

# Comparison of Two Random Vortex Methods and Corrected SLIC Algorithm For Simulation of Combustion Flow

Amir Hossein Derakhshanfard<sup>1</sup>

1. Department of Mechanic, Hamedan Branch, Islamic Azad University, Hamedan, Iran

## ARTICLE INFO

**Keywords:** *Smart Cylinder, FGPM, Asymmetric Loading, Hygrothermo-Magneto-Elastic Response.*

## ABSTRACT

In this research, a combination of two random vortex methods and the corrected SLIC algorithm have been used to simulate the transient combustion flow around the cylinder. This model consists of two parts, in the first part, the random vortex method is used to solve the rotating field, and in the second part, the corrected SLIC algorithm is used to solve the flame propagation. In the initial times, a symmetric region is formed, with the passage of time, the vortices begin to dissipate, when the flow is accompanied by combustion, the vortices that form behind the cylinder are stronger.

## **Introduction**

One of the successful, accurate and at the same time attractive methods in investigating the viscous flow in calm and turbulent regimes, which is used in a wide range of Reynolds numbers, is the random eddy method, which is based on the simulation of the viscous flow inside different bodies using the potential flow theory. And zeroing the speed on the surface of the object is done by eddies, springs, and wells. Flow analysis by random eddy method was first proposed in Alexander Churin's [1] paper in 1973 to model the flow around cylinders. Later, this method was completed and increased the accuracy, among them the article of Bill [2] in 1981 and Benfatto [3] in 1984. Among the recent researches in this field, we can refer to the book of Catt and his colleagues [4] in 2002 and the research of Rama Chandran [5] related to modeling the flow in The channel with sudden expansion and also on the cylinder referred to the method of random vortices. It is also possible to refer to the article of Behrouz Zafarmand and Mohsen Kahoram in relation to the simulation of slow and simultaneous flow inside the channel with sudden expansion [6].

The general principles of the random vortices method to solve the flow are that first the potential flow bed around the cylinder is considered, then in order to satisfy the

verlap, in this way, after making sure of The simulation performed to remove the vortex area and reduce the forces entering the cylinder of the device as suction behind it in different parts of the building is used so that the flow around the cylinder becomes a so-called streamline and has an appearance similar to the flow of potential.

## **Governing equations in random vortices method**

condition that there is no oil on the surface of the cylinder, the generation of the number of vortices with the following conditions is used in each time step. The eddies have zeroed the surface with a horizontal velocity, but on the other hand, a vertical velocity is induced on the surface. In order to make this velocity zero, the number of springs and wells in the shaft is used in such a way that the vertical velocity is also zero. are induced in the potential flow bed at any point of the solution domain, that a flow is simulated with good accuracy similar to the real flow, in fact, in the random vortices method, the fluid motion is well simulated without gridding.

In this article, firstly, the turbulent flow around a cylinder in the first exercise 140000 is modeled with the method of random vortices, and the results obtained are compared with the laboratory results of the study of Contiol and Kales [7], which have a very strong o

In this method, the real fluid flow is simulated in a time-dependent manner in a domain without grid by a large number of vortices, instead of directly solving the Navier-Stokes equation with the effect of the curl operator on this material and combining it with the continuity equation, the vorticity transfer equation, the governing equation It is expressed as follows.

$$\frac{\partial \vec{w}}{\partial t} + (\vec{u} \cdot \nabla) \vec{w} = \frac{1}{Re} \nabla^2 \vec{w} \quad . \quad \vec{w} = \nabla * \vec{u} \quad (1)$$

In the above equation,  $u$  is the fluid velocity vector,  $W$  is the vorticity vector, and  $Re$  is the Reynolds number of the flow. The initial condition of the above equation is the existence of a potential flow around the cylinder and its only boundary condition is the zero velocity on the surface of the cylinder. The vorticity level is of course produced by constant rotation. For this purpose, first divide the surface of the cylinder into the number of sections equal to the length  $h$ . If we denote the tangential velocity of each part of the wall by  $u_t$ , and the rotation required to zero this velocity by

$r_s$ , the maximum rotation value for each vortices is denoted by  $r_{max}$ . The number of vortices in each section will be as follows:

$$M = \frac{|\Gamma_s|}{\Gamma_{max}} = \frac{u_t \cdot h}{\Gamma_{max}} \quad (2)$$

The movement of vortices from the Lagrangian point of view is carried out according to equation (1) with two mechanisms of movement in equation (3) and diffusion in equation (4) [8].

$$\frac{\partial \vec{w}}{\partial t} + (\vec{u} \cdot \nabla) \vec{w} = 0 \quad (3)$$

$$\frac{\partial \vec{w}}{\partial t} = \frac{1}{Re} \nabla^2 \vec{w} \quad (4)$$

The displacement mechanism by calculating the speed of each vortex and the diffusion mechanism also causes the movement of the vortices with the probability of a normal random variable with zero mean and standard deviation. The displacement mechanism of a large number of vortices is in motion and induces velocity on each other and the surrounding space, which is the velocity induced by N vortices on the jth vortex in the form of the following equation [9]:

$$\vec{w}(z_j) = \sum_{i=1, i \neq j}^N \frac{-i\Gamma_i |z_j - z_i|}{\pi \max(|z_j - z_i|, \delta)} \left( \frac{z_j - z_i}{|z_j - z_i|^2} \right) \cdot \delta = \frac{h}{2\pi} \quad (5)$$

As mentioned, we use the number of springs and wells on the surface to zero the vertical velocity on the surface. The induced velocity of the spring and well located at point  $z_j$  on any point  $z_i$  is obtained from the following equation:

$$\vec{w}_s(z_i) = \mp \frac{\alpha(j)}{2\pi} \left( \frac{1}{z_i - z_j} \right) \cdot i \neq j \quad (6)$$

In this way, with the effect of the potential flow, the effect of the eddies on each other, as well as the effect of the spring and the well on the eddies, Verdisite's explanation is formed as a function of temperature by the random collection method. It is distributed in the vertical direction on the hand, if the vortices are placed inside the province, it is removed, but the vortices distributed around the cylinder in the next step are based on both the distribution and displacement mechanisms and also have an effect, and the children's eyes move. If we show the coordinates of each water circle at the moment t in complex coordinates as  $Z_i(t)$ , its position at that moment is  $(t+\Delta t)$  obtained from the following relationship:

$$Z_i(t + \Delta t) = Z_i(t) + [W_p(i) + W_v(i) + W_s(i)]\Delta t + n_i \quad (7)$$

In the following, in order to streamline the fluid flow and reduce the forces on the cylinder, two individual wells are used at different points behind the cylinder according to equation 6, and the lowest power value of the flow chart is introduced as the optimal value, and adding this well At the end of the cylinder, the position of the vortex is obtained from the following equation:

$$Z_i(t + \Delta t) = Z_i(t) + [W_p(i) + W_v(i) + W_s(i) + W_{s-2}(i)]\Delta t + n_i \quad (8)$$

## Results for flow around a cylinder without suction at 14000 Reynolds

The distribution of vorticity and the formation of von Karman vortices behind the cylinder are shown in Figure (1) and the flow lines and average velocity vector are shown in Figure (5).

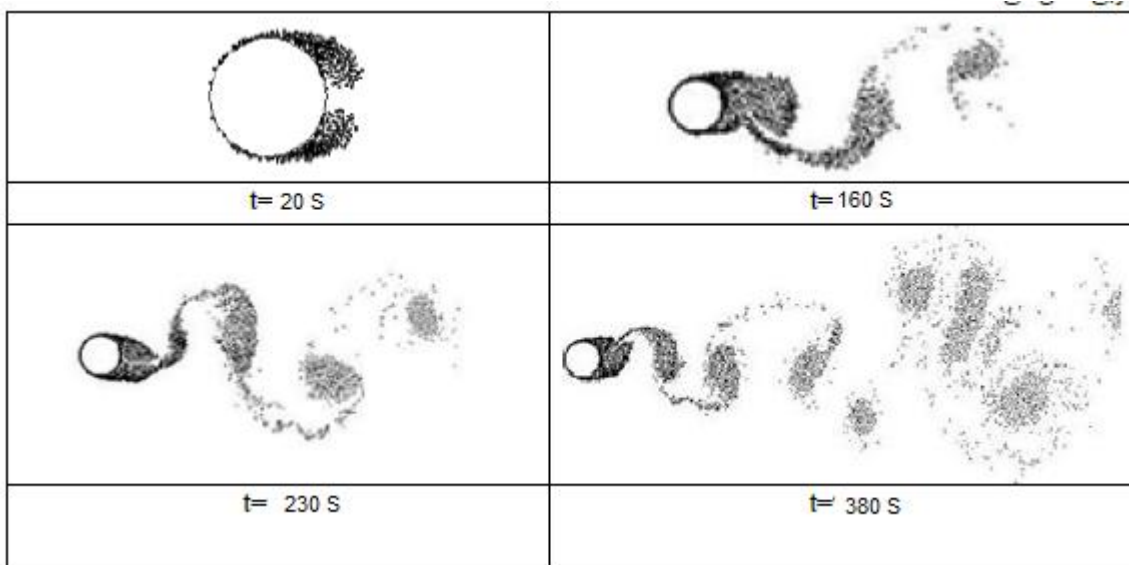


Fig 1. Motion of vortices and formation of von Karman vortices for suction-free flow around a cylinder

After obtaining the description of the data around the cylinder, a solution area is defined and conditioned, and then the speed of the time-dependent moments on the grid points is obtained, which is obtained by averaging the average speed vector over a certain period of time. By comparing these results with the laboratory results related to Cantioi and Kales research, it can be seen that the results are very accurate. These results are shown in figures 2 and 3.

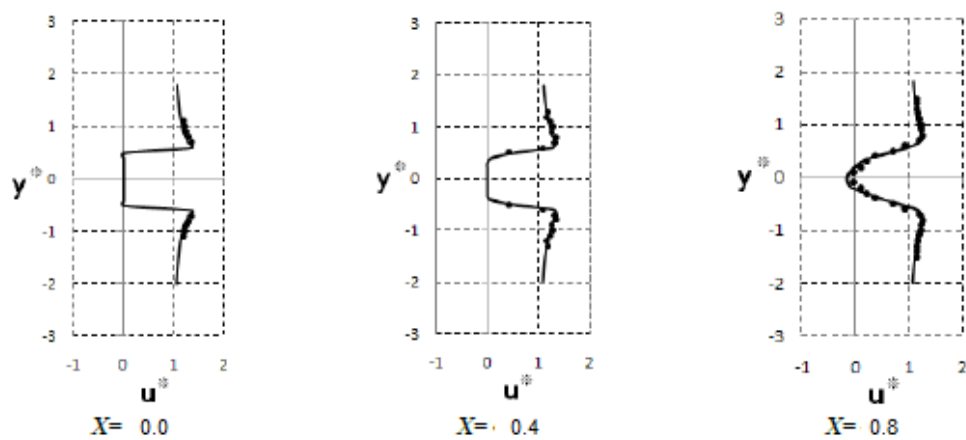


Fig 2. Comparison of average horizontal velocity distribution for program results

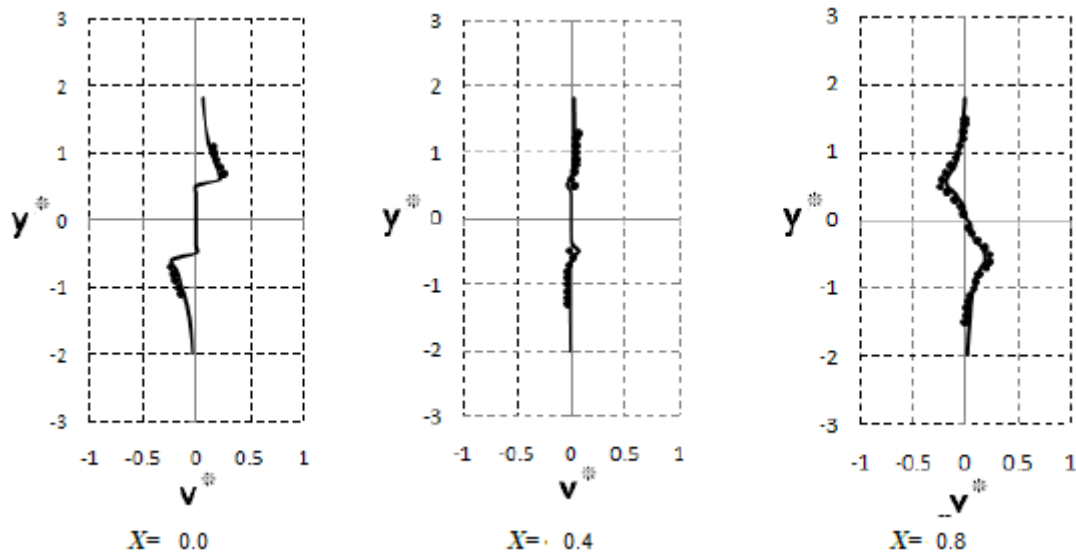


Fig 3. Comparison of the average vertical velocity distribution for the results of the program

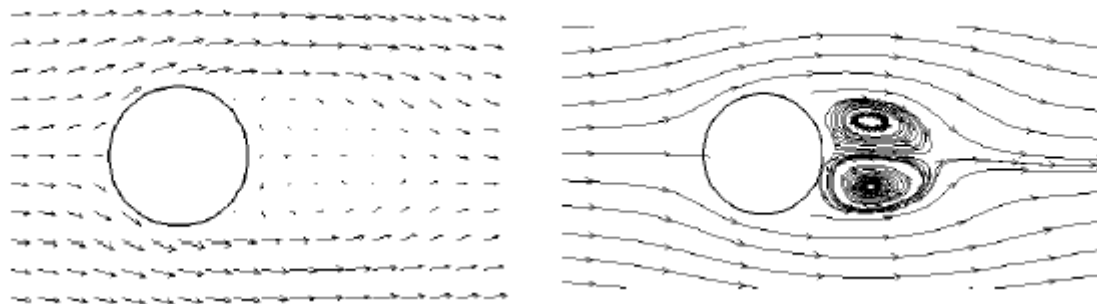


Fig 4. Streamlines and mean velocity vector for no-suction flow around a cylinder at 140,000 Reynolds

### Results for flow around a cylinder with suction at 14000 Reynolds

After running the computer program in non-pause mode and comparing and ensuring the program used by adding a well as Mecca in killing Muslims in meter positions according to Figure 5, the most optimal power in the best position for return flows and reducing incoming forces The province will be introduced.

At this stage, by adding two wells in different positions behind the cylinder as suction, interesting results are obtained. From 11 shekels, the best position of the well is related to the angle of 15 degrees.

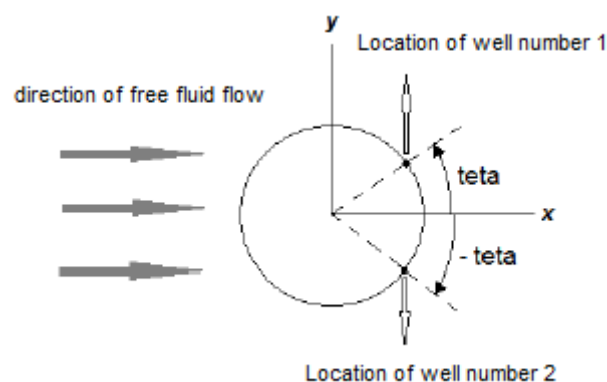


Fig 5. Position the wells to create suction behind the cylinder

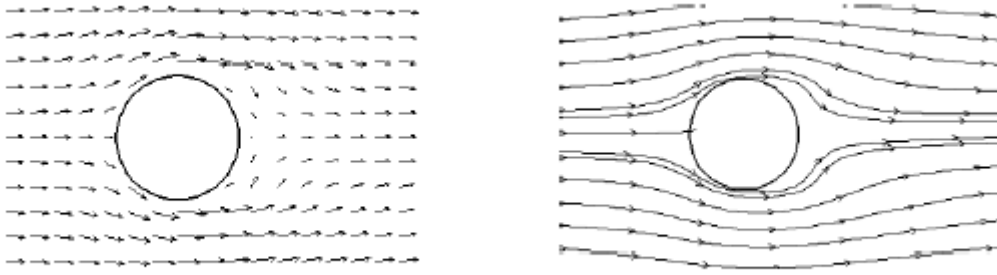


Fig 6. Average velocity vector and streamlines for  $\text{Teta}=0$  and spring strength 0.06

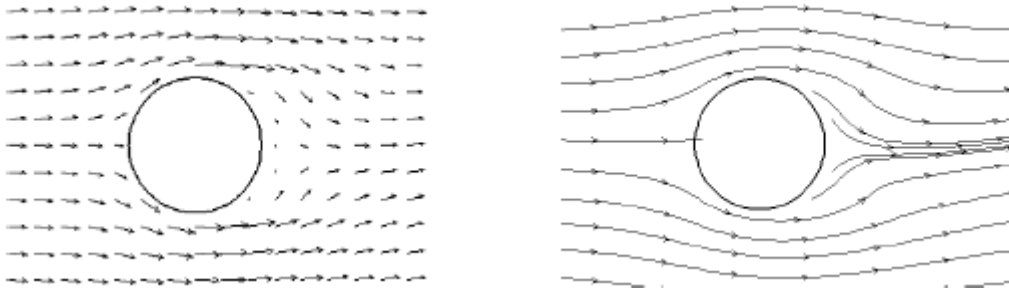


Fig 7. Average velocity vector and streamlines for  $\text{Teta}=15$  and spring strength 0.02

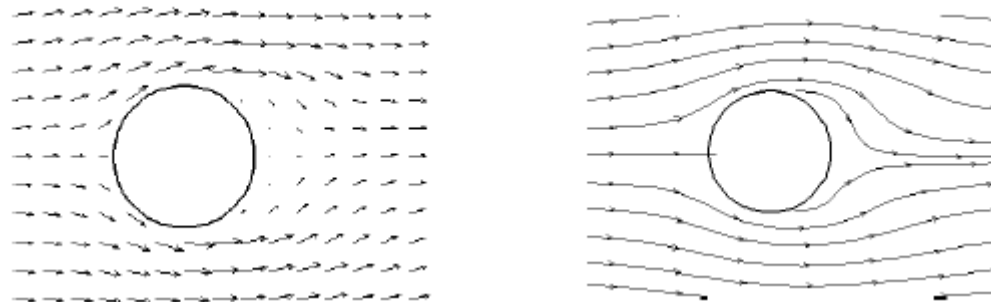


Fig 8. Average velocity vector and streamlines for  $\text{Teta}=30$  and spring strength 0.05

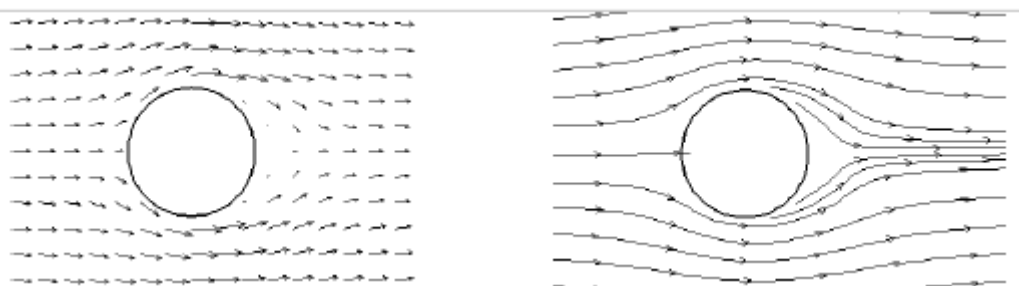


Fig 9. Average velocity vector and streamlines for  $\text{Teta}=60$  and spring strength 0.1

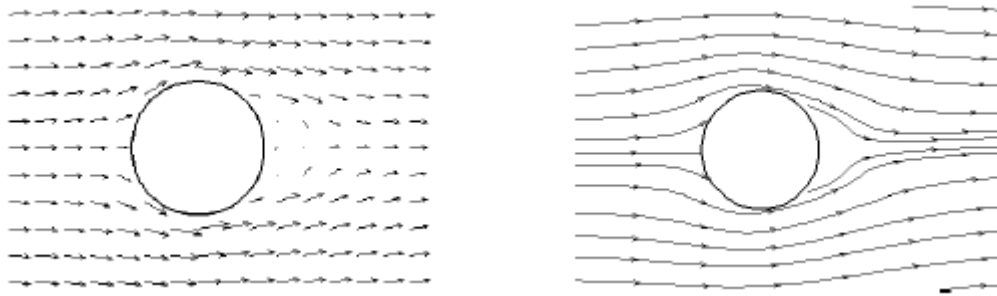


Fig 10. Average velocity vector and streamlines for Theta=90 and spring strength 0.17

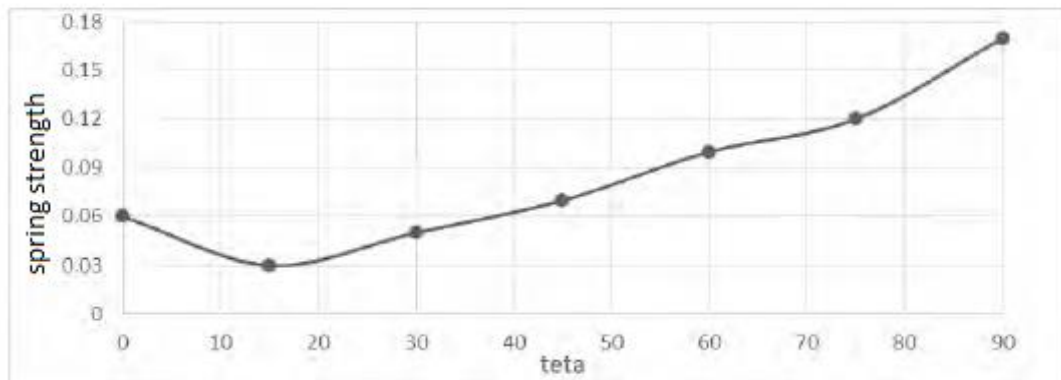


Fig 11. Optimum power changes for the well depending on the location of the well

## Conclusion

The work done in this article with random rotation method is actually different from other numerical solution methods with the explanation that in other methods direct solution of Navier-Stokes equations is used, which usually requires annualization of the equation. Such a requirement to create a network in the area of Helestan has an impact on the accuracy of the answer, but in the method of random vortices, instead of using speed and pressure and their average values in the Navier-Stokes equation, vorticity and its transfer equation are used, and the problem is related to the nonlinear form. The equation without annualization is lost and there is no need to create a network around the cylinder to show the fluid movement. Besides, the obtained speeds are momentary and dependent on time. Average speed is obtained by averaging instantaneous speeds in a certain time period.

Since there are no experimental values for the flow around the cylinder with suction, to prove the correctness of the computer program and the accuracy of the random vortices method, the comparison of the average speed for the flow around the cylinder without suction is used, which has a similar computer program method. May this case be proved in figures 2 and 3.

The purpose of creating suction on the back of the cylinder is to eliminate the vortices behind it and in fact streamline and reduce the forces entering the cylinder. Therefore, different powers are needed for different angles of the well position to achieve this goal.

As seen in the diagram of Figure 11, the best position for the suction direction at Reynolds 140000 is in the angles of 5 to 25 degrees and the worst is in the range of 70 degrees and above. The difference between the minimum required well power, i.e. 0.03 at an angle of 15 degrees, and the maximum, i.e. 0.17 at an angle of 90 degrees, is quite remarkable.

## Reference

1. Chorin, Alexander Joel, Numerical study of slightly viscous flow. *Journal of Fluid Mechanics*, vol 57, part 4, pp 78-796, 1973.
2. Beal J T, Majda A, Rate of convergence for viscous splitting of Navier Stokes equation. *Journal of Mathematics Computation*, vol 37, pp 243, 1981.
3. Benfatto G, Pulvirenti M, Convergence of vorticity near the boundary in planar Navier Stokes flows. *Comm Math Phys*, vol 96, pp 59, 1983.
4. Cottet G H, Koumoutsakos, *Vortex method (Theory and Practice)*. McGraw Hill, 2000.
5. Ramachandran P, Development and study of high resolution two dimensional random vortex method. *Indian Institute of Technology, Madras, Chennai*, 4002.
6. Cantwell B J, Coles D, A flying hot wire study of the turbulent near wake of a circular cylinder at  $Re=140000$ . Ph.D. thesis, *California Institute of Technology*, 1976.
7. Prabhu R, Development and study of a high resolution two dimensional random vortex method . Ph.D. These, *Department Aerospace Engineering Indian Institute of Technology*, 2004.
8. Chorine A J, Vortex sheet approximation of boundry layers. *Journal of Computational Physics*, Vol 27, pp 428, 1978.
9. Chawdhury, S., Morgenthal, G., 2015. Flow reproduction using vortex particle methods for wake buffeting analysis of bluff structures. In: *ICFMA 2015: 17th International Conference on Fluid Mechanics and Applications*.
10. Cottet, G.H., Koumoutsakos, P.D., 2000. *Vortex Methods: Theory and Practice*. Cambridge University Press, Cambridge.
11. Ge, Y., Xiang, H., 2008. Computational models and methods for aerodynamic flutter of long-span bridges. *J. Wind Eng. Ind. Aerodyn.* 96, 1912–1924.
12. Hejlesen, M.M., Rasmussen, J.T., Larsen, A., Walther, J.H., 2015. On estimating the aerodynamic admittance of bridge sections by a mesh-free vortex method. *J. Wind Eng. Ind. Aerodyn.* 146, 117–127.
13. Ibrahim, K., Morgenthal, G., Chawdhury, S., 2014. Wake flow reproduction in vortex particle methods for simulating buffeting response. In: *The 6th International Symposium on Computational Wind Engineering—CWE2014*.
14. Irwin, P.A., 2008. Bluff body aerodynamics in wind engineering. *J. Wind Eng. Ind. Aerodyn.* 96, 701–712.
15. Koumoutsakos, P., Leonard, A., 1995. High-resolution simulations of the flow around an impulsively started cylinder using vortex methods. *J. Fluid Mech.* 296, 1–38.
16. Larsen, A., Walther, J.H., 1997. Aeroelastic analysis of bridge sections based on discrete vortex simulations. *J. Wind Eng. Ind. Aerodyn.* 67–68, 253–265.
17. Larsen, A., Walther, J.H., 1998. Discrete vortex simulation of flow around five generic bridge deck sections. *J. Wind Eng. Ind. Aerodyn.* 77, 591–602.
18. Liu, Z., Kopp, G.A., 2011. High-resolution vortex particle simulations of flows around rectangular cylinders. *Comput. Fluids* 40, 2–11.
19. McRobie, A., Morgenthal, G., Abrams, D., Prendergast, J., 2013. Parallels between wind and crowd loading of bridges. *Philos. Trans. R. Soc. Lond. A: Math. Phys. Eng. Sci.* 371, 20120430.
20. Morgenthal, G., 2002. *Aerodynamic Analysis of Structures using High-resolution Vortex Particle Methods* (Ph.D. thesis), *University of Cambridge*.



21. Morgenthal, G., 2005. Advances in numerical bridge aerodynamics and recent applications. *Struct. Eng. Int. (IABSE)* 15, 95–100. Morgenthal, G., Corriols, A.S., Bendig, B., 2014. A GPU-accelerated pseudo-3D vortex method for aerodynamic analysis. *J. Wind Eng. Ind. Aerodyn.* 125, 69–80.