6th European Conference on Computational Mechanics (ECCM 6)
7th European Conference on Computational Fluid Dynamics (ECFD 7)
1115 June 2018, Glasgow, UK

ON PARTITIONED AND MONOLITHIC COUPLING STRATEGIES IN LAGRANGIAN VORTEX METHODS FOR 2D FSI PROBLEMS

KSENIIA S. KUZMINA[1,2], ILIA K. MARCHEVSKY[1,2] AND EVGENIYA P. RYATINA[1,2]

[1] Bauman Moscow State Technical University
105005, Russia, Moscow, 2-nd Baumanskaya st., 5

[2] Ivannikov Institute for System Programming of the RAS
109004, Russia, Moscow, Alexander Solzhenitsyn st., 25

kuz-ksen-serg@yandex.ru, iliamarchevsky@mail.ru, evgeniya.ryatina@yandex.ru

Key words: Vortex Methods, Viscous Incompressible Media, Boundary Integral Equation, Fluid-Structure Interaction, VM2D Code

Abstract. Lagrangian vortex methods are powerful tool for numerical simulation in CFD and coupled fluid structure interaction (FSI) problems. The main feature of vortex methods is vorticity considering as a primary computed variable, while velocity and pressure distributions can be reconstructed by using the Biot — Savart law and the Cauchy — Lagrange integral analogue. Advantages of vortex methods include the possibility to "concentrate" computational resources in the domain with non-zero vorticity and to simulate arbitrary large displacements of the airfoils in the flow. Moreover, in contrast to mesh methods, computational cost of such simulation just a little higher than for flow simulation around immovable airfoil. In this research the problems of flow simulation around oscillating and immovable circular cylinders using vortex methods are considered. The wind resonance phenomenon is investigated, when eigenfrequency of the mechanical system is close to vortex shedding frequency. All numerical results are in good agreement with experimental data.

1 INTRODUCTION

One of the most typical Fluid-Structure Interaction (FSI) problem is flow simulation around elastically fixed circular cylinder which oscillates under hydrodynamic forces. Such problem arises in number of engineering applications: strain estimation of smoke stacks and high towers caused by a wind load, dynamics simulation of underwater and overwater oil and gas pipelines, vibration simulation of heat exchanger pipelines in nuclear reactors, Aeolian vibration and galloping simulation of overhead power line wires, etc. We consider two-dimensional FSI problem of flow simulation around circular airfoil, which oscillates with rather high amplitude (comparable with diameter of the cylinder) at low and intermediate Reynolds numbers, when it is impossible to assume the flow to be potential and use simplified approaches to hydrodynamic loads prediction. Moreover, we consider the eigenfrequency of the mechanical system to be close to vortex shedding frequency; it means that it is necessary to simulate unsteady flow and solve coupled hydroelastic problem. If flow velocity is rather low, when the media compressibility influence can be neglected, such problem can be solved efficiently by using vortex methods which belong to the class of meshless Lagrangian CFD methods. The aim of this research is to estimate the accuracy and efficiency of the VM2D open source code developed by the authors and investigate the possibility of the partitioned coupling strategy usage in the mentioned model problem.

2 GOVERNING EQUATIONS

The viscous incompressible flow of the constant density is described by the continuity equation and the Navier — Stokes equations

$$\nabla \cdot \boldsymbol{V} = 0; \tag{1}$$

$$\frac{\partial \boldsymbol{V}}{\partial t} + (\boldsymbol{V} \cdot \nabla)\boldsymbol{V} - \nu \Delta \boldsymbol{V} = -\frac{\nabla p}{\rho}, \tag{2}$$

where $\boldsymbol{V} = \boldsymbol{V}(\boldsymbol{r},\, t)$ is the flow velocity, ν is the kinematic viscosity coefficient, $p = p(\boldsymbol{r},\, t)$ is the pressure, $\rho = \text{const}$ is the density of the flow.

The parameters of the incident flow are considered to be constant:

$$\boldsymbol{V} \to \boldsymbol{V}_\infty \quad \text{and} \quad p \to p_\infty \quad \text{at} \quad |\boldsymbol{r}| \to \infty. \tag{3}$$

The position of the airfoil surface line K is determined by the airfoil dynamics, so the no-slip boundary condition should be satisfied at movable boundary:

$$\boldsymbol{V}(\boldsymbol{r},\, t) = \boldsymbol{V}_K(\boldsymbol{r},\, t), \quad \boldsymbol{r} \in K, \tag{4}$$

where $\boldsymbol{V}_K$ is the velocity of the airfoil surface line.

The aerodynamic loads acting on the airfoil, in particular, the drag and lift force, can be calculated if the pressure distribution as well as viscous stresses distributions are known over the surface line. In order to compute pressure force, the integral expression is derived [1], which follows from the Cauchy — Lagrange integral analogue for non-potential flows [2].

We consider motion of a circular airfoil with one degree of freedom; it is fixed with a linear viscoelastic Voigt-type constraint can oscillate across the flow due to the constraint force and lift force action, Fig.1. The airfoil moves in the flow according to the following equation:

$$m\ddot{y} + b\dot{y} + ky = F_y, \tag{5}$$

where m is the mass of the airfoil, b is the damping coefficient, k is the rigidity of the constraint, F_y is the lift force, and y is the airfoil deviation from the equilibrium position.

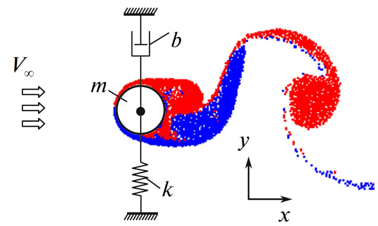

Figure 1: Oscillations simulation with 1 degree of freedom for circular cylinder

At the Reynolds numbers $\mathrm{Re} = V_\infty D/\nu > 40$, a chain of vortices descending one after another is formed (the so-called von Kármán vortex trail). The vortex shedding causes a periodical variation in the pressure distribution and the resulting aerodynamic force and, therefore, the airfoil starts to oscillate.

3 NUMERICAL SIMULATION IN FSI PROBLEMS BY USING THE VM2D CODE

The description of vortex methods can be found in [3, 4, 5, 6]. However, vortex methods are not implemented in any known software packages: both commercial and freely distributed. The authors of the present paper have suggested some modifications of numerical algorithms of vortex methods [7, 8] and have started to develop new software package **VM2D — Open source code for two-dimensional flows simulation using vortex methods** [9, 10]. Parallel algorithms are implemented in VM2D, that makes it possible to accelerate numerical simulations significantly. Note, that GPU usage seems to be the most efficient way, one graphical accelerator (GPU) permits to achieve nearly the same performance as $200\ldots1000$ central processor (CPU) cores [11, 12].

The numerical solution of FSI problem problem requires simultaneous solving of equations of both hydrodynamical and mechanical equations. There are two different approaches to implement the coupling strategy in FSI-problems: partitioned and monolithic ones. According to the partitioned strategy, each time step should be split into hydrodynamical and mechanical substeps. At the first substep we simulate the flow around the airfoil moving with known velocity and position and compute hydrodynamic loads acting it. At the mechanical substep hydrodynamic forces are assumed to be known from the previous substep and dynamics equations for the mechanical system are solved. This strategy is rather simple in implementation. The main advantage of such strategy is the possibility of independent solvers usage, interacting only through the interfaces, so it is easy to develop new methods and algorithms, improve existing ones or even use some "outer" solvers. However, this strategy can't be applied for the problems when the density of the airfoil is comparable with the density of the flow. In this case numerical algorithm becomes unstable.

The second strategy (monolithic) is based on simultaneous solution of two sub-problems being discretized and written down in form of one system of linear algebraic equations. In rather simple cases it is possible due to linear expression of the integral forces through parameters of the vortex sheet being generated on the airfoil surface [1]. In more complicated problems, i.e., in case of deformable airfoil or non-linear constraint, monolithic approach in general case leads to non-linear equations, which require application of iterative methods. The main advantages of the monolithic method are high accuracy and stability, however in the framework of this approach it's impossible to split the code into different modules.

4 THE FLOW AROUND IMMOVABLE AIRFOIL

In order to verify the VM2D code and the algorithms implemented there, the flow around immovable circular airfoil was simulated. The dimensionless parameters were the following: $V_\infty = 3.0$, $\rho = 1.0$, the kinematic viscosity is $\nu = 0.02$, and the diameter of the airfoil is $D = 1.0$. Such flow regime is characterized by the Reynolds number $\mathrm{Re} = V_\infty D/\nu = 150$. The airfoil was discretized into $N_p = 400$ panels, the time step was chosen equal to $\tau = 0.005$, the radius of the vortex element was $\varepsilon = 0.004$.

The Strouhal number $\mathrm{St} = fD/V_\infty$ calculated with respect to the frequency f of lift force oscillations is about 0.178. This result is in good agreement with experimental results [13] and CFD simulations, i.e., [14].

The obtained results show that the vortex method implemented in the VM2D code permits to simulate the viscous incompressible flow with acceptable accuracy in a short period of time: the flow reaches quasi-steady regime at $t \approx 25$, so we need to perform 5 000 time steps. Performing computations by using 1 CPU (4 core, Intel i7-920) + 1 GPU (Nvidia Geforce 970), we spent less then 1500 seconds (25 minutes) in order to obtain the flow in steady state regime. It means that 1 time step performing requires only 0.3 sec.

Figure 2 shows the typical form of the vortex wake (von Kármán vortex trail), which is formed behind the immovable circular airfoil.

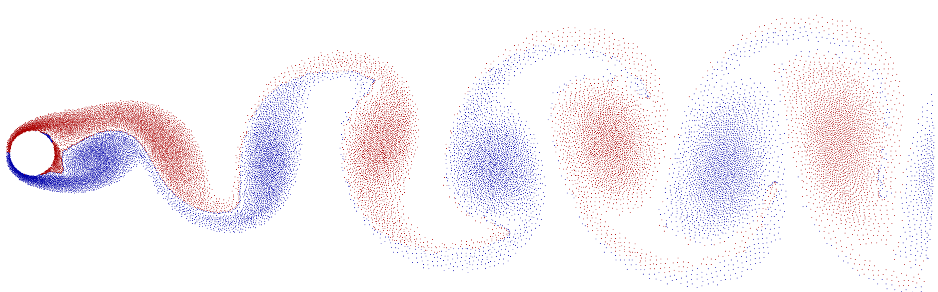

Figure 2: von Kármán vortex trail behind the circular airfoil

Hydrodynamic forces acting on the cylinder, are also in good comparison with experimental data [15].

5 WIND RESONANCE SIMULATION

For simulation of the wind resonance phenomenon (Fig. 1) the same dimensionless parameters as in case of an immovable airfoil were used. The airfoil's mass was $m = 39.15$

and the damping coefficient was $b = 0.731$. Varying the rigidity of the constraint, it is possible to change the eigenfrequency of the mechanical system $\omega \approx \sqrt{k/m}$. Note, that it is possible to neglect the influence of the damping in the constraint, since it causes the eigenfrequency variation of less than 1 %.

The wind resonance phenomenon takes place when the Strouhal number

$$\mathrm{St}_\omega = \frac{\omega}{2\pi}\frac{D}{V_\infty},$$

which means the dimensionless frequency of natural oscillations, is close to the Strouhal number St calculated with respect to the frequency of vortex shedding, which determines the frequency of the lift force oscillations.

Oscillation processes (Fig. 1) were simulated within a dimensionless period of time $0 \le t \le 250$. At the initial time, the airfoil is in an equilibrium position and its initial velocity is equal to zero. Each simulation process was performed using 1 CPU (4 core, Intel i7-920) + 1 GPU (Nvidia Geforce 970); when the oscillations are far from the resonance regime, the computational time is about 0.34 sec. per step, in case of resonance regime computational time is about 0.58 sec. per step. The time dependencies of the airfoil oscillations amplitude for two different regimes are shown in Fig. 3-4.

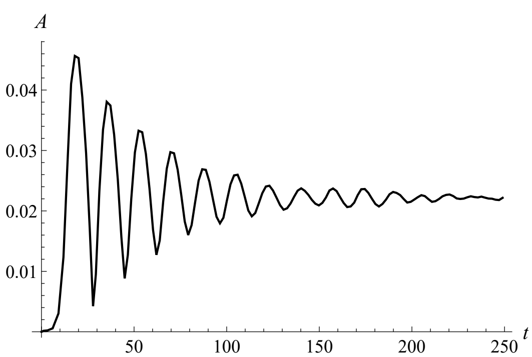

Figure 3: Oscillations amplitude at $\mathrm{St}_\omega = 0.156$

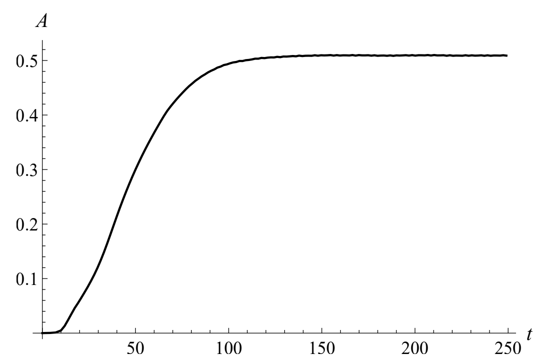

Figure 4: Oscillations amplitude at $\mathrm{St}_\omega = 0.2$

It can be seen that the amplitude of oscillation behavior is quite different. When the Strouhal number corresponding to the frequency of natural oscillation $\mathrm{St}_\omega = 0.156$, differs significantly from the Strouhal number, which corresponds to the frequency of vortex shedding from the immovable cylinder, the amplitude of the airfoil oscillations (Fig. 3) remains rather small: not more than 5 % of the diameter in transient regime and about 2 % of the cylinder diameter in steady-state regime.

In the case when St_ω is close to the Strouhal number, which corresponds to the frequency of vortex shedding from the immovable cylinder, or at slightly higher values of St_ω (due to well-known vortex shedding frequency lock-in phenomenon [16]) the transient regime is very "smooth", the amplitude increases monotonously and it reaches rather high values — about 50 % of the cylinder diameter. The time dependence for the amplitude of oscillations at $\mathrm{St}_\omega = 0.200$ is shown in Fig. 4.

Figure 5 shows the dependence of the average value of the oscillations amplitude in steady-state regime on the Strouhal number in the range $\mathrm{St}_\omega = 0.154 \ldots 0.224$.

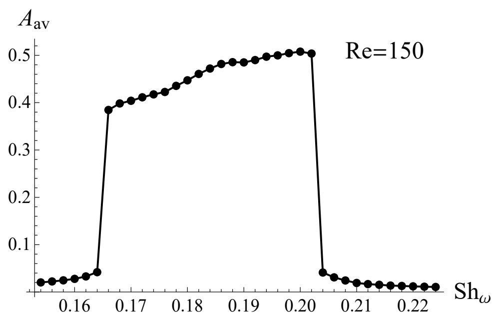

Figure 5: Dependence of the average oscillations amplitude A_{av} on the Strouhal number Sh_ω

It is seen that for the Strouhal numbers $\mathrm{St}_\omega = 0.166 \ldots 0.202$, i.e., inside the lock-in range, the amplitude increases significantly — more than 35 % of the cylinder diameter. This phenomenon is called "wind resonance".

For the chosen parameters, the maximal amplitude of the airfoil oscillations is about 0.507 of its diameter, it occurs when the Strouhal number $\mathrm{St}_\omega = 0.200$. The results of numerical flow simulation around oscillating circular cylinder using vortex method are in good agreement with the experimental data: maximal amplitude of the circular airfoil oscillation is about 0.5 of its diameter [18, 17]. The obtained results for the drag and lift coefficients C_x and C_y are shown in Fig. 6 for resonance regime with the highest amplitude ($\mathrm{St}_\omega = 0.200$) and in Fig. 7 for the non-resonance regime ($\mathrm{St}_\omega = 0.156$).

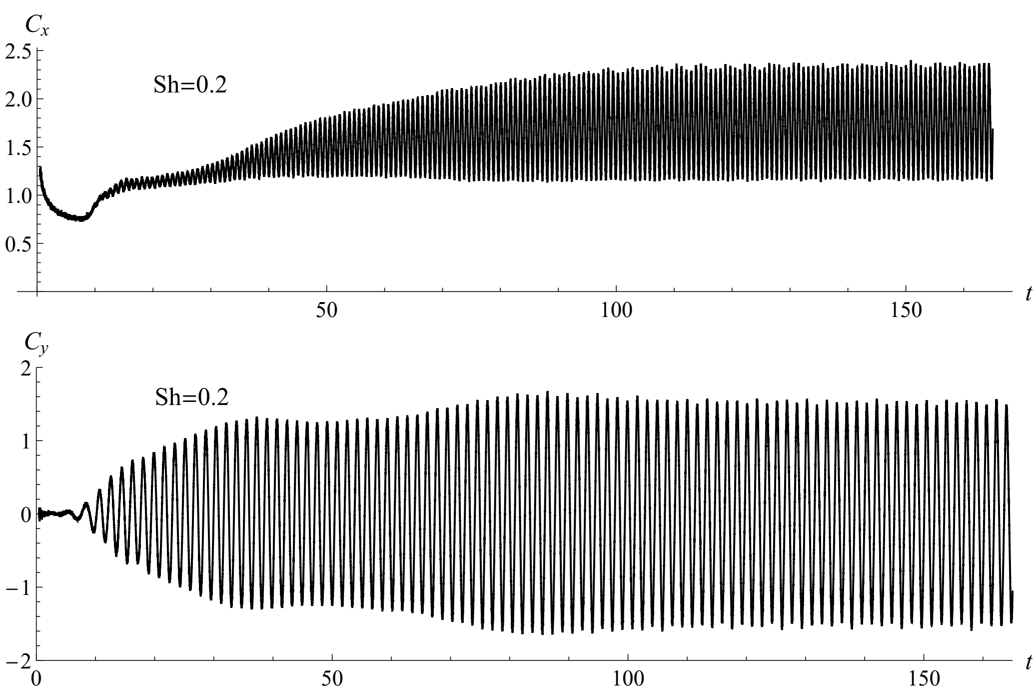

Figure 6: Unsteady drag coefficient C_x and lift coefficient C_y for resonance regime

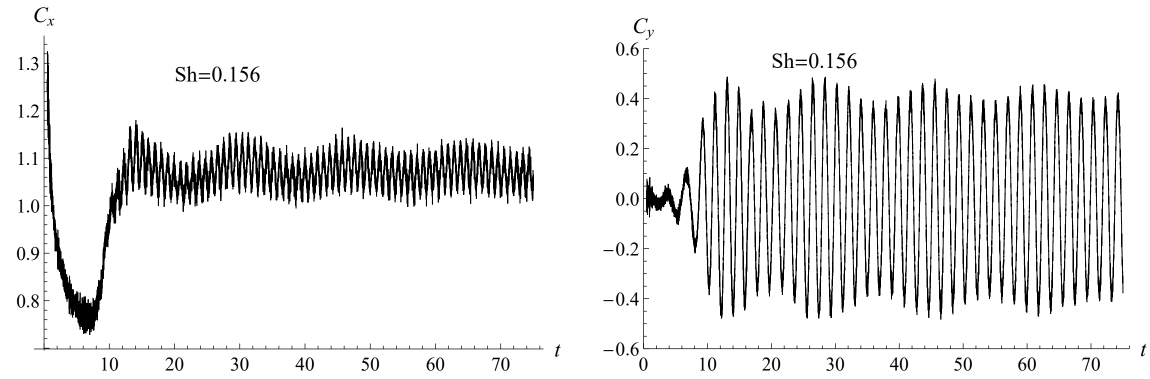

Figure 7: Unsteady drag coefficient C_x and lift coefficient C_y for non-resonance regime

It should be noted, that all the described results have been obtained by using partitioned approach (coupling strategy). It is possible due to high density ratio of the airfoil and the flow: in the considered example $\rho_{\text{body}}/\rho_{\text{flow}} \approx 50$. However, the partition coupling strategy remains applicable for lower values of density ratio. Numerical simulations show, that it is possible to simulate the oscillations of the airfoil with the same discretization parameters and the same eigenfrequency for $\rho_{\text{body}}/\rho_{\text{flow}} > 2$. At the lower values of the density ratio the numerical instability occurs, and it can not by suppressed, for example, by reducing the time step value.

The only way for numerical simulation by using vortex method in such cases is monolithic coupling strategy usage.

ACKNOWLEDGEMENTS

The research is supported by the Russian Foundation for Basic Research (projects no. 17-08-01468 and 18-31-00245) and the Grant of the President of the Russian Federation for Young Ph.D. scientists (project no. MK-743.2018.8).

REFERENCES

[1] Dynnikova, G.Ya. The integral formula for pressure field in the nonstationary barotropic flows of viscous fluid. *Journal of Mathematical Fluid Mechanics.* (2014) **16:1**: 145–162.

[2] Dynnikova, G.Ya. An analog of the Bernoulli and Cauchy – Lagrange integrals for a time-dependent vortex flow of an ideal incompressible fluid. *Fluid Dynamics.* (2000) **35:1**:24–32.

[3] Lifanov, I.K. *Singular Integral Equations and Discrete Vortices*. Utrecht: VSP, 1996.

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

[5] Andronov, P.R., Guvernyuk, S.V. and Dynnikova G.Ya. *Vortex Methods for Unsteady Hydrodynamic Loads*. Moscow, MSU, 2006. [in Russian]

[6] Dynnikova, G.Ya. Vortex motion in two-dimensional viscous fluid flows. *Fluid Dynamics.* (2003) **38:5**:670–678.

[7] Kuzmina, K.S., Marchevskii, I.K., Moreva, V.S. and Ryatina, E.P. Numerical scheme of the second order of accuracy for vortex methods for incompressible flow simulation around airfoils. *Russian Aeronautics.* (2017) **60:3**:398–405.

[8] Kuzmina, K.S., Marchevskii, I.K. and Moreva, V.S. Vortex Sheet Intensity Computation in Incompressible Flow Simulation Around an Airfoil by Using Vortex Methods. *Mathematical Models and Computer Simulations.* (2018) **10:3**:276–287.

[9] Kuzmina, K.S., Marchevsky I.K. and Ryatina E.P. Open Source Code for 2D Incompressible Flow Simulation by Using Meshless Lagrangian Vortex Methods. *IEEE Xplore Digital Library. Ivannikov ISPRAS Open Conference.* (2017) 97–103.

[10] Kuzmina, K.S., Marchevsky I.K. and Ryatina E.P. Open-source code VM2D for 2D incompressible flow simulation by using vortex methods. *Communications in Computer and Information Science.* (2018) In press.

[11] Kuzmina, K.S., Marchevsky, I.K. and Moreva, V.S. Parallel Implementation of Vortex Element Method on CPUs and GPUs. *Procedia Comp. Science.* (2015) **66**:73–82.

[12] Kuzmina, K.S., Marchevsky I.K. and Ryatina E.P. On CPU and GPU Parallelization of VM2D Code for 2D Flows Simulation Using Vortex Method. *6th European Conference on Computational Mechanics (ECCM 6), 7th European Conference on Computational Fluid Dynamics (ECFD 7)* (2018) In press.

[13] Roshko, A. On the development of turbulent wakes from vortex streets. *NACA Tech. Rept. No. 1191* (1954).

[14] Jiang, H. and Cheng, L. Strouhal-Reynolds number relationship for flow past a circular cylinder. *J. of Fluid Mechanics.* (2017) **832**:170–188.

[15] Zahm, A.F. Flow and Drag Formulas for Simple Quadrics. *NACA Tech. Rept. No. 253* (1927).

[16] Anagnostopoulos, P. and Bearman, P.W. Response characteristics of a vortex-excited cylinder at low reynolds numbers. *J. of Fluids and Structures.* (1992) **6:1**:39–50.

[17] Klamo, J.T., Leonard, A. and Roshko, A. On the Maximum Amplitude for a Freely Vibrating Cylinder in CrossFlow. *J. of Fluids and Structures.* (2005) **21**:429–434.

[18] Govardhan, R.N. and Williamson, C.H.K. Defining the 'Modified Griffin Plot' in Vortex-Induced Vibration: Revealing the Effect of Reynolds Number Using Controlled Damping. *J. Fluid Mech.* (2006) **561**:147–180.