

## ON OPENFOAM EFFICIENCY FOR SOLVING FLOW-STRUCTURE INTERACTION PROBLEMS

K.S. KUZMINA<sup>1</sup>, I.K. MARCHEVSKY<sup>1,2</sup>

<sup>1</sup>*Bauman Moscow State Technical University, kuz-ksen-serg@yandex.ru*

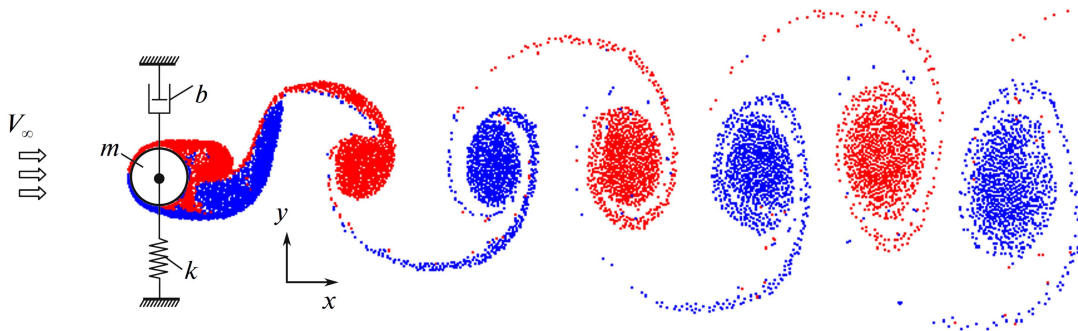
<sup>2</sup>*Institute for System Programming of the Russian Academy of Sciences, iliamarchevsky@mail.ru*

**Keywords:** FSI-problem, wind resonance, finite volume method, particle finite element method, vortex element method

In number of engineering applications we deal with bodies which are immersed into gas or fluid flow and exposed to aerohydrodynamic loads. Fully coupled 3D flow-structure interaction problems are extremely complicated both from mathematical and computational point of view. In many practical cases the average density of the immersed body is higher than the density of the flow, so it is possible to apply well-known ‘splitting’ approach, when we divide the time step into at least two substeps. At the first substep we use semi-implicit scheme when body motion parameters are assumed to be known and we simulate the flow around it; at the second one explicit scheme is being used for body motion simulation under known hydrodynamic loads.

The other simplification is connected with the fact that in practice we often deal with extended constructions (with high elongation), so as a rough approximation we can consider 2D problem of flow interaction with the corresponding airfoil. In fact, when the cross-section airfoil has angle point and sharp edges, this approach is correct. In case of bluff-bodies (for example, flow around cylindrical rods or pipes) they oscillate under von-Karman vortices shedding and frequency-lock phenomenon takes place, which also lead to identical (in some sense) flow around different cross-sections.

A very good test problem can be suggested for comparison of the accuracy and performance of different numerical methods and codes. It is well-known phenomenon of wind-resonance of circular cylinder airfoil. The design model is shown in Figure 1.



**Figure 1: Circular airfoil in the flow under viscoelastic constraints**

The most interesting case corresponds the situation when the eigenfrequency of the system (in the flow!) is close to von-Karman vortices shedding frequency. It is well-investigated both experimentally and numerically for wide range of the parameters: Reynolds number, airfoil surface roughness, incident flow turbulence, *etc.*

We consider the simplest case, when we neglect the roughness influence and the incident flow assumed to be laminar. Number of numerical codes, both commercial and open-source, can be used for simulation of airfoil oscillations in the flow. We consider three numerical methods and the corresponding open-source codes: finite volume method (FVM) [1] with deformable mesh in OpenFOAM [2]; particle finite element method (PFEM) [3] with deformable mesh in Kratos software [4]; and meshfree lagrangian vortex element method (VEM) [5, 6] in ‘homemade’ numerical code.

The advantage of VEM is the absence of the mesh, so the computation of the flow around movable and fixed airfoil have nearly the same complexity, and it’s possible to simulate arbitrary movements and deformations of the airfoil. Moreover, in some engineering applications this method allows to obtain acceptable results much faster in comparison with ‘classical’ mesh methods.

In the basis of PFEM method the following approach lies: in order to take into account the convectional term in Navier — Stokes equation ‘flow marker particles’ are introduced and they move with flow velocity. It allows to solve the governing equations in semi-lagrangian way: non-linearity is taken into account directly, while the remaining equation with linear terms is being solved by using well-proven finite element technique. The advantage of this method is low numerical dissipation and the opportunity to solve linear equation with high values (up to 10 or even more) of CFL number.

Unfortunately, it is impossible to use VEM and PFEM when we need to simulate high-Reynolds flows. In VEM it is principally impossible to implement any turbulence models; for PFEM it can be done, but today's Kratos software realization doesn't have the corresponding functionality.

We use experimental data [7] in order to compare the accuracy and performance of the mentioned codes. When the Reynolds number is small (not more than 5 000 . . . 10 000) all the mentioned codes allow to obtain the results, which are in good agreement with experimental data. The efficiencies of all considered methods and codes are comparable.

But when simulating high-Reynolds regimes, when it is necessary to take into account the turbulence, OpenFOAM usage is the only possibility to solve the problem with acceptable computation cost. It should be noted, however, that turbulence model as well as RANS/LES approach choice is non-trivial problem, and non-suitable methods of turbulence simulation can lead to wrong results. Moreover, OpenFOAM is well parallelized, so time of computations can be reduced significantly. At the same time parallel algorithms in VEM are efficient only for small number of computational cores, and PFEM realization in Kratos software now can work in parallel mode only using OpenMP technology (on shared-memory systems).

As the result, the following conclusion can be done: OpenFOAM usage is preferable for numerical simulation in such FSI-problems, when we need to investigate the behavior of the system in wide range of parameters.

### Acknowledgments

This research was partially supported by Russian Federation Grant for young PhD scientists (proj. MK-7431.2016.8) and The Ministry of Education and Science of the Russian Federation Support Program for Universities (base part of state assignment, proj. No 1712). The computations were performed on cluster MVS-100K provided by Joint Supercomputer Center of the Russian Academy of Sciences (JSCC RAS).

### References

- [1] J. Ferziger and M. Peric, *Computational methods for fluid dynamics*. Berlin, New York: Springer, 2002.
- [2] "Openfoam — the open source computational fluid dynamics (cfD) toolbox." [Online]. Available: <http://www.openfoam.com>
- [3] S. Idelsohn, N. Nigro, A. Limache, and E. Onate, "Large time-step explicit integration method for solving problems with dominant convection," *Computational Methods in Appl. Mech. and Eng.*, vol. 217-220, pp. 168–185, 2012. [Online]. Available: <http://dx.doi.org/10.1016/j.cma.2011.12.008>
- [4] "Kratos multi-physics." [Online]. Available: <http://www.cimne.com/kratos>
- [5] G.-H. Cottet and P. D. Coumoutsakos, *Vortex Methods. Theory and Practice*. Cambridge University Press, 2008.
- [6] I. K. Marchevsky, V. S. Moreva, and V. V. Puzikova, "The efficiency comparison of the vortex element method and the immersed boundary method for numerical simulation of airfoils hydroelastic oscillations," in *VI Int. Conf. on Computational Methods for Coupled Problems in Sci. and Eng. (COUPLED PROBLEMS 2015)*, B. Schrefler, E. Onate, and M. Papadrakakis, Eds., 2015, pp. 800–811. [Online]. Available: [http://congress.cimne.com/coupled2015/frontal/doc/Ebook\\_COUPLED\\_15.pdf](http://congress.cimne.com/coupled2015/frontal/doc/Ebook_COUPLED_15.pdf)
- [7] R. D. Blevins and C. S. Coughran, "Experimental investigation of vortex-induced vibration in one and two dimensions with variable mass, damping, and reynolds number," *J. of Fluids Eng.*, vol. 131, no. 10, pp. 101 202–1–101 202–7, 2009. [Online]. Available: <http://dx.doi.org/10.1115/1.3222904>