25th ABCM International Congress of Mechanical Engineering
October 20-25, 2019, Uberlândia, MG, Brazil

COB-2019-0494
DEVELOPMENT OF A PRESSURE FORMULATION TO CONSIDER FRICTION DRAG IN THE VORTEX METHOD

Renan Galhardo Fagundes Renoldi
Alex Mendonça Bimbato
Department of Energy, São Paulo State University (UNESP), Guaratinguetá, Brazil
alex.bimbato@unesp.br , galhardo84@gmail.com

Luiz Antonio Alcântara Pereira
Mechanical Engineering Institute, Federal University of Itajubá (UNIFEI), Itajubá, Brazil
luizantp@unifei.edu.br

Abstract. *In the present paper the vortex method is used in conjunction with panel method to study the two-dimensional, incompressible, subsonic flow with constant properties around a four digits NACA airfoil. Using this lagrangian approach, the vorticity dynamic is studied following discrete vortices at each time step of the numerical simulation; with this purpose the vorticity transport equation is satisfied by an advection and a diffusion dislocation of each discrete vortex used to represent the vorticity field. Flat panels with linear vorticity distribution are used to represent the airfoil geometry. The main contribution of this paper is to develop a pressure formulation that is able to take into account the friction drag which is dominant in problems that involve flow around slender bodies with low angles of attack. The results will be compared with experimental ones available in the literature.*

Keywords: *friction drag, airfoil, vortex method, panel method, Lagrangian approach*

1. INTRODUCTION

The two-dimensional simulation of fluid flows through different types of bodies, submitted to several different conditions has been subject of interesting studies developed by the scientific community for over 50 years. These important studies result in computational and experimental advances in the fields of acoustic science, applied mathematics, material sciences and many others.

The Navier-Stokes equations capable of modeling fluid flows through the determination of velocity and pressure fields in the beginning of the 19^{th} century, plays an important role for modern fluid dynamics. It is important to notice that these equations have no general solutions, although with assumption of hypothesis and simplifications important solutions can be achieved by numerical methods.

Different fluid phenomena have been studied, both numerically and experimentally, such as the study of the heaving of symmetrical, rigid and flexible airfoils to induce movement (Im, 2017). Ground effect has also been studied, which consists of the influence of the boundary layer formed by a plane wall near the studied body, generating an air cushion (Moura, 2007). Zhang (2006) revised many progresses for both experimental and numerical simulations for race car elements. The review was directed to fundamental aerodynamics instead of practical applications. It was observed many aerodynamic phenomena such as separation, shear layer instability, wall jet, and others.

As one can see, the flow around airfoils is of great interest in Fluid Mechanics. A wide range of situations can be studied (Yemenici, 2014; Şahin & Acir, 2015). Over the years, several publications have used the Panel Method to solve potential flows around arbitrary shapes. Lewis (1981) coupled the Discrete Vortex Method and the Panel Method to simulate viscous flows around complex geometries. Su & Kinnas (2015) presented the application of the Panel Method to evaluate the performance of a marine impeller subjected to the cavitation effect.

Recently, Han (2016) analyzed the friction drag contribution for the total drag coefficient for a three-dimensional numerical simulation, over three different types of bodies: a flat plate, a rectangular section and the Sutong Bridge section (a bridge with a span of 1088m) under different wind angles of attack. The author evaluated the simulation through a Reynolds Averaged Navier-Stokes equations and grid meshes. The results imply the more streamlined the surface is, higher the contribution of friction drag over the total drag coefficient; besides, it was also discovered that the larger the aspect ratio or smaller the fairing angle is, the contribution of the friction drag for the total drag increases; finally it was registered that the Reynolds number has small influence over the friction drag and its effects can even be ignored.

Most of modern Computational Fluid Dynamic (CFD) techniques modulate fluid flows through meshed eulerian methods, which calculates fluid properties for each point of the whole mesh. This form of dealing with the problem requires a high amount of mesh refinement for precise results and this procedure can lead to several instability issues and a high amount of computational power. From this perspective a study based on the analysis of vorticity over a two-dimensional airfoil with the discrete vortex method, which requires the tracking of each vortex particle is interesting since the computational efforts are directed only to the regions where vorticity is present.

For this present study, the Panel Method (Katz & Plotkin, 1991) plays a fundamental role of the solution of the potential problem. It is used to represent any known form of body through series of panels. The Discrete Vortex Method was created to overcome the limitations imposed by the potential analysis. This meshfree method is able to evaluate the viscous effects of the fluid flow. Sarkar & Venkatraman (2008) used this technique to study the behavior of an oscillatory NACA0012 airfoil operating under dynamic stall regime. Hu et al. (2015) modeled a wind turbine using the Discrete Vortex Method; the Panel Method modeled the surface vortex sheets while the Discrete Vortex Method simulated the vorticity generated on the blade.

The Discrete Vortex Method was first introduced by Chorin (1973), which observed the process of vorticity generation and dispersal through a random algorithm using the Navier-Stokes equations in two-dimensional form. The algorithm consists on the assumption of the separation of the viscous part of the vorticity transport equation. From this point, the advection effects of the vorticity can be evaluated independently of the diffusion effects of this propriety, however, within the same time increment in a way of simplifying the numerical implementation of the vortex method.

In this present paper, the focus of attention is the friction drag contribution to the drag coefficient over a symmetrical airfoil; it is known that for this kind of problems the friction drag is dominant. This new pressure formulation will take into account the vorticity field present in the fluid domain and the obtained results will be compare to the experimental ones available in the literature.

2. GOVERNING EQUATIONS AND BOUNDARY CONDITIONS

Fig. 1 shows a two-dimensional, incompressible, viscous flow around an airfoil. The fluid mainstream has a velocity U_∞^* and its domain Ω is defined by the surface $S = S_1 \cup S_2$, where S_1 and S_2 are the body surface and a far away boundary, respectively. The airfoil has a chord c^* and an angle of attack θ with respect to the flow.

The governing equations for the situation illustrated in Fig. 1 are the continuity and the Navier-Stokes equations. In furtherance of making the problem non-dimensional, the quantities U^*, c^* and the ratio $U^* \;/ c^*$ are employed as velocity, length and time scales respectively. Doing so, the governing equations and the boundary conditions (impermeability and no-slip conditions) becomes:

$$\nabla.\boldsymbol{V} = 0 \tag{1}$$

$$-\nabla p + \frac{1}{Re}\nabla^2\boldsymbol{V} = \frac{\partial \boldsymbol{V}}{\partial t} + \boldsymbol{V}.\nabla\boldsymbol{V} \tag{2}$$

$$V_n^p - V_n^{S_1} = 0 \tag{3}$$

$$V_t^p - V_t^{S_1} = 0 \tag{4}$$

where $\boldsymbol{V}$ and p are the velocity and pressure fields, $Re = (U^* c^*)/\upsilon$ is the Reynolds number (with ν being the kinematic viscosity), V_n^p and V_t^p are the normal and tangential velocity components of a fluid particle in contact with the surface, $V_n^{S_1}$ and $V_t^{S_1}$ are the normal and tangential velocity components of the airfoil surface.

3. NUMERICAL SOLUTION

This work couples the Discrete Vortex Method with the Panel Method, to numerically solve Eq. (1) and Eq. (2) subjected to the boundary conditions of Eq. (3) and Eq. (4).

The Panel Method (Katz & Plotkin, 1991) provides the geometry discretization through a series of flat panels with linearly varying strength vortex distribution.

The Discrete Vortex Method simulates the flow vorticity field solving the vorticity transport equation in its 2-D form (Batchelor, 1967):

$$\frac{\partial \omega}{\partial t} + (\mathbf{V}.\nabla)\omega = \frac{1}{Re}\nabla^2\omega \tag{5}$$

Eq. (5) shows that the discrete vortices move through the flow field by advection (left hand side) and by diffusion (right hand side). Chorin (1973) proposed a split algorithm, in which the diffusion and advection terms are solved separately at the same time-step of the numerical simulation.

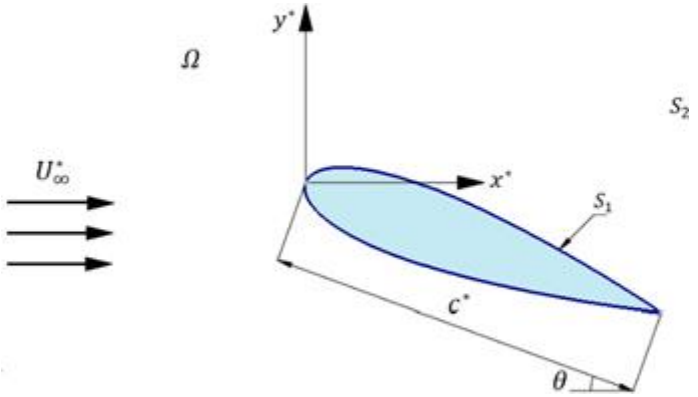


Figure 1. Flow past an airfoil

3.1 Advection of vorticity

The first advance of one of the nascent discrete vortex is necessarily performed with a first-order Euler scheme on Eq. (6). All other discrete vortex move using second-order Adams-Bashford scheme on Eq. (7).

$$p_{adv_k}(t+\Delta t) = p_{adv_k} + u_{t_k}(t)\Delta t \quad (6)$$

$$p_{adv_k}(t+\Delta t) = p_{adv_k} + [1{,}5u_k(t) + 0{,}5u_k(t-\Delta t)]\Delta t \quad (7)$$

where u_{t_k} and p_{adv} are induced velocity and position, respectively, of vortex *k* due to advection process. The velocity u_{t_k} is obtained from tree velocities represented on Eq. (8). The first is mainstream velocity, u_m presented by Eq. (9); the second is a velocity due to the boundary (panel method), u_b, presented by Eq. (10); and velocity induced by discrete vortices, obtained through Biot-Savart law, u_v, presented by Eq. (11).

$$u_{t_k} = u_m + u_b + u_v \quad (8)$$

$$u_{m_1} = 1 \quad u_{m_2} = 0 \quad (9)$$

$$u_{B_k}^n = \sum_{i=1}^{m} \gamma_i . U_{B_{k,j}}^n \qquad n = 1{,}2 \;\; and \; k = 1, z \quad (10)$$

$$u_{V_k}^n = \sum_{i=1}^{m} \Gamma_i . U_{V_{k,j}}^n \qquad n = 1{,}2 \;\; and \; k = 1, z \quad (11)$$

where *z* is the number of vortices on fluid domain, *m* is the number of panels used to discretization of airfoil, γ_i is the linear-strength vortex panel distribution, Γ_j is the intensity of the vortex *j*, $U_{B_{k,i}}^n$ is the n-th component of velocity induced on vortex *k* by panel *i* and $U_{V_{k,j}}^n$ is the n-th velocity component induced on vortex *k* by a discrete vortex *j*.

3.2 Diffusion of vorticity

The diffusion of vorticity is developed by Chorin (1973) based on brownian movement of Einstein (1956).

$$p_{k_d}(t) = \sqrt{\frac{4.\,\Delta t}{Re} \ln\frac{1}{P}} \;\; [cos(2\pi Q) + \sin(2\pi Q)] \quad (12)$$

3.3 Aerodynamic Loads

3.3.1 Pressure Formulation

The pressure formulation used in this work is the same one used by Carvalho Júnior et al. (2017) to study the flow around a NACA 0012 airfoil. The mentioned pressure formulation is an extension of Shintani and Akamatsu (1994) method. Ricci (2002) used Shintani and Akamatsu (1994) formulation to determine the pressure coefficient acting on a panel. With this propose, Eq. (13) was numerically implemented by Ricci (2002):

$$\xi \tilde{Y}_i + \int_{S_1} \frac{1}{2\pi} \frac{n_x(x-x_i) + n_y(y-y_i)}{(x-x_i)^2 + (y-y_i)^2} \tilde{Y}\, dS = -\int_{\Omega} \frac{1}{2\pi} \frac{v(x-x_i) - u(y-y_i)}{(x-x_i)^2 + (y-y_i)^2} \omega\, d\Omega - \frac{1}{Re} \int_{S_1} \frac{n_y(x-x_i) - n_x(y-y_i)}{(x-x_i)^2 + (y-y_i)^2} \omega\, dS \tag{13}$$

where ξ is a constant that is 0.5 on body surface and 1 in fluid domain. The unit vector n has horizontal component n_x and vertical component n_y. Finally, $\tilde{Y}$ is specific work defined by $\tilde{Y} = p - p_\infty \frac{1}{2}[u^2 - 1]$. The first integral is the body contribution in the pressure calculation; the second integral is the effect of vorticity on fluid domain; the last integral is the influence of vorticity located on body's surface.

According to Ricci (2002), the pressure coefficient can be obtained by:

$$C_p = 2\tilde{Y} + 1 \tag{14}$$

3.3.2 Friction Formulation

The main contribution of this work is to develop a formulation that takes into account the friction drag. The formulation described in this section must be additioned to the one presented in Section 3.3.1.

In two-dimensional flows the relation between vorticity and velocity fields can be expressed by Eq. (15) and, using it on Newton's law of viscosity, Eq. (16) is obtained:

$$\omega = -\frac{\partial u}{\partial y} \tag{15}$$

$$\tau = -\mu . \omega \tag{16}$$

where u is the horizontal velocity component, ω is the vorticity and μ is the dynamic viscosity.

Eq. (16) can be rewrite in non-dimensional form assuming the form of Eq. (17).

$$\tau^* = \frac{-2.\omega^*}{Re} \tag{17}$$

where symbol * corresponds to non-dimensional physical quantities and will be omitted from this point of the text.

The vorticity in a point can be calculated by (Kundu, 1990):

$$\omega = \sum_{p=1}^{z} \frac{\Gamma_p}{\pi \sigma_p^2} . exp\left(\frac{-r^2}{\sigma_p^2}\right) \tag{18}$$

Thus, each discrete vortex induces vorticity in the control point of each panel used to represent the body surface (see Fig. (2). In Eq. (18), r is the distance between a discrete vortex and a control point and σ is the Lamb vortex core (Mustto, 1998).

As consequence, the shear stress on the body surface is:

$$\tau_j = \frac{-2}{Re} \sum_{p=1}^{z} \frac{\Gamma_p}{\pi {\sigma_p}^2} . exp\left(\frac{-{r_{p,j}}^2}{{\sigma_p}^2}\right) \tag{19}$$

where τ_i is the shear stress on panel j obtained through contribution of each discrete vortex (p) presented in the fluid domain.

In the work developed by Ricci (2002), drag and lift coefficients are obtained only through pressure integration in each panel. In the present work the friction contribution is considered too. In Fig. 2, D is the drag force, L is the lift force, P is the pressure and τ is the friction.

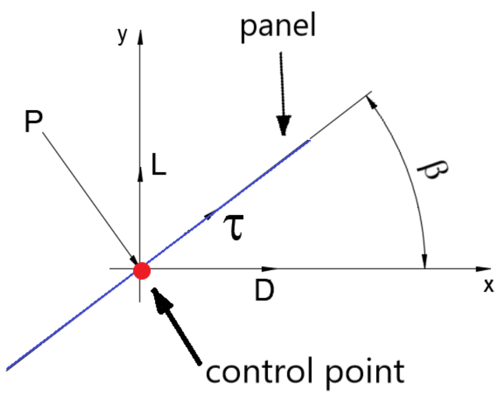


Figure 2. Forces acting on a panel

Finally, using pressure and friction formulations, it is possible to calculate drag and lift coefficients as:

$$c_D = \sum_{i=1}^{m} \left[c_{P_i} . sin\beta_i + \tau_i \; cos\beta_i \right] . \Delta s_i \tag{20}$$

$$c_L = \sum_{i=1}^{m} \left[-c_{P_i} . cos\beta_i + \tau_i \; sin\beta_i \right] . \Delta s_i \tag{21}$$

where c_D is the drag coefficient, c_L is the lift coefficient and Δs_i is the panel length.

4. RESULTS

In order to validate the friction formulation implemented in this work, the numerical results with and without the friction formulation is presented for the NACA 0012 aerodynamic profile with zero angle of attack. However, before start the simulations it is necessary to define parameters like number of flat panels used to represent the airfoil, the time increment, the Reynolds number and the Lamb vortex core. These parameters are presented on Table 1.

Table 1. Numerical parameters used in the numerical simulations with and without the friction formulation.

Parameter	Symbol	Value
Number of Panels	m	200
Time-step increment	Δt	0.01
Total number of time-step increment	Kt	1000
Lamb vortex core	σ	0.019
Reynolds number	Re	170000

The number of flat panels used to represent the airfoil and the numerical time-step defines how much accurate the model is. On the other hand, the Lamb vortex core is a parameter that needs to be founded by numerical experiments.

Fig. 3 presents the pressure distribution around the aerodynamic profile. The pressure coefficient was obtained between non-dimensional time 5 and 10.

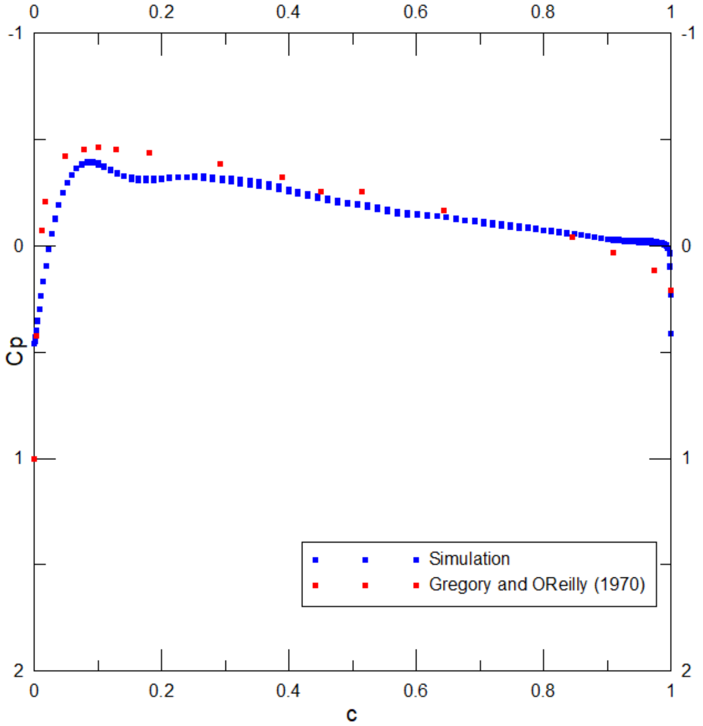


Figure 3. Pressure distribuition over NACA 0012 (θ=0° and Re = 170000)

Observing Fig. 3, it can be seen that pressure coefficient has close values for almost all the chord when compared with experimental results. However, in trailing edge the value is overestimated while for rest of the chord the value is less than experimental.

Probably, if the airfoil was discretized in a greater number of panels, the result for all chord could be better, though, the computational work would be greater.

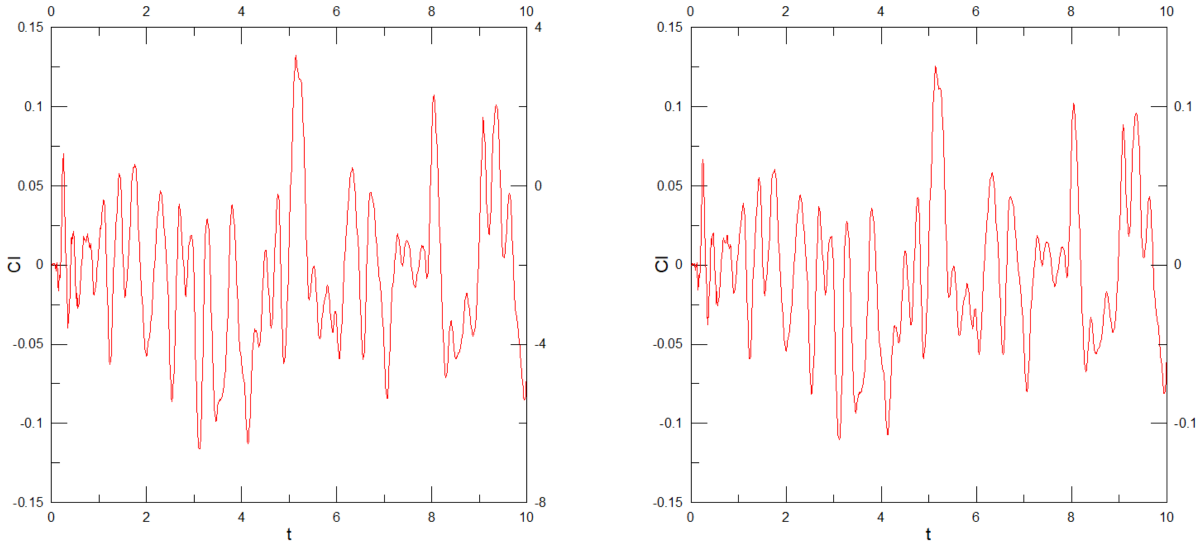
At the end of simulation, the number of discrete vortex on fluid domain is 200000 and the Fig. 4 shows the viscous wake in the end of the simulation.

Figure 4. Viscous wake on NACA 0012 (θ=0° and Re = 170000)

Fig. 5 shows the lift coefficient (C_L) plotted between the non-dimensional time 0 and 10, but the average value is obtained between non-dimensional time 5 and 10. As NACA 0012 is a symmetrical airfoil and considering a zero angle of attack, as expected the value of lift coefficient is zero.

Fig. 6 shows the drag coefficient (C_D) plotted between the non-dimensional time 0 and 10, but the average value is obtained between non-dimensional time 5 and 10 in order to overcome the numerical transient.

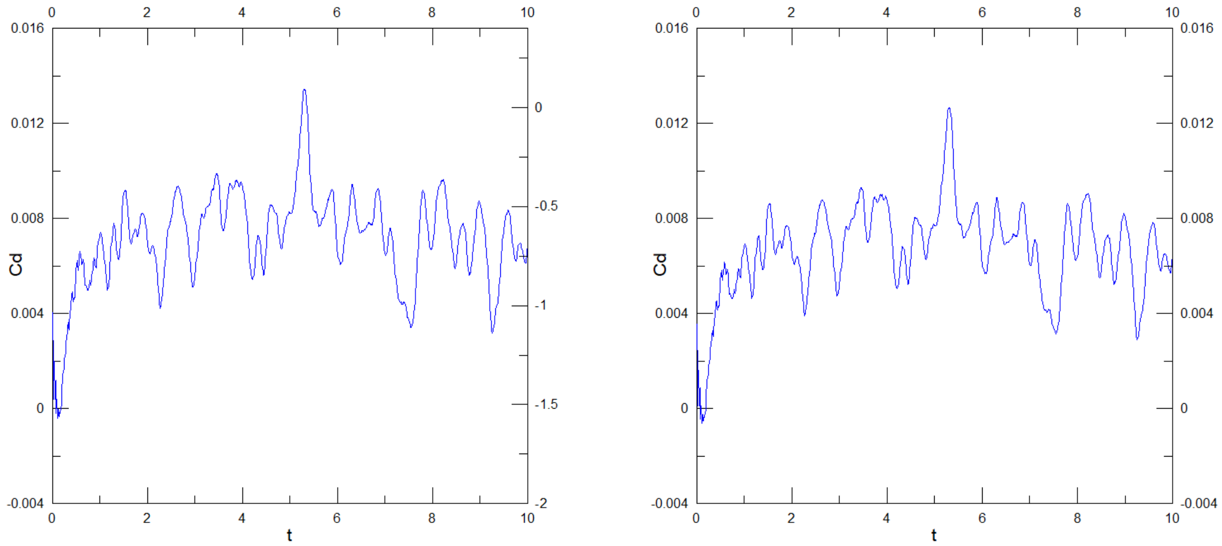
Gregory and OReilly (1970) work has not the value of drag coefficient to compare with the performed simulations, but it can be expected that the drag coefficient is close to zero.



a) Without Friction ($\overline{C_L}$ = 0.00358) b) With Friction ($\overline{C_L}$ = 0.00340)

Figure 5. Lift coefficient for NACA 0012 (θ=0° and *Re* = 170000)

Comparing the results with and without friction formulation, it is observed that they are too close. In fact, the mean drag coefficient with and without friction formulation has a difference of 1%.



a) Without Friction ($\overline{C_D}$ = 0.00745) b) With Friction ($\overline{C_D}$ = 0.00697)

Figure 6. Drag coefficient for NACA 0012 (θ=0° and *Re* = 170000)

5. CONCLUSION

In this work was presented a Lagrangian method called Discrete Vortex method to simulate the viscous flow around the NACA 0012 airfoil with zero angle of attack. The main contribution of this work is the development of a friction formulation to be added to the pressure formulation that already exists (Shintani and Akamatsu, 1994).

As one can observe, the pressure coefficient along the chord is close comparing the numerical and the experimental results; as expected, in leading edge and trailing region the numerical results disagree with experimental ones. These regions are difficult to represent accurately because the presence of high velocity gradients.

In future works it is desire to test the implementation developed here with other aerodynamic geometries, including non-zero angles of attack.

6. ACKNOWLEDGEMENTS

This study was supported by the São Paulo Research Foundation (FAPESP), Proc. 2018/00856-6.

7. REFERENCES

Batchelor, G., 1967. *An Introduction to Fluid Dynamics*, Cambridge University, New Delhi Press, 1st edition.

Carvalho Júnior, C. F., Bimbato, A. M., Alcântara Pereira, L. A., 2017. "Numerical study of the flow around slender bodies using a lagrangian vorthex method". In 24th ABCM International Congress of Mechanical Engineering, December 3-8, Curitiba, PR, Brazil.

Chorin, A.J., 1973. "Numerical Study of Slightly Viscous Flows", Jounal of Fluid Mechanics, Vol. 57, pp. 785-796.

Einstein, A. 1956, *Investigation on the Theory of Brownian Motion*.

Gregory, N., O'Reilly, C.L., 1970. "*Low-Speed Aerodynamic Characteristics of NACA0012 Airfoil Section, Including the Effects of Upper-Surface Roughness Simulating Hoar Frost*", Aeronautical Research Council Reports and Memoranda, R&M 3726, London.

Han, Y., Chen, H., Cai, C.S., Xu, G., Shen, L., Hu, P., 2016. "Numerical analysis on the difference of drag force coefficients of bridge deck sections between the global force and pressure distribution methods", *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 159, pp. 65-79.

Hu, H., Gu, B., Zhang, H., Song, X., Zhao, W., 2015. "Hybrid Vortex Method for the Aerodynamic Analysis of Wind Turbine", *International Journal of Aerospace Engineering.*

Im, S., Park, S.G., Cho, Y., Sung, H.J., 2018. "Schooling behavior of rigid and flexible heaving airfoils", *International Journal of Heat and Fluid Flow*, Vol. 69, pp. 224-233.

Katz, J., Plotkin, A., 1991. *Low-Speed Aerodynamics: from Wing Theory to Panel Method*, McGraw-Hill, Singapore, 1st edition.

Kundu, P. K. (1990), *Fluid Mechanics*, Academic Press.

Lewis, R.I., 1981. "Surface Vorticity Modeling of Separated Flows from Two-Dimensional Bluff Bodies of Arbitrary Shape". *Journal of Mechanical Engineering and Sciences*, Vol. 23.

Moura, W.H., 2007. *Analysis of the Flow around an Oscillating Circular Cylinder in the Vicinity of a Ground Plane*. M.Sc. Dissertation, Mechanical Engineering Institute, UNIFEI, Itajubá, Brazil (in Portuguese).

Mustto, A.A. 1998. "Simulação Numérica do Escoamento em torno de um Cilindro Circular com e sem Rotação Utilizando o Método de Vórtices". M. Sc. Dissertation, UFRJ, Rio de Janeiro, Brazil (in Portuguese).

Ricci, J. E. R., 2002. "Simulação Numérica do Escoamento ao redor de um Corpo de Forma Arbitrária, Estacionado nas Imediações de uma Superfície Plana, com o Emprego do Método de Vórtices". Ph.D. thesis, Escola Federal de Engenharia de Itajubá, Itajubá.

Şahin, I., Acir, A., 2015. "Numerical and Experimental Investigation of Lift and Drag Performances of NACA 0015 Wind Turbine Aerofoil". *International Journal of Materials, Mechanics and Manufacturing*, Vol. 03, pp. 22-25.

Sarkar, S., Venkatraman, K., 2008. "Influence of pitching angle of incidence on the dynamic stall behavior of a symmetric airfoil", *European Journal of Mechanics*, Vol. 27, pp. 219-238.

Shintani, M., Akamatsu, Y., 1994. "Investigation of Two-Dimensional Discrete Vortex Method with Viscous Diffusion Model", *Computational Fluid Dynamics Journal*, Vol. 3, pp. 237-254.

Su, Y., Kinnas, S.A., 2015. "Application of a Panel Method in Performance Prediction of Cavitating and Non-Cavitating Tip Loaded Propellers". In *9th International Symposium on Cavitation.*

Yemenici, O., 2014. "An Experimental Study on the Aerodynamics of a Symmetrical Airfoil with Influence of Reynolds Number and Attack Angle". In *2nd International Conference on Research in Science*, Engineering and Technology, Dubai.

Zhang, X., Toet, W., Zerihan, J., 2006. "Ground Effect Aerodynamics of Race Cars", *Applied Mechanics Reviews*, Vol. 59, pp. 33-49.

8. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.