vortex particle methods based on the experience of using the VM2D code and to discuss some directions for the future development of this code in particular and vortex methods for modelling 2D flows in general.

2 NUMERICAL SIMULATION OF VISCOUS FLOWS IN VM2D CODE

Note that the methods implemented in the VM2D code were applied to the flow simulation around different types of airfoils and for solving coupled FSI problems. Specifically, in Kuzmina et al. [10] the application of vortex particle methods for the simulation of various two-dimensional flows was explored. That paper shows wide range of applicability of the method for different types of airfoils (circular cylinders; airfoils with corner points; wing airfoils).

Some other results obtained using the VM2D code are described in Marchevsky et al. [7], where a comprehensive review of the code's capabilities for simulating two-dimensional incompressible flows using vortex particle methods is presented. That paper highlights the flexibility of VM2D and its applicability to various hydrodynamic tasks (flow simulation around the Savonius rotor, steady-state regime in the channel with backward-facing step, Blasius flow). Also, in Marchevsky et al. [7], Marchevskii et al. [9] and Kuzmina et al. [10]

one can see the accuracy analysis between the simulation results and the reference results, especially in tabular and plot form, together with the error dependence on the level of discretisation.

In the present study let us focus on the issue of accounting for viscosity in two-dimensional flow simulation. Historically, modern versions of vortex particle methods for solving two-dimensional problems trace back to the discrete vortex method [11]–[14] (in some publications, both the term 'discrete vortex method' and 'lumped vortex method' are used). Emerging in the 1950s, this method appears to be one of the earliest methods of computational fluid dynamics. It provides numerical modelling of the flow of an ideal (non-viscous) fluid that is described by the Euler equations; in such a formulation, it is possible to simulate potential or, at least, irrotational flows. Note that it is possible to reproduce much more complex unsteady separated flow regimes around airfoils – this is especially relevant for thin airfoils (plates), wing airfoils and, in general, for airfoils with sharp edges and corner points. The position of the vortex sheet separation is set at the ends of thin plate (one or both, Fig. 1) or at the corner points; the descending vortex sheet that forms the vortex wake, is considered as a thin surface of tangent discontinuity of velocity.

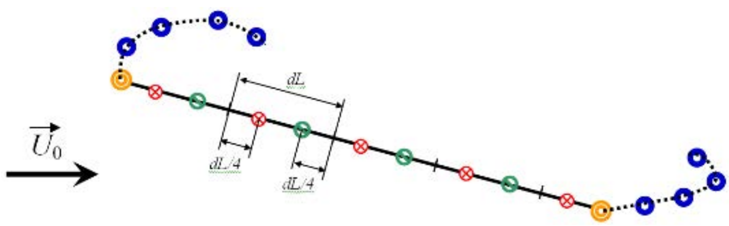

Figure 1: Numerical scheme for flow simulation around a thin plate using the method of discrete vortices with vortex sheets shedding from both ends. Plate splitting into panel is shown as well as positions of control points (red), attached vortices (green), free vortices (orange) and vortex sheets modelled as thin discontinuity surfaces with point vortices.

Although such a model is rather inaccurate, it allows for solving a number of problems of practical interest. Its fundamental issue is the inability to solve problems related to the flow simulation around smooth airfoils, where the position of the flow separation point cannot be specified a priori. It is also impossible to simulate flows at low Reynolds numbers, which are mainly influenced by viscous forces.

The development of vortex particle methods has mainly followed the strategy of taking into account the influence of viscosity when solving the Navier–Stokes equations. Today, the following approaches are known: the random walk method, developed in the 1970s [15], the particle strength exchange (PSE) method, proposed in 1989 [16], the core spreading method, first proposed in 1973 [17] and later developed in 1996 [18], the diffusive velocity method, known in several modifications, and some others. Among these is the method of viscous vortex domains [2], [8], which is implemented in the VM2D code and appears to offer several advantages over the other methods. Numerical experiments using the VM2D code show that flows with a predominance of viscous effects can be simulated quite well: velocity profiles in the Poiseuille flow in a channel, in the Couette flow between two coaxial rotating cylinders (Fig. 2), and in the boundary layer on a thin plate in the Blasius problem (Fig. 3) are reproduced quite accurately, even with a relatively small number of vortex particles.

WIT Transactions on Engineering Sciences, Vol 136, © 2024 WIT Press
www.witpress.com, ISSN 1743-3533 (on-line)

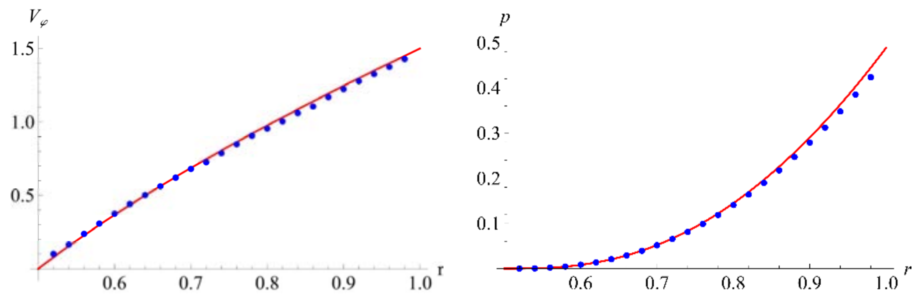

Figure 2: Velocity and pressure profiles for Couette flow between co-rotating cylinders (internal cylinder of radius 0.5 is immovable, external one with radius 1 is rotating); dots indicate numerical solution; solid line represents exact solution against the distance to the axis.

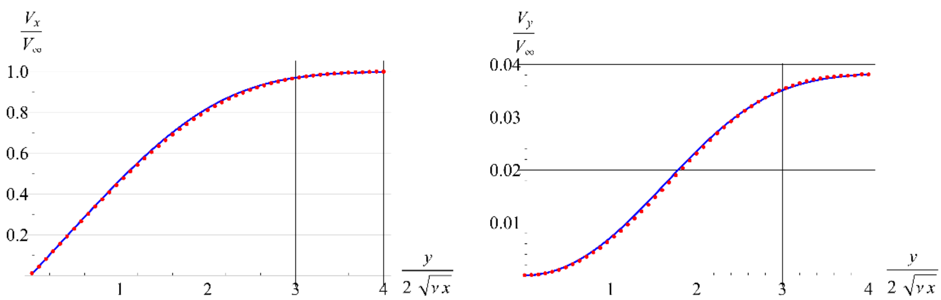

Figure 3: Horizontal V_x and vertical V_y dimensionless velocity profiles in the cross section on a thin plate (Blasius problem); the dependency against the self-similar variable is shown (ν is kinematic viscosity, V_∞ is incident flow velocity); dots indicate numerical solution; solid line represents exact solution.

Simulating the flow around airfoils at moderate and high Reynolds numbers presents a more complex challenge. On the one hand, the influence of viscous forces is comparatively small (and in the flow domain far from the airfoil boundary is in most cases negligible). However, at the same it remains essential to account for viscosity to accurately model the processes in the near-wall boundary layer, and thus correctly reproduce the separation of the flow from the airfoil contour line.

3 UNSTEADY FLOW SIMULATION AROUND CIRCULAR CYLINDER

Given that vortex particle methods are particularly efficient for simulating unsteady and transient flow regimes, let us consider the following model problem of flow simulation around a cylinder that impulsively starts moving with constant speed in still medium. Unsteady drag force acting on the cylinder at different values of the Reynolds number is a 'classical' test problem that has been considered by many researchers. Fig. 4 shows the vortex wakes in the initial phase of the cylinder motion obtained using the VM2D code for two cases that differ only in the values of the viscosity coefficient of the medium. The corresponding values of the Reynolds number are Re = 40 and Re = 200. The ratio of the cylinder diameter to the incident flow velocity is chosen as the time scale.

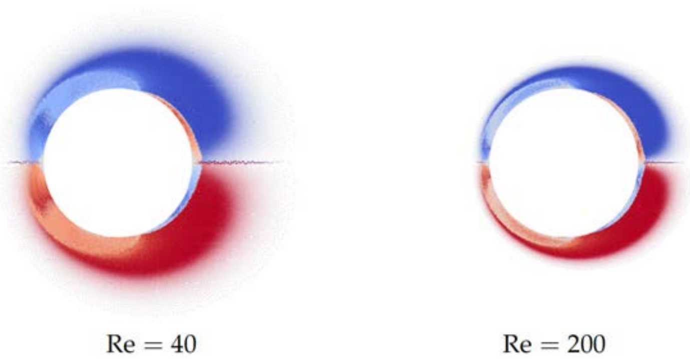

Figure 4: Vortex particles positions in vortex wakes after the impulsively started circular cylinder at Re $= 40$ and Re $= 200$ at time step $t_* = 0.5$.

The dependences of the total drag force, caused by the pressure distribution and viscous friction, on the dimensionless time for the considered cases are shown in Fig. 5 in comparison with the results of Bar-Lev and Yang [19], Collins and Dennis [20] and Kousoutsakos and Leonard [21]. It is seen that the simulation results are in good agreement with them.

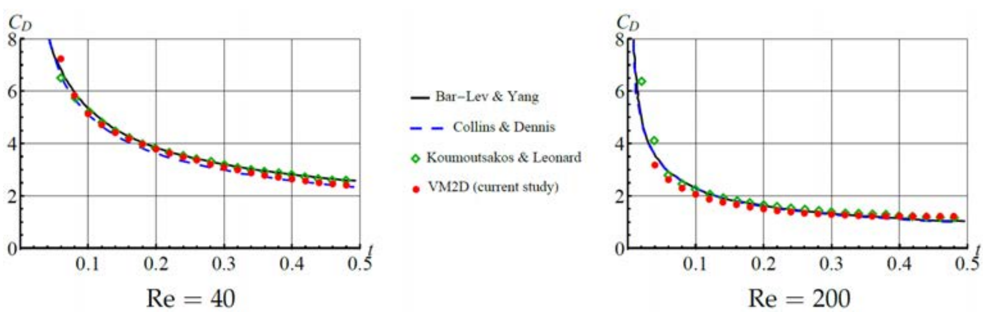

Figure 5: Unsteady drag force coefficient acting on the impulsively started circular cylinder at Re $= 40$ and Re $= 200$.

The similar problem was considered for the cylinder velocity corresponding to a higher Reynolds number Re = 3000. The dimensionless simulation time was chosen to be $t_* = 5.0$. The vortex wakes at $t = 1.0$, $t = 2.0$, $t = 3.0$, $t = 4.0$, $t = 5.0$ are shown in Fig. 6.

This problem turns out to be more complicated. To ensure the simulation results aligned with those from other researchers [21]–[27] for Re $= 3000$, it was necessary to consider a very accurate airfoil discretisation, a small time step, and a sufficiently large number of vortex particles in the flow domain (about half a million). The combination of such high resolution and implementation of the T-schemes for solving the boundary integral equation, made it possible to avoid numerical oscillations of drag and lift forces. These oscillations are common for many known implementations of vortex particle methods. Fig. 7 shows the results for unsteady drag force dependence on time, using the VM2D code in comparison with the other authors results.

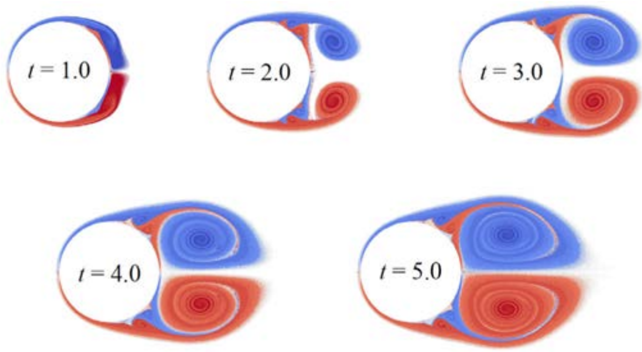

Figure 6: Vortex particles positions in vortex wakes after the impulsively started circular cylinder at Re = 3000 at different time steps.

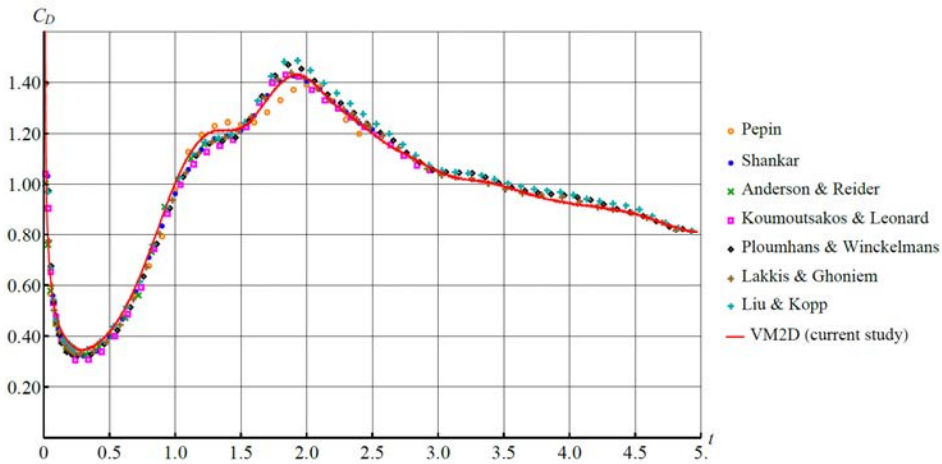

Figure 7: Unsteady drag force coefficient acting the impulsively started circular cylinder at Re = 3000.

Attempts to simulate flows around a circular airfoil at higher Reynolds numbers lead to an undesirable result: when modelling a quasi-steady flow regime using VM2D, the result is qualitatively correct (a Karman vortex street with periodic vortex shedding is observed), but quantitatively incorrect. For example, it is not possible to reproduce the well-known effect of ‘stabilisation’ of the drag coefficient: it is well-known that averaged drag coefficient of a cylinder remains close to 1.2 in a wide range of Reynolds numbers, and in the region of Re ≈ 1000 even decreases below 1.0 (Fig. 8).

When performing simulation using vortex particle methods, the drag coefficient turns out to be significantly overestimated, close to 1.5…1.6. In addition, it is clear from Fig. 8 that the value $\text{Re} = 2 \cdot 10^5$ corresponds to the so-called ‘drag crisis’, where a sharp decrease in the drag coefficient is observed. Although the qualitative effect of the ‘drag crisis’ is observed using vortex particle methods (as also noted in Dynnikova [29]), it appears prematurely at approximately $\text{Re} = 10^5$.

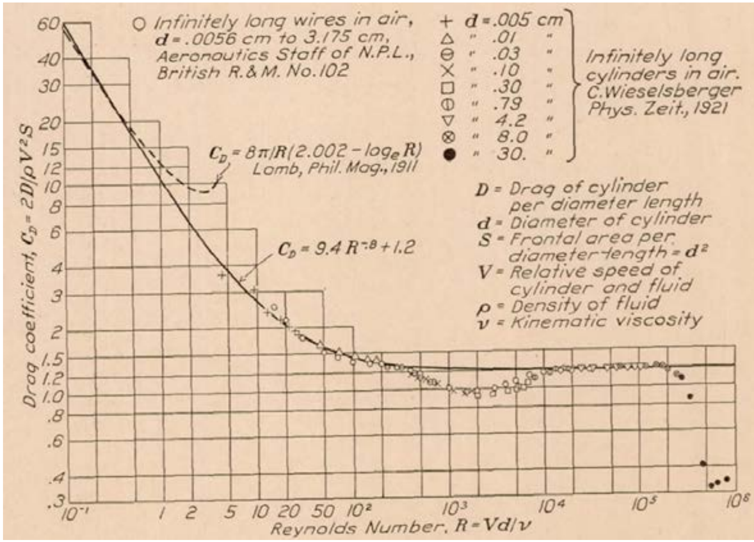

Figure 8: Stationary drag coefficient acting on a circular cylinder in dependence to Reynolds number [28].

4 FLOW SIMULATION AROUND WING AIRFOIL

Let us consider the problem of the flow simulation around wing airfoil at moderate Reynolds number values. Fig. 9 shows the vortex wake after a symmetric NACA-0012 airfoil at an angle of incidence of $\alpha = 6°$ in the steady-state flow regime at $\mathrm{Re} = 10^4$ and $\mathrm{Re} = 10^5$. The parameters of simulation were chosen similar to those used for the case of flow around a cylinder considered in the previous section.

$\mathrm{Re} = 10^4$:

$\mathrm{Re} = 10^5$:

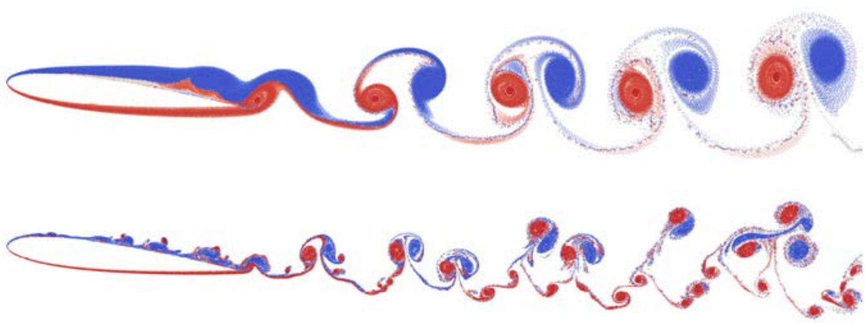

Figure 9: Vortex wake after the NACA-0012 airfoil for angle of incidence $\alpha = 6°$ at $\mathrm{Re} = 10^4$ and $\mathrm{Re} = 10^5$.

The dependences of the averaged drag and lift coefficients on the angle of attack (Fig. 10) are in acceptable agreement with the experimental data.

Despite slightly overestimated drag coefficient values, this is not critical due to their smallness. In fact, we can conclude that the flow is simulated correctly even at high Reynolds numbers.

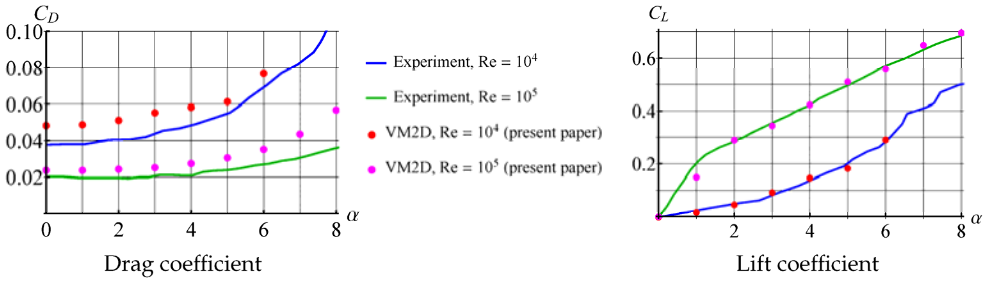

Figure 10: Drag and lift force coefficients against the angle of incidence for NACA-0012 airfoil.

If we continue to increase the angle of incidence of the wing, the computational results in VM2D significantly diverge from the experimental data: mainly, a greatly overestimated value of the lift coefficient is observed. At the same time, as the angle of incidence increases, the flow regime also changes, separation takes place not only at the sharp trailing edge but also on the smooth upper surface of the airfoil. Apparently, these effects are simulated incorrectly.

5 ON TURBULENT FLOW SIMULATION USING 2D VORTEX PARTICLE METHODS

Based on the results presented above, the following conclusion can be made. Flow simulation in two-dimensional formulation allows one to obtain correct results in two cases: at low Reynolds numbers – both for smooth airfoils and airfoils with edges and corner points, and at high Reynolds numbers, but only in cases where separation from a smooth surface is not simulated.

Thus, the following hypothesis seems justified: in the mentioned cases, the real (three-dimensional) flow around the structure is in fact essentially flat; at higher Reynolds numbers, the flow can be flat only on the macroscales, while microscale flows turn out to be essentially three-dimensional, but this microscale three-dimensionality affects the macroscopic flow characteristics. This effect is well known, and widespread mesh-based methods reproduce it using the so-called turbulence models, within the framework of the RANS or LES approaches. The averaged (filtered) flow is flat, while the pulsating component of the velocity field is three-dimensional, but not explicitly resolved.

It is well-known that the introduction of special closure models, which effectively embody the specifics of a given turbulence model, arises from the inability to provide the necessary spatial resolution of the computational mesh. If one performs direct numerical simulation, the typical spatial scale of structures that must be resolved in the flow domain is called the Kolmogorov scale and is inversely proportional to the value of $\mathrm{Re}^{3/4}$. In discussing small-scale effects, it should be noted that in relatively simple models, the turbulence is assumed to be isotropic in three spatial directions. ‘Turbulence’ itself is defined as essentially three-dimensional unsteady motion. Due to the stretching of vortices, a distribution of velocity pulsations occurs in the range of wavelengths, from the minimum, determined by the Kolmogorov scale, to the maximum determined by the size of the flow domain. The spectral distribution of the kinetic energy of turbulence has a so-called inertial range (intermediate between the range of large energy-containing vortices and the dissipation range, where the energy of vortices is transferred into heat as a result of the action of viscous forces). In inertial range turbulence can be considered homogeneous and isotropic, and according to the Kolmogorov–Obukhov law, the spectral density of the kinetic energy of turbulence E, is proportional to $k^{5/3}$, where k is the wave number [30].

In vortex particle methods for 2D flows, 'direct' reproduction of the mentioned effects is impossible because the primary computational variable, i.e., vorticity is a priori taken to be orthogonal to the flow domain. This, in fact, excludes the possibility of modelling the process of vortex stretching. Nevertheless, if one performs derivation of the estimation for energy spectrum (keeping in mind two-dimensionality of the flow), the inertial range consists of two subregions in 'two-dimensional turbulence': for large wave numbers, the asymptotics $E \sim k^{-3}$ is valid, and the law $E \sim k^{-5/3}$ applies only in a relatively narrow range [31]. This dependence can be practically tested for flow simulation using the VM2D code.

As an example, a simulation of the flow after a circular airfoil was performed at a Reynolds number $\mathrm{Re} = 10^6$. The value of the steady-state (averaged) drag coefficient is close to 0.45 (see the issue of the drag crisis discussed above). The plots of the spectral density of the kinetic energy of turbulence are shown in Fig. 11. The left plot was obtained by direct processing the simulation results: the velocity of the medium was calculated on a uniform grid with a small step in square area. The side length of the square was equal to half the radius of the circle and it was located behind it at a distance equal to 1/20 of the radius of the cylinder. Then two-dimensional discrete Fourier transform was performed; the plot shows the squared absolute values of the Fourier coefficients against the wave vector modulus; the plot is shown in a double logarithmic scale. The right plot was obtained by processing the time dependence for the value of velocity at a fixed point placed at the distance of 1/15 of the radius at angle 120°. A discrete Fourier transform was performed, the value $k = f/U$ was plotted along the horizontal axis, where f is the frequency, U is the incident flow velocity, and the squared absolute values of the corresponding Fourier coefficient were plotted along the vertical axis.

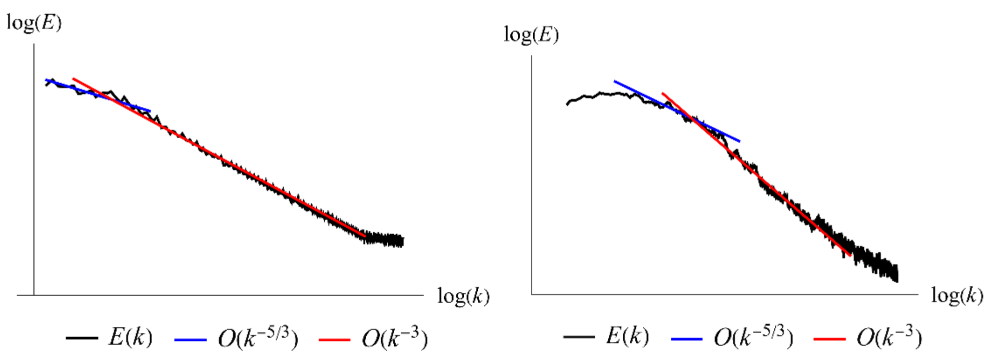

Figure 11: Spectral density plots of turbulence kinetic energy. (Left) For spatial velocity field at fixed time; and (Right) For velocity dependency against time at fixed point).

It is seen that the kinetic energy density spectrum of 'two-dimensional turbulence' is correctly reproduced in the inertial range without involving any additional models. This allows us to conclude that the algorithms for 2D flow simulation implemented in VM2D are sufficiently effective. Attempts to achieve a similar result using mesh-based methods (the

www.witpress.com, ISSN 1743-3533 (on-line)

finite volume method implemented in OpenFOAM) without involving turbulence models were unsuccessful. Such attempts would require an unacceptably tiny grid and time step, leading to prohibitively high computational costs.

Currently, only a few studies have attempted to combine vortex methods with RANS and LES approaches for simulating turbulent flows [32]–[35]. However, such approaches are not widespread, at least for near-wall flows simulation. This can be partly attributed to the challenges in accurately solving the boundary integral equation for the vortex sheet intensity generated on the airfoil, a problem that persisted until recently. Actually, this significantly limited the researchers in achieving high resolution in the near-wall region. The use of the above-mentioned *T*-schemes allows us to solve this problem and opens up the possibility of developing new modifications of vortex particle methods, including the possibility of turbulent effects simulation.

However, we would like to clarify that the primary goal of this work is to observe the problem and highlight the significant influence these 3D effects can have on flow simulations. This study aims to provide an initial exploration and set the stage for more detailed analyses. Quantitative comparisons and a thorough assessment of the extent of this influence fall beyond the scope of the current paper and warrant a dedicated, comprehensive investigation. Such a study is essential and we plan to address it in a separate follow-up work, which will focus specifically on the quantification and detailed analysis of these effects.

6 CONCLUSION

Using the VM2D code, the possibility of applying vortex methods based on the viscous vortex domain method to solving model problems on simulating flows around airfoil at moderate and high Reynolds numbers is considered. At low Reynolds numbers, the simulation results are in good agreement with experimental data. However, significant errors are observed at high Reynolds numbers.

For small angles of incidence, the presented dependencies of drag and lift forces on the airfoil at Reynolds numbers $\mathrm{Re} = 10^4$ and $\mathrm{Re} = 10^5$ are close to the experimental results. However, as the angle of incidence increases, discrepancies emerge again. This can be explained by the fact that at higher angles, flow separation takes place not only on the sharp edge, but also on the upper smooth surface of the airfoil, and such separation as in the case of a circular airfoil, is simulated incorrectly. The latter is due to the fact that at high Reynolds numbers the flow can be modelled as flat, but only on macroscales, while microscale flow is essentially three-dimensional, especially in the zone of separation. Such microscale three-dimensionality cannot be reproduced in two-dimensional algorithms of vortex particle methods, but it has a significant impact on the flow.

At the same time, analysis of the spectrum of the kinetic energy of turbulence shows that it corresponds to ‘two-dimensional turbulence’ with high accuracy. Thus, we can conclude that the algorithms for modelling flat flows implemented in VM2D are sufficiently effective.

REFERENCES

[1] Cottet, G.-H. & Koumoutsakos, P.D., *Vortex Methods: Theory and Practice*, Cambridge University Press, 328 pp., 2000.

[2] Andronov, P.R., Guvernyuk, S.V. & Dynnikova, G.Y., *Vortex Methods for Non-Stationary Hydrodynamic Loads Estimation*, Moscow State University, 184 pp., 2006.

[3] Mimeau, C. & Mortazavi, I., A review of vortex methods and their applications: from creation to recent advances. *Fluids*, **6**(2), pp. 1–25, 2021.

[4] Kempka, S.N., Glass, M.W., Peery, J.S., Strickland, J.H. & Ingber, M.S., Accuracy considerations for implementing velocity boundary conditions in vorticity formulations. *SANDIA Report*, SAND96-0583, UC-700, 52 pp., 1996.
[5] Dynnikova, G.Y., An analog of the Bernoulli and Cauchy–Lagrange integrals for a time dependent vortex flow of an ideal incompressible fluid. *Fluid Dynamic*, **35**(1), pp. 31–41, 2000.
[6] Devnin, S.I., *Hydroelasticity of Structures under Separated Flow*, Sudostroenie: Leningrad, 99 pp., 1975.
[7] Marchevsky, I., Sokol, K., Ryatina, E. & Izmailova, Y., The VM2D open source code for two-dimensional incompressible flow simulation by using fully Lagrangian vortex particle methods. *Axioms*, **12**(3), pp. 1–24, 2023.
[8] Dynnikova, G.Y., The Lagrangian approach to solving the time-dependent Navier–Stokes equations. *Doklady Physics*, **49**, pp. 648–652, 2004.
[9] Marchevskii, I.K., Sokol, K.S. & Izmailova, Y.A. T-schemes for mathematical modelling of vorticity generation on smooths airfoils in vortex particle methods. *Herald of the Bauman Moscow State Technical University, Series Natural Sciences*, **6**(105), pp. 33–59, 2022.
[10] Kuzmina, K., Marchevsky, I., Soldatova, I. & Izmailova, Y., On the scope of Lagrangian vortex methods for two-dimensional flow simulations and the POD technique application for data storing and analysing. *Entropy*, **23**(1), pp. 1–17, 2021.
[11] Belotserkovsky, S.M. & Lifanov, I.K., *Method of Discrete Vortices*, CRC Press: Boca Raton, 464 pp., 1992.
[12] Lifanov, I.K., *Singular Integral Equations and Discrete Vortices*, VSP: Utrecht, The Netherlands, 475 pp., 1996.
[13] Katz, J. & Plotkin, A., *Low-Speed Aerodynamics: From Wing Theory to Panel Methods*, McGraw-Hill: Singapore, 632 pp., 1991.
[14] McBain, G.D., *Theory of Lift: Introductory Computational Aerodynamics in MATLAB/Octave*, Wiley: New York, 342 pp., 2012.
[15] Chorin, A.J., Numerical study of slightly viscous flow. *Journal of Fluid Mechanics*, **57**(4), pp. 785–796, 1973.
[16] Degond, P. & Mas-Gallic, S., The weighted particle method for convection-diffusion equations. I. The case of an isotropic viscosity. *Mathematics of Computation*, **53**(188), pp. 485–507, 1989.
[17] Kuwahara, K. & Takami, H., Numerical studies of two-dimensional vortex motion by a system of point vortices. *Journal of the Physical Society of Japan*, **34**(1), pp. 247–253, 1973.
[18] Rossi, L.F., Resurrecting core spreading vortex methods: A new scheme that is both deterministic and convergent. *SIAM Journal on Scientific Computing*, **17**(2), pp. 370–397, 1996.
[19] Bar-Lev, M. & Yang, H.T., Initial flow over an impulsively started circular cylinder. *Journal of Fluid Mechanics*, **72**(4), pp. 625–647, 1975.
[20] Collins, W.M. & Dennis, S.C.R., The initial flow past an impulsively started circular cylinder. *Quarterly Journal of Mechanics and Applied Mathematics*, **26**(1), pp. 53–75, 1973.
[21] Koumoutsakos, P. & Leonard, A., High-resolution simulations of the flow around an impulsively started cylinder using vortex methods. *Journal of Fluid Mechanics*, **296**, pp. 1–38, 1995.

[22] Pepin, F.M., Simulation of the flow past an impulsively started cylinder using a discrete vortex method. PhD thesis, California Institute of Technology, Pasadena, USA, 1990.
[23] Shankar, S., A new mesh-free vortex method. PhD thesis, FAMU-FSU College of Engineering, Tallahassee, USA, 1996.
[24] Anderson, C.B. & Reider, M.B., A high order explicit method for the computation of flow about a circular cylinder. *Journal of Computational Physics*, **125**(1), pp. 207–224, 1996.
[25] Ploumhans, P. & Winckelmans, G.S., Vortex methods for high-resolution simulations of viscous flow past bluff bodies of general geometry. *Journal of Computational Physics*, **165**(2), pp. 354–406, 2000.
[26] Lakkis, I. & Ghoniem, A., A high resolution spatially adaptive vortex method for separating flows. Part I: Two-dimensional domains. *Journal of Computational Physics*, **228**, pp. 491–515, 2009.
[27] Liu, Z. & Kopp, G.A., High-resolution vortex particle simulations of flows around rectangular cylinders. *Computers and Fluids*, **40**(1), pp. 2–11, 2011.
[28] Zahm, A.F., Flow and drag formulas for simple quadrics. NACA Technical Report No. 253, 23 pp., 1927.
[29] Dynnikova, G.Y., Fast technique for solving the N-body problem in flow simulation by vortex methods. *Computational Mathematics and Mathematical Physics*, **49**(8), pp. 1389–1396, 2009.
[30] Kolmogorov, A.N., The local structure of turbulence in incompressible viscous fluid for very large Reynolds numbers. *Proceedings of the Royal Society A: Mathematical, Physical and Engineering Sciences*, pp. 9–13, 1991.
[31] Kraichnan, R.H., Inertial ranges in two-dimensional turbulence. *Physics of Fluids*, **10**(7), pp. 1417–1423, 1967.
[32] Pereira, L.A.A., Hirata, H. & Silveira Neto, A., Vortex method with turbulence sub-grid scale modelling. *Journal of the Brazilian Society of Mechanical Sciences and Engineering*, **25**(2), pp. 140–146, 2003.
[33] Yokota, R. & Shinnosuke, O., Vortex methods for the simulation of turbulent flows: Review. *Journal of Fluid Science and Technology*, **6**(1), pp. 14–29, 2011.
[34] Branlard, E., Papadakis, G., Gaunaa, M., Winckelmans, G. & Larsen T., Aeroelastic large eddy simulations using vortex methods: unfrozen turbulent and sheared inflow. *Journal of Physics: Conference Series*, **625**, 012019, 2015.
[35] Alvarez, E.J. & Ning, A., Reviving the vortex particle method: A stable formulation for meshless large eddy simulation. arXiv preprint arXiv:2206.03658, 2022.