24th COBEM - 2017

ABCM Associação Brasileira de Engenharia e Ciências Mecânicas

24th ABCM International Congress of Mechanical Engineering
December 3-8, 2017, Curitiba, PR, Brazil

COBEM-2017-0432
NUMERICAL STUDY OF THE FLOW AROUND SLENDER BODIES USING A LAGRANGIAN VORTEX METHOD

Carlos Felipe de Carvalho Junior
Alex Mendonça Bimbato
Departament of Energy, São Paulo State University (UNESP), Guaratinguetá, Brazil
fcj.carlos@gmail.com, alexbimbato@feg.unesp.br

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

Abstract. *This work presents the development of a computational program for the numerical study of a two-dimensional, incompressible flow, with constant properties, around airfoils. A Lagrangian approach, which associates the Discrete Vortex Method and the Panel Method, is employed for the analysis of both potential and the viscous flows over a NACA0012 airfoil. The Panel Method is applied in the geometry discretization and the potential flow solution. The airfoil surface is represented by series of flat panels with linearly varying strength vortex distribution, which satisfies the impermeability boundary condition. At each time step of the numerical simulation, discrete vortices are generated near the airfoil surface in order to satisfy the no-slip boundary condition and also to represent the flow boundary layer. These Discrete Vortices are transported to the flow field by advection and by diffusion. The fluid-dynamic loads are calculated and the numerical results are compared with experimental data.*

Keywords: *discrete vortex method, panel method, Lagrangian approach, flow over airfoils*

1. INTRODUCTION

The flow around airfoils is of great interest in a wide range of situations, such as turbomachinery rotors, the flow around bodies subjected to the ground effect and the stall regime of aerodynamic profiles. Examples of recent publications are Yemenici (2014), Şahin and Acir (2015) and Wang *et al.* (2015).

In order to study the potential solution of such flows, Hess and Smith (1967) developed the Panel Method, which is a numerical scheme used for the analysis of potential subsonic or supersonic flows around arbitrary surfaces. Although this method does not include the effects of fluid viscosity, it can yield satisfactory results, as pointed out by Su and Kinnas (2015), where the authors used the Panel Method to evaluate the performance of a marine impeller subjected to the cavitation effect.

The Discrete Vortex Method is a Lagrangian numerical approach used in the simulation of fluid flows in which the effect of the fluid viscosity must be considered. In such method, each fluid particle is individually followed during the numerical simulation, granting it the great advantage of being a mesh-free technique. Moreover, the Discrete Vortex Method is mainly concerned with the discretization of the fluid vorticity field, which is then used in conjunction with the fluid mechanics version of the Biot-Savart Law (Batchelor, 1967), to compute the fluid velocity field. By using this approach the Discrete Vortex Method is able to use computational elements only where the vorticity is not zero (saving computational time) and also fulfill the boundary conditions even in far away distances from the studied body.

Historically, the first simulations with the Discrete Vortex Method began in 1930. More recently, the modern development of this technique began with Chorin (1973). Recently, Cottet *et al.* (2014) presented a model for active and passive control of the flow boundary layer and viscous wake using the Discrete Vortex Method. Both two and three-dimensional flows were considered, and a circular cylinder was used as reference geometry.

With respect the flow analysis of slender bodies, the Discrete Vortex Method has been used by Sarkar and Venkatraman (2008), while studying the behavior of a NACA0012 airfoil submitted to an oscillatory movement pattern in a stall regime. In such case, the evolution and growth of separation bubbles at the leading and trailing edge has been simulated in a body discretized using 80 panels, with 5 sub panels each, summing 400 discrete vortices generated at each time step of the numerical simulation.

Hu *et al*. (2015) presented a work where a wind turbine has been modeled. The Panel Method was used as surface vortex sheets, and the Discrete Vortex Method simulated the vortex generated on the surface of a NREL Phase VI wind turbine blade. According to the authors, the method showed good results, with errors of 4% and 3.4% for the calculated values of the torque and axial thrust coefficients, respectively.

2. GOVERNING EQUATIONS

Figure 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 faraway boundary, respectively. The airfoil has a chord $c^*$and an angle of attack θ with respect to the flow.

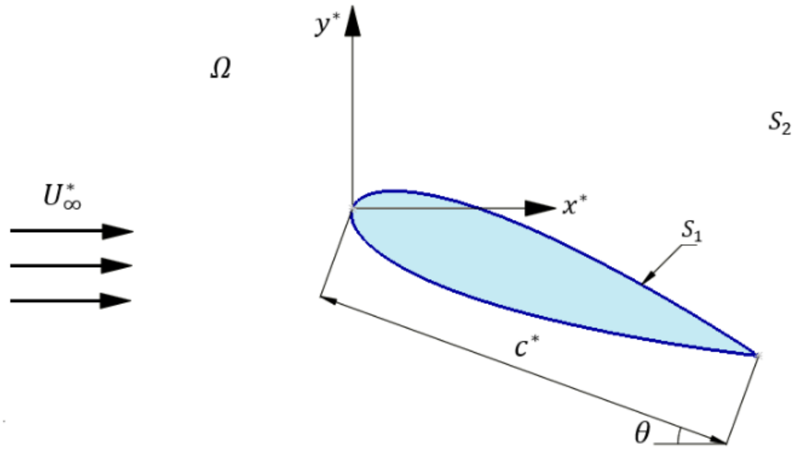

Figure 1 – Flow past an airfoil.

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 c^*/U_∞^* are employed as velocity, length and time scales respectively. By doing so, the governing equations and the boundary conditions (impermeability and no-slip conditions) become:

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

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

where $\boldsymbol{V}$ and p are the velocity and pressure fields, $Re = (U_\infty^* c^*)/\nu$ is the Reynolds number (with ν being the kinematic viscosity of the fluid).

On the situation depicted in Fig. 1, the adherence boundary condition takes place, ensuring no relative velocity between the airfoil's surface and all fluid particles in contact with it. In terms of its components, the adherence condition is expressed by the impermeability condition, Eq. (3), which states that the normal component of the fluid particles velocity in contact with the boundary S_1 (V_n^p) are equal to the normal component of the airfoil surface velocity (V_n^{S1}). Similarly, Eq. (4) expresses the no-slip condition (equality of the tangential components).

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

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

3. NUMERICAL SOLUTION

The present work deals with a Lagrangian approach, which combines the Discrete Vortex Method and the Panel Method, to numerically solve Eq. (1) and Eq. (2) subjected to the boundary conditions of Eq. (3) and Eq. (4).

First, the Panel Method (Katz and Plotkin, 1991) provides the geometry discretization through a series of flat panels with linearly varying strength vortex distribution, which satisfies the impermeability boundary condition, therefore providing the potential solution.

Then, the Discrete Vortex Method simulates the vorticity field. At each time step of the numerical simulation, Lamb discrete vortices are generated near of the airfoil surface, in such a way that their nucleus tangentially touches the panels' control point (located at its middle) representing the flow boundary layer, as shown in Fig. 2.

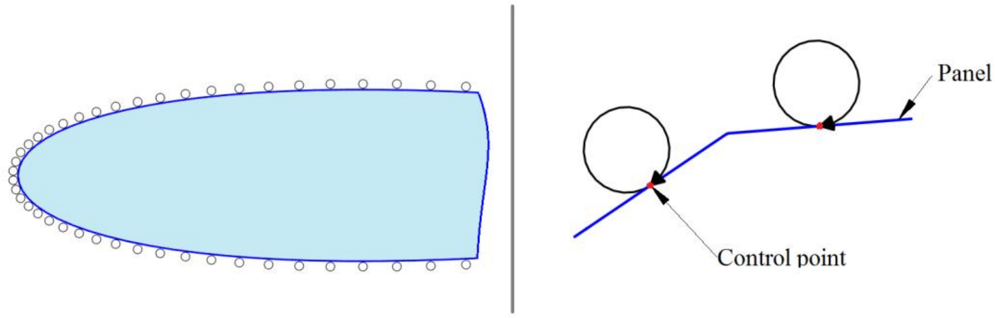

Figure 2 – Discrete vortices generation near the airfoils' surface.

The Lamb discrete vortex model is the solution of the vorticity diffusion equation. Furthermore, as shown in Fig. 3, both of it vorticity distribution and induced tangential velocity, are continuous over the whole fluid domain.

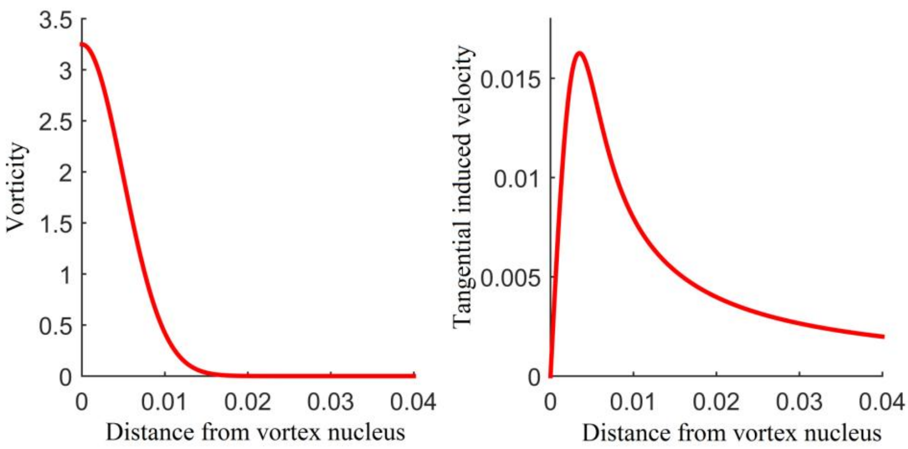

Figure 3 - Vorticity distribution and tangential induced velocity in the Lamb vortex model.

In order to simulate the fluid motion dynamics, the vortices are able to move freely in the fluid domain, governed by the vorticity transport equation. Taking the curl of Eq. (2) and combining the result with Eq. (1), the two-dimensional vorticity transport equation becomes (Batchelor, 1967):

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

Equation (5) shows that the discrete vortices move through the flow field by advection (left hand side) and by diffusion (right hand side).

3.1 The Viscous Splitting Algorithm

In order to solve Eq. 5, Chorin (1973) proposed a split algorithm; in which the diffusion and the advection of the discrete vortices are solved separately, but at the same time-step of the numerical simulation.

$$\frac{\partial \omega}{\partial t} + (\nabla . \boldsymbol{V})\omega = 0 \quad \text{(advection)} \tag{6}$$

$$\frac{\partial \omega}{\partial t} = \frac{1}{Re}\nabla^2 \omega \qquad \text{(diffusion)} \tag{7}$$

During the advection, an Euler scheme, Eq. (8), is applied to the recently generated discrete vortices in the vicinity of the body. For all other discrete vortices present in the domain an Adams-Bashforth scheme is used, Eq. (9).

$$p_{k\,c}^{n}(t+\Delta t)=p_{k\,c}^{n}(t)+u_k^n(t)\Delta t \qquad n=1,2 \tag{8}$$

$$p_{k\,c}^{n}(t+\Delta t)=p_{k\,c}^{n}(t)+[1{,}5u_k^n(t)-0{,}5u_k^n(t-\Delta t)]\Delta t \qquad n=1,2 \tag{9}$$

where $p_{k\,c}^{n}$ and u_k^n represent the position and induced velocity, respectively, of any vortex k at a given time t due to the advection process, and Δt represents the non dimensional time increment of the numerical simulation.

The induced velocity, u_k, Eq. (10); is composed by three contributions: The mainstream velocity, ui, Eq. (11); the solid body (Panel Method) , u_b, Eq. (12); and the induced velocity from all other vortices in the entire fluid domain u_v, Eq. (13).

$$u_k=u_i+u_b+u_v \tag{10}$$

$$ui_1=1 \; and \; ui_2=0 \tag{11}$$

$$ub_k^n(x,t)=\sum_{p=1}^{NP}\gamma_p c_{kp}^n[x_k(t)-x_p] \qquad n=1,2 \; and \; k=1,NV \tag{12}$$

$$uv_k^n(x,t)=\sum_{j=1}^{NV}\Gamma_j c_{kj}^n[x_k(t)-x_j(t)] \qquad n=1,2 \; and \; k=1,NV \tag{13}$$

where NV is the total number of discrete vortices present in the flow domain at the instant t, NP is the total number of panels used in the geometry discretization, γ_p is the linear-strength vortex panel density distribution, $c_{kp}^n[x_k(t)-x_p]$ is the $n-th$ component of the velocity induced at discrete vortex k by the panel p, Γ_j is the intensity of the j vortex and $c_{kj}^n[x_k(t)-x_j(t)]$ is the $n-th$ component of the induced velocity at the discrete vortex k by a discrete vortex j.

The viscous diffusion, Eq. (14), is modeled using the Random Walk Method developed by Chorin (1973).

$$p_{k\,d}(t)=\sqrt{\left[\frac{4\Delta t}{Re}\ln\left(\frac{1}{Z}\right)\right]}\,[\cos(2\pi Q)+\sin(2\pi Q)] \tag{14}$$

where $Z \; and \; Q$ are random numbers between 0 and 1.

3.2 Aerodynamic Loads

For the calculation of aerodynamic loads, the present work uses an extension of the Shintani and Akamatsu (1994) method, which is detailed presented by Ricci (2002). From the vorticity field and assuming the incompressible flow hypothesis, a Poisson equation for the pressure can be derived, leading to (Ricci, 2002):

$$\xi\tilde{y}_i+\int_{S1}\frac{1}{2\pi}\frac{n_x(x-x_i)+n_y(y-y_i)}{(x-x_i)^2+(y-y_i)^2}\tilde{y}dS=-\int_{\Omega}\frac{1}{2\pi}\frac{v(x-x_i)+u(y-y_i)}{(x-x_i)^2+(y-y_i)^2}\omega d\Omega -\frac{1}{Re}\int_{S1}\frac{1}{2\pi}\frac{n_y(x-x_i)+n_x(y-y_i)}{(x-x_i)^2+(y-y_i)^2}\omega dS \tag{15}$$

where ξ is a constant that assumes the value of 0.5 at the solid boundaries and 1 in the fluid domain, n_x and n_y are the vertical and horizontal components of a unit vector $\boldsymbol{n}$, x and y are the coordinates of any point belonging to the fluid domain that induces a specific work $\tilde{y}$ (defined as $\tilde{y}=p-p_\infty+\frac{1}{2}[u^2-1]$) in the point $i(x_i,y_i)$, and dS is a infinitesimal element of the solid boundary.

In Eq. (15), the first integral represents the body's surface contribution in the pressure calculation. The second one takes into account the vorticity effect present in fluid domain Ω and the third counts the influence of vorticity located on the body's surface. Solving the Eq. (15) for $\tilde{y}$, the pressure coefficient C_p, can be found (Ricci, 2002):

$$C_p = 2\tilde{y} + 1 \quad (16)$$

4. SIMULATION OF THE FLOW AROUND SLENDER BODIES

The case studied was the NACA 0012 airfoil with zero angle of attack. First the potential solution has been analyzed and the result is presented in Fig. 4.

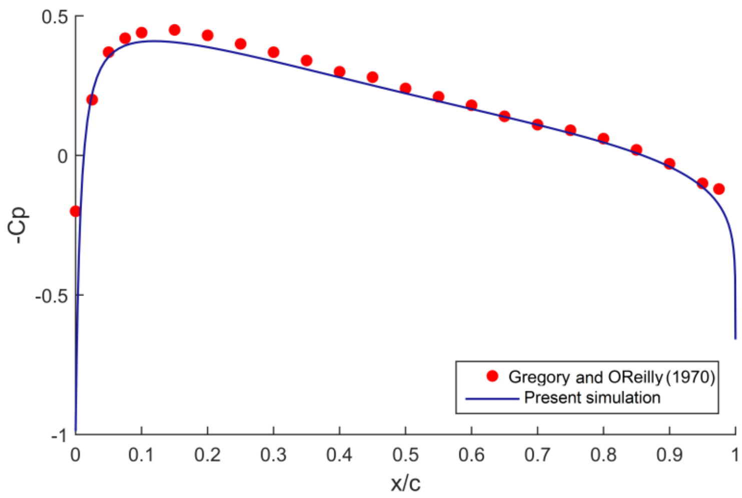

Figure 4 – Potential solution for the NACA 0012 airfoil $(\theta = 0°)$.

Observing the Fig. 4, it can be seen that the potential solution provides fair values when compared to the experimental work, for the major length of the airfoil chord. The suction peak predicted by the Panel Method is found to be overly low in comparison with the experimental results, therefore being one of the main discrepancies observed. The minimum value provided by the Panel Method for the pressure coefficient is approximately 14% lower than the results obtained by Gregory and O'Reilly (1970). Also, the pressure coefficient, which is expected to be zero for this case, has been found to be around 0.6, an unrealistic value.

The results bought by the Panel Method shows that a more precise model is needed to overcome the limitations of this over-simplified technique, thus providing a more precise solution.

In this work, such improvement is performed using the Discrete Vortex Method. Table1 shows the numerical parameters employed in this method. Each one of these constants has its own choosing criteria in order to improve the simulation results.

The number of panels, time-step increment and the total number of time-steps are determined through extensively tryouts during the early development of the numerical code. For the other parameters, the values have been chosen according to the work brought by Silva (2002) which presents good results.

Table1: Numerical parameters used.

Parameter	**Symbol**	**Value**
Number of panels	m	300
Time-step increment	Δt	0,025
Total number of time-step increments	Kt	400
Lamb's discrete vortex radius	eps	0,0050
Reynolds number	Re	170000

Figure 5 shows the viscous wake formed at the end of the numerical simulation.

Figure 5 – Viscous wake: NACA 0012 ($\theta = 0°;\ Re = 1{,}7\text{x}10^5$)

The fluid dynamic loads were calculated using Shintani and Akamatsu (1994) method. Figure 6 shows the pressure coefficient (C_p)distribution over the airfoil chord. One can see that the Discrete Vortex Method adequately represents the physical situation studied. Although a slight difference between the fluid dynamic loads over the airfoil surfaces are present, the overall result is consistent with the experimental data. In addition, the behavior of the suction surface pressure around the 10% chord region is better represented with the Discrete Vortex Method when compared to the potential solution.

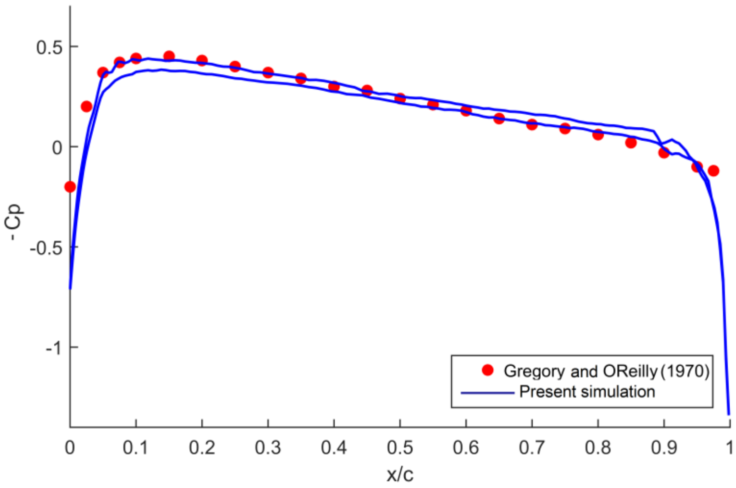

Figure 6 – Pressure distribution over NACA 0012 airfoil $\theta = 0°, Re = 170000.$

Figure 7 shows the lift coefficient (C_l) plotted during the non-dimensional time between 5 and 10, has decent results. Since the lift coefficient for a NACA 0012 parallel with the flow ($\theta = 0°$) must be zero, the obtained result can be interpreted as the error associated with the Discrete Vortex Method implementation. After averaging C_l over the specified time period, it turns out that this error is about 4.28%, a suitable result.

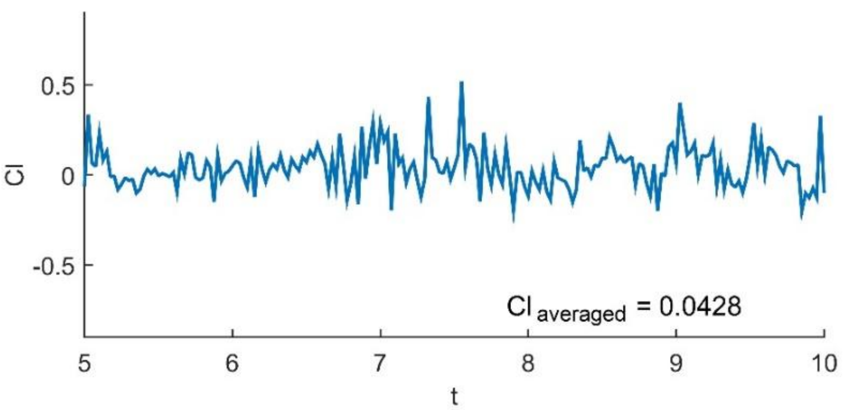

Figure 7 – Lift coefficient for NACA 0012 airfoil ($\theta = 0°;\ Re = 1{,}7\text{x}10^5$)

5. CONCLUSIONS

The authors applied the Discrete Vortex Method to simulate the viscous flow around the NACA 0012 airfoil with zero angle of attack. The method was able to represent the physical situation analyzed. The distribution of the pressure coefficient along the aerofoil chord showed good agreement with the experimental results. The leading edge and trailing edge regions, naturally dominated by perturbations and high velocity gradients, showed slight divergence from the experimental results but with little influence on the overall simulation result.

The Discrete Vortex Method presents several challenges and opportunities for future work. A first continuity of this work would contemplate the analysis of the other airfoils with several angles of attack.

6. REFERENCES

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

Chorin, A. J., 1973. "Numerical Study of Slightly Viscous Flows". In *Journal of Fluid Mechanics*. Vol. 57, p.785-796.

Cottet, G., Hildebrand, R., Koumoutsakos P., Mimeau, C., Mortazavi, I. and Poncet, P., 2014. "Passive and active flow control using vortex methods". In *Proceedings of the 6th International Conference on Vortex Flows and Vortex Models-ICVFM*. Nagoya, Japan.

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

Hess, J. L. and Smith, A. M. O., 1967. "Calculation of potential flow about arbitrary bodies". In *Progress in Aeronautical Sciences*, Vol. 08, p 138.

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

Katz, J. and Plotkin, A., 1991. *Low-Speed Aerodynamics*, McGraw-Hill, Singapore, 1st edition.

Şahin, I. and Acir, A., 2015. "Numerical and Experimental Investigation of Lift and Drag Performances of NACA0015 Wind Turbnine Aerofoil". In *International Journal of Materials, Mechanics and Manufacturing*, Vol.03, p 22-25.

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

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

Su, Y. and 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*, 2015.

Silva, F. D. C. E., 2002. *Simulação Numérica do Escoamento ao redor de Aerofólios Via Método de Vórtices Associado ao Método dos Painéis*. Master thesis, COPPE/UFRJ, Rio de Janeiro.

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.

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, United Arab Emirates.

Wang, Q., Wang, J., Chen, J., Luo, S. and Sun, J., 2015. "Aerodynamic shape optimized design for wind turbine blade using new airfoil series". In *Journal of Mechanical Science and Technology*, p. 2871-2882.

7. RESPONSIBILITY NOTICE

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