# Wing Profile Evolution driven by Computational Fluid Dynamics

# Evolución de Perfil de Ala Guiada por Dinámica de Fluidos Computacional

Jose L. Hernandez[1], Cristian C. Rendon[1],
Oscar Ruiz - Salguero[1], Carlos A. Alvarez[2] and Mauricio Toro[2]

[1] Laboratorio de CAD CAM CAE, Universidad EAFIT, Carrera 49 No. 7 Sur-50, 050022 Medellin, Colombia
[2] Universidad EAFIT, Carrera 49 No. 7 Sur-50, 050022 Medellin, Colombia

E-mails: jlhernande@eafit.edu.co, crendo11@eafit.edu.co,
oruiz@eafit.edu.co, calvar52@eafit.edu.co, mtorobe@eafit.edu.co



**Abstract**

In the domain of fluid dynamics the problem of shape optimization is relevant because is essential to increase lift and reduce drag forces on a body immersed in a fluid. The state of the art consists of two variants: (1) evolution from an initial guess, using optimization to achieve a very specific effect, (2) creation and genetic breeding of random individuals. These approaches achieve optimal shapes and evidence of response under parameter variation. Their disadvantages are the need of an approximated solution and / or the trial - and - error generation of individuals. In response to this situation, this manuscript presents a method which uses Fluid Mechanics indicators (e.g. streamline curvature, pressure difference, zero velocity neighborhoods) to directly drive the evolution of the individual (in this case a wing profile). This pragmatic strategy mimics what an artisan (knowledgeable in a specific technical domain) effects to improve the shape. Our approach is not general and it is not fully automated. However, it shows to efficiently reach wing profiles with the desired performance. This approach shows the advantage of application domain - specific rules to drive the optimization, in contrast with generic administration of the evolution.

**Resumen**

En el dominio de mecánica de fluidos, el problema de optimización de forma es relevante porque es esencial incrementar la fuerza de elevación y reducir la de arrastre en un cuerpo inmerso en un fluido. El estado del arte tiene dos variantes: (1) evolución a partir de una estimación inicial usando optimización para lograr un efecto muy específico, (2) creación y crianza genética de individuos aleatorios. Estos enfoques logran formas óptimas y evidencian la respuesta bajo la variación de parámetros. Sus desventajas son la necesidad de una solución aproximada y / o la generación de individuos por ensayo - y - error. En respuesta a esta situación, este manuscrito presenta un método que usa indicadores de Mecánica de Fluidos (e.g. curvatura en lineas de corriente, diferencia de presión, zonas de velocidad cero) para dirigir la evolución de un individuo (en este caso un perfil de ala). Presentamos una estrategia pragmática que imita un las acciones de un artesano (conocedor de un dominio técnico específico) para mejorar la forma. Nuestra aproximación no es general y no está completamente automatizada. Sin embargo, presenta eficiencia al alcanzar perfiles de alas con el desempeño deseado. Este aproximación presenta la ventaja de usar reglas específicas al dominio de aplicación para realizar la optimización, en contraste con una administración genérica de la evolución.

**Keywords.** Shape evolution, wing profile, Fluid Mechanics.

**Palabras clave.** Evolución de forma, perfil alar, Mecánica de Fluidos.



# Glossary

| | |
|---|---|
| $\Omega$ | Rectangular orthogonal simulation domain $\in R^2$ with center in (0,0). $x \in [-w, w]$ and $y \in [-h, h]$. |
| $\Gamma$ | Wing profile represented as a simple closed curve $\in R^2$ immersed in $\Omega$. |
| $V_\infty$ | Flow velocity at $x = -w$. |
| $V$ | Velocity magnitude at a point $\in \Omega$. |
| $P_{ref}$ | Magnitude of reference pressure. |
| $P$ | Pressure magnitude at a point $\in \Omega$. |
| $F_L$ | Lift force acting on $\Gamma$. |
| $F_D$ | Drag force acting on $\Gamma$. |
| $C$ | Streamlines curvature. |
| $N_e$ | Number of mesh elements. |

# 1 Introduction

In nature, constant perturbations of a fluid in objects make to change their shape in order to develop their dynamic behavior and evolve. Examples are raindrops formation, eolic erosion or abrasion of rocks by streams. Similarly, engineering applies shape evolution techniques to develop devices or tools with optimal performance. Aeronautics focuses in the optimization of aerodynamic performance in aircraft with CFD.

Due to current computational power and mathematical models, this optimization can be partially conducted in silico, saving in costly wind tunnel and other experiments. The present work presents a methodology of experimentation with computational fluid dynamics (CFD) observing flow characteristics of an individual to evolve its shape achieving a required lift- and minimize drag-force.

# 2 Literature Review

The optimization process of a wing profile can be carried out in two ways, (1) evolution from an initial guess, using optimization, (2) creation and genetic breeding of random individuals.

1. Optimization methods use an objective function to be satisfied (e.g. gradient-based method [1, 2]). Optimization methods are successful under one or two criteria to achieve a specific effect (e.g. lift production and / or drag reduction). The disadvantage is the need of an initial guess.

2. The creation and genetic breeding of random individuals modifies its flow conditions and / or the geometry, searching to improve the aerodynamic performance of the individual. Refs. [3, 4] change the flow direction on the individuals. Refs. [5, 6, 7] modify surface geometry of the individuals. These experimentations can be conducted in wind tunnels and / or CFD. The disadvantage of these methods is the trial - and - error way to achieve the desired performance.

## 2.1 Conclusions of literature review

Optimization methods need of an initial guess to be carried out. Creation of random individuals present a trial - and - error methodology. This work intends to evolve, gradually, an initial rectangular profile into a wing profile using Fluid Mechanics indicators. Our approach is a pragmatic strategy to drive the optimization. However, it is not general and it is not fully automated. Table 1 presents an overview of the literature review.

Table 1: Different approaches and our contribution

| **Approach** | **Refs.** | **Advantages** | **Disadvantages** |
|---|---|---|---|
| Evolution from an initial guess, using optimization methods | [1, 2, 6, 8, 9] | (1) Successful to achieve a specific effect. | (1) Initial guess needed. |
| Creation and genetic breeding of random individuals. | [3, 4, 5, 7] | (1) Evidence of response under parameter variation. | (1) Trial and error methodology. |
| Our approach: To drive the evolution of a random individual using Fluid Mechanics indicators | | (1) The method presents an evolution sequence.<br>(2) It is a pragmatic methodology favoring the understanding of the phenomenon. | (1) It is not fully automated. |

# 3 Methodology

## 3.1 Computer experimental setup

The experiment is carried out in the software *ANSYS Academic Research Fluent, Release 17.2*. The initial model consists in a 2D profile ($\Gamma$) immersed in a fluid ($\Omega$) moving at a certain velocity ($V_\infty$) such that $\vec{V}(x = -w) = V_\infty \hat{\mathbf{i}} + 0\hat{\mathbf{j}}$ as seen in Figure 1. $\Omega$ is bounded for parameters *w* and *h*. $\Gamma$ is defined at the first stage by the parameter *a* and *b* in Table 2.

**Assumptions**

1. $\Omega$ is a Newtonian fluid regio $\in R^2$ with constant density and viscosity. This is because the Mach number for $V_\infty$ is less than 0.3 being an incompressible flow [10].

2. $\Gamma$ rigid with no slip condition. Therefore, Velocity ($V$) in body boundary is 0.

3. Steady state flow (i.e. the derivative of the fluid properties with respect to time is equal to zero).

4. Shear stress transport model (SST) for CFD solution. SST model is highly accurate in the predictions of flow separation. Captures eddies phenomena and reaches convergence.

   Meshing could not be the appropriated for this model but it is functional to capture the phenomenon. The same methodology of meshing is used in all stages of the evolution process. Figure 2 presents the sizing- and inflation- methods in *ANSYS Academic Research Mesh, Release 17.2*.

Table 2: Experimental setup / Initial conditions

| $\Omega$ | $\Gamma$ | $V_\infty$ | $P_{ref}$ | $w$ | $h$ | $a$ | $b$ |
|---|---|---|---|---|---|---|---|
| air at $25°C$ | body boundary | 80 $m/s$ | 1 atm | 35 $m$ | 30 $m$ | 1.5 $m$ | 3 $m$ |

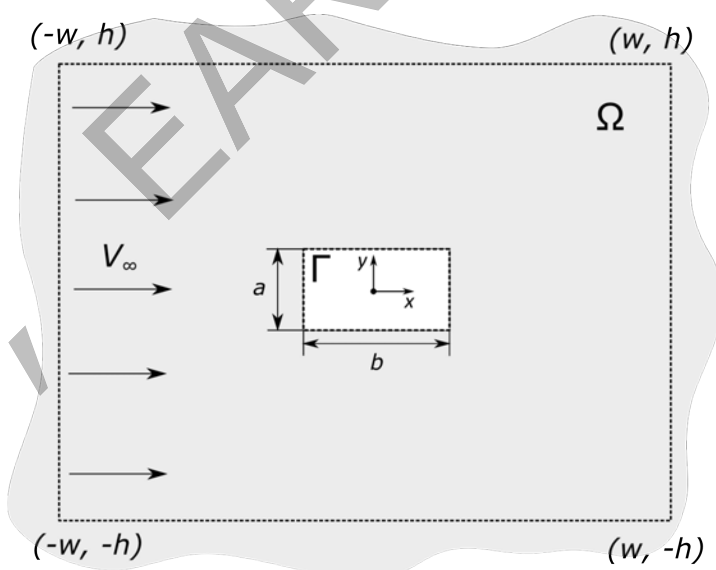
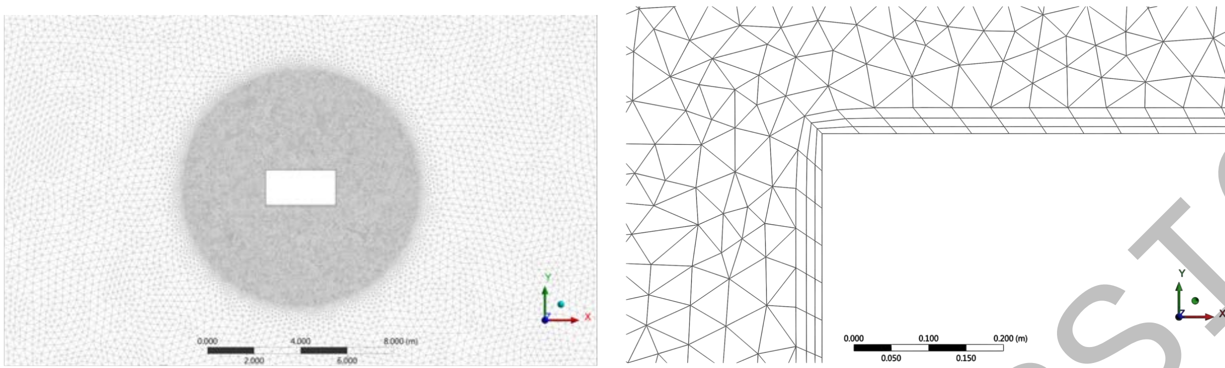



Figure 1: Diagram of the model at initial stage.



(a) Detail of sizing

(b) Detail of inflation

Figure 2: Mesh for initial stage.

## 3.2 Shape evolution process

Shape evolution process is carried out in a pragmatic and intentional way, evolving the shape from a rectangular profile into a wing profile adding or removing material. Figure 3 illustrates the evolution process.

1. **Goal:** To satisfy a lift force s.t. $F_L \geq 10000\ N$ and to reduce drag force $F_D$ with respect to $F_{D_0}$. Eq. 1 and Eq. 2 show how the forces are computed whit their discrete form [10].

2. **Criteria:** Reduction of pressure on the upper surface by increasing there the stream velocity in order to produce pressure difference (i.e. lift force). Reduction of drag by producing laminar flow (avoid streamlines divergence from $\Gamma$). Avoid zero velocity neighborhoods.

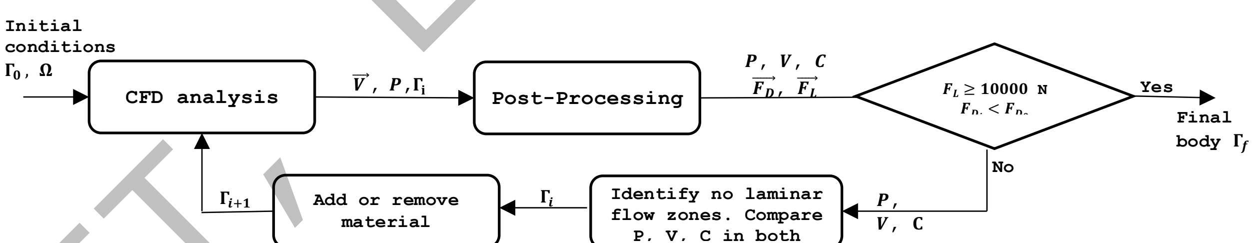



Figure 3: Evolution process diagram

$$F_L = \oint P dx \approx \sum P_n(\Delta x_n) \quad (1)$$

$$F_D = \oint P dy \approx \sum P_n(\Delta y_n) \quad (2)$$

## 3.3 Fluid Mechanics indicators

The Fluid Mechanical indicators to conduct the shape evolution are three. Velocity scalar map, pressure scalar map and streamlines curvature. These indicators are analyzed in each stage of the evolution. Velocity- and pressure- scalar map are taken directly from the *ANSYS* postprocessor as a result of the solution of the navier-stokes equations.

Curvature of the streamlines are obtained as follow. A function interrogates *ANSYS* database. Then, curvature is calculated from Eq. 3 as a discrete curve how it is indicated in [11].

$$C_i = \frac{|t_i - t_{i+1}|}{|[(v_i + v_{i+1})/2] - [(v_{i-1} + v_i)/2]|} \tag{3}$$

Where $v_i$ is the i-th vertex of the streamline, $t_i$ is the vector going from $v_i$ to $v_{i+1}$ and $C_i$ is the curvature at $v_i$. The calculation of $C_i$ in all the streamlines allows to draw the curvature scalar map.

# 4 Results

Figs. **Error! Reference source not found.** and **Error! Reference source not found.** show force- and shape- evolution respectively, as follows.

- Stage 1. Figure 4 (a), (b), (c) presents symmetry between the upper and lower surfaces, resulting in null lift. High pressure in front produces a drag significantly grater than lift. Streamlines diverging from the profile boundary suggest non-laminar flow (to be avoided). There are high curvature values in front and corners of $\Gamma_0$.

- Stage 2. Figure 4 (d), (e), (f). To reduce high pressure in front of $\Gamma$ and the high curvature, the corners are rounded. The stage presents a significantly reduction of drag and emergence of lift. Streamlines are tighter to the profile. Asymmetry appears.

- Stage 3. Figure 4 (g), (h), (i). Lift presents high increase with respect previous stages (see Figure 5(b)). The back is rounded reducing the zero velocity neighborhoods.

- Stage 4. Figure 4 (j), (k), (l). The lift reaches $13000\,N > 10000\,N$ (see Figure 5 (b)). The zero velocity zones are filled by the object. The streamlines fit completely to the profile. Velocity at lower surface is largely equal to the flow velocity $V_\infty$.

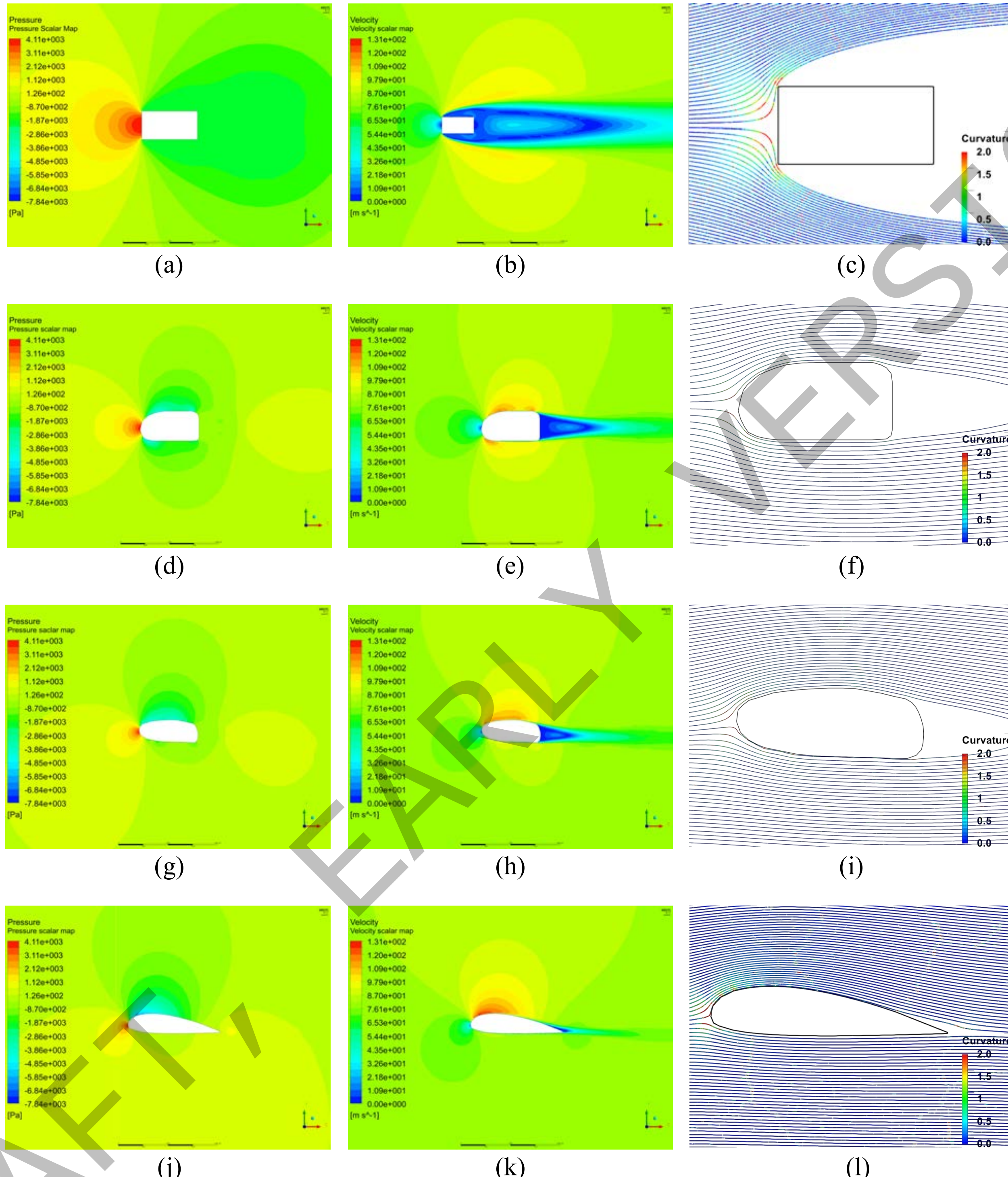
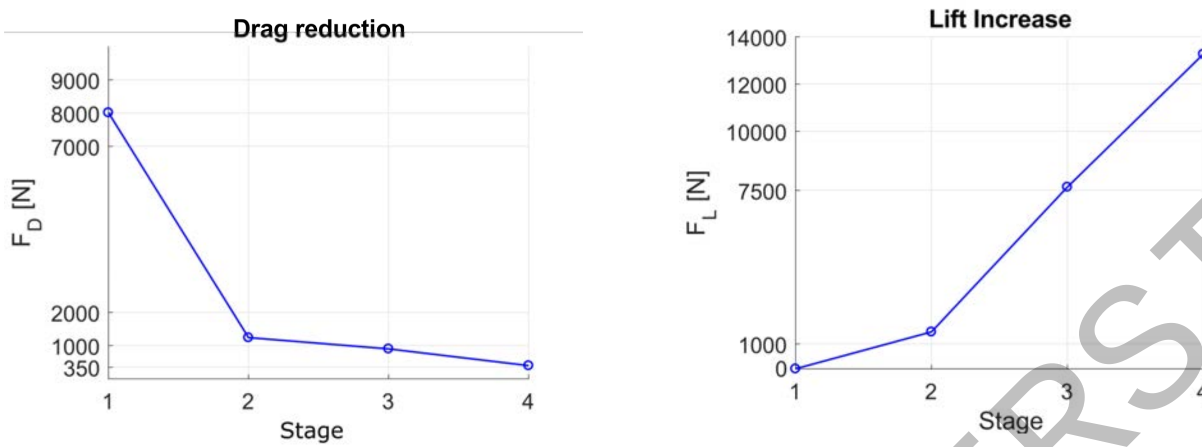



Figure 4: Evolution scalar maps of velocity, pressure and streamlines.




(a) Drag decrease

(b) Lift increase

Figure 5: Lift and Drag evolution

## 4.1 Algorithms complexity

Three algorithms are implemented for the stages analysis. To calculate the complexity of these algorithms the measure variable is the number of elements in the mesh $N_e$. Being the number of elements in a horizontal line in $[-w, w]$ or vertical line in $[-h, h]$ is $O(\sqrt{N_e})$. Table 3 shows a briefly description of the algorithms.

Table 3: Algorithms description and complexity

| Algorithm | Description | Complexity |
|---|---|---|
| ANSYS Database interrogation | This functions interrogates ANSYS data base to import velocity, pressure and streamlines information. | $O(N_e)$ |
| Lift and Drag calculation | Function that applies Eqs. 1 and 2 to find the forces acting on the wing profile. | $O(N_e)$ |
| Curvature calculation | Function that applies Eq. 3 to a streamline in order to calculate curvature on its vertex. | $O(\sqrt{N_e})$ |

# 5 Conclusions and future work

Figure 4 (b) and Figure 5 (a) show that perpendicular surfaces to the flow increase drag by high pressure zone in front. Streamlines show the response of the corner rounding favoring both reduction of drag (Fig. Figure 5 (a) shows higher reduction of drag ) and laminar flow (see Figure 4 (f)). Streamlines along the evolution validate the reduction of drag by making the flow more closed to laminar [8]. It occurs when there is not separation between streamlines and the profile. Production of lift seems favored by an asymmetric shape respect flow direction where the inclination is an determinant aspect.

Zero velocity combined with low pressure zones suggest presence of eddies and this zones can be filled by the object improving the aerodynamic behavior. In this sense, mathematical models based into reducing zero velocity and low pressure zones can be developed taking into account that there is no transfer of momentum at their boundary. Both, the experimental method presented and a hypothetical mathematical model could be automated in a future work. This methodology can be applied for the development of devices and the understanding of fluid dynamics with submerged bodies.

# References


[1] A. Fathi and A. Shadaram, "Multi-level multi-objective multi-point optimization system for axial flow compressor 2d blade design," *Arabian Journal for Science and Engineering*, vol. 38, no. 2, pp. 351–364, 2013.

[2] P. Nørtoft and J. Gravesen, "Isogeometric shape optimization in fluid mechanics," *Structural and Multidisciplinary Optimization*, vol. 48, no. 5, pp. 909–925, 2013.

[3] S. Askari and M. Shojaeefard, "Shape optimization of the airfoil comprising a cross flow fan," *Aircraft Engineering and Aerospace Technology*, vol. 81, no. 5, pp. 407–415, 2009.

[4] L. Patruno, M. Ricci, S. de Miranda, and F. Ubertini, "Numerical simulation of a 5: 1 rectangular cylinder at non-null angles of attack," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 151, pp. 146–157, 2016.

[5] C. Park and S. Lee, "Effects of free-end corner shape on flow structure around a finite cylinder," *Journal of Fluids and Structures*, vol. 19, no. 2, pp. 141–158, 2004.

[6] S. Jain, V. D. Bhatt, and S. Mittal, "Shape optimization of corrugated airfoils," *Computational Mechanics*, vol. 56, no. 6, pp. 917–930, 2015.

[7] S.-H. Seo and C.-H. Hong, "Performance improvement of airfoils for wind blade with the groove," *International Journal of Green Energy*, vol. 13, no. 1, pp. 34–39, 2016.

[8] D. SS, V. PMHW *et al.*, "Design of fuselage shapes for natural laminar flow," NASA Langley Technical Report Server, Tech. Rep., 1986.

[9] Y. Ogawa and M. Kawahara, "Shape optimization of body located in incompressible viscous flow based on optimal control theory," *International Journal of Computational Fluid Dynamics*, vol. 17, no. 4, pp. 243–251, 2003.

[10] Y. A. Çengel, J. M. Cimbala, V. C. Olguín, and S. F. Skarina, *Mecánica de fluidos: fundamentos y aplicaciones*. plus 0.5em minus 0.4emMcGraw-Hill São Paulo, 2006, vol. 1.

[11] P. Manfredo, *do Carmo. Differential geometry of curves and surfaces*, 1st ed. plus 0.5em minus 0.4emPrentice Hall, 1976, chapter 1.

[12] B. Mohammadi and O. Pironneau, “Shape optimization in fluid mechanics,” *Annual Reviews of Fluid Mechanics*, vol. 36, pp. 255–279, 2004.