Available online www.ejaet.com

European Journal of Advances in Engineering and Technology, 2016, 3(8): 93-100

Research Article ISSN: 2394 - 658X

On The Characteristics of NACA 0006: A Hybrid Non- Mesh Solution

A Jafarimoghaddam[1] and S Aberoumand[2]

[1]*Department of Aerospace Engineering, K N Toosi University of Technology, Tehran, Iran*
[2]*Department of Mechanical Engineering, Islamic Azad University, Takestan, Iran*
a.jafarimoghaddam@gmail.com

ABSTRACT

In this study we aimed to perform an exact simulation based on the combination of Vortex Panel Method and Falkner Skan Method for NACA 0006 in low angle of attacks. As it is obvious, lift coefficient is governed by the pressure distribution around the airfoil. Besides, skin friction coefficient (related to skin drag coefficient) is highly depended on the distribution of velocity in viscos region. Therefore, in the present work, a hybrid solver is derived in which couples the Vortex Panel Method with Falkner Skan Method for simultaneously solving the boundary layer and invisid region. Dealing with Falkner Skan and Panel method, we have already assumed that the flow is incompressible and also the self- similarity assumption exists for the entire boundary layer, which means that flow separation is simply ignored. This last assumption is mostly valid for low Reynolds numbers and/ or low angle of attacks and/ or thin airfoils as NACA 0006.

Key words: Falkner Skan coupled with Vortex Panel Method, Subsonic flows, NACA0006, Skin friction and lift Coefficient, Non- mesh solution

INTRODUCTION

Finding exact solutions for external flow problems has been controversial so far. Many years of research has not been yet concluded to the exact solution of Naveir- Stokes equations but only for a few specific problems. Numerical approaches were then born as an alternative to overcome this issue. But as it is crystal clear, the basis of these solvers is doing an iterative process which is considerably a time consuming process. Similarity analysis allowed us to convert the governing PDEs (Partial Differential Equations) of momentum equations into ODE (Ordinary Differential Equations) ones. This methodology led us to some more exact solutions for some more specific problems. Transformation of PDEs to ODEs is usually considered as non- mesh methods. Therefore, it is sensible to attempt finding similarities in fluid flow problems. External flows are in the attention because they are applicable in many industries especially for airfoil design [1- 6]. Assessing the performance of an airfoil, it is essential to have exact information of especially lift and drag coefficient. The exerted forces on an airfoil are caused by shear stresses and pressure. Order of magnetite analysis reveals that pressure gradients are negligible in the boundary layer of a flat plate and it is only depended on the pressure gradients over the boundary layer of any external flow. So, pressure gradients can be simply calculated using Euler momentum equation over the boundary layer. Therefore, one must seek for the velocity gradients over the boundary layer of a streamline body. The velocity gradients (over the boundary layer where the potential flow dominates) are quantified using the potential flow algorithms such as Panel Methods [4]. For calculating the shear forces, boundary layer must be first solved. Blasius developed a similarity solution for solving the boundary layer over a flat plate. After that, this solution was applied for wedges. This methodology is known as Falkner Skan transformation [6]. Doing this method, the velocity profiles in the boundary layer of a wedge would be obtained. Hence, shear stresses would be easily calculated for this case. By having this over review, it is possible to couple the Panel method with Falkner Skan method. By doing this, both pressure forces and shear forces exerted on a streamline body can be achieved by dividing the external body into finite numbers of attached wedges. In the present work, we have developed a solver for simultaneously solving the Panel Method and Falkner Skan transformation equation. The streamline body is assumed to be NACA 0006. The study has been pursued in zero, 1^o, 2^o, 3^o, 4^o and 5^o angle of attacks. In the next sections, we deal with the implementation of this solver after a brief discussion on both Panel Method and Falkner Skan method is presented. In further research, we will introduce a simple and user- friendly software for calculations related to the bluff and streamline bodies and also an optimized airfoil (the highest C_l/ C_d) will be introduced by employing Genetic

Algorithm. Considering that drag coefficient in a laminar and incompressible flow regime is affected by several factors including shape drag, skin drag (caused by shear forces) and induced drag, in the mentioned future work, the effect of induced drag will be also included. Induced drag exists because airfoils are actually three- dimensional. Therefore, this factor includes the three- dimensional effect of airfoil. Fortunately, this factor is also can be obtained by non- mesh methods. But as it is obvious, this effect is ignored in the present work and we have only dealt with the skin friction coefficient. Further explanations on this factor will be presented in our future study.

A Brief Discussion on Panel Method

Panel methods are technique for solving incompressible potential flows over 2D or 3D thick bluff bodies. There are several of these Panel Method techniques. These methods are applied especially for calculating the pressure distribution over an airfoil. Among them, Vortex Panel Method has the advantage of application for airfoils in different angle of attacks. Other Panel Methods whose do not consider the vortex potential flow in their simulation procedure, cannot be applied for asymmetric airfoils or airfoils in different angles of attacks. In the present work, Vortex Panel Method is introduced briefly. In 2-D, the airfoil surface is divided into piecewise straight line segments or panels or "boundary elements" and vortex sheets of strength Γ are placed on each panel. The philosophy of the existence of lift force on an airfoil comes from this fact that the upper surface boundary layer contains, in general, clockwise rotating vorticity and the lower surface boundary layer contains, in general, counter clockwise vorticity. Because there is more clockwise vorticity than counter clockwise vorticity (simply, for existence of lift, the length of the upper surface should be greater than that of lower surface), there is net clockwise circulation around the airfoil. In panel methods, we replace this boundary layer, which has a small but finite thickness with a thin sheet of vorticity placed just outside the airfoil. This net clockwise circulation around the airfoil can be understood as the existence of lift force on the airfoil. In this model, the vorticity around the airfoil is modeled by assuming vortexes in each panel around the airfoil. At first, the strength of these vortex flows is not identified. So by considering a specific value for the airfoil as the stream line, and using the superposition method for calculating the effect of other panels on a certain panel, the main equation of Vortex Panel Method is obtained. Finally, because we have assumed a certain value for the airfoil as a stream line, this value must be also identified. So, for n numbers of panel, we have n equations and n+ 1 unknown. Here, Kutta condition is applied to balance the number of equations with the unknowns. Kutta condition states that the pressure above and below the airfoil trailing edge must be equal, and then the flow must smoothly leave the trailing edge in the same direction at the upper and lower edge. Because the methodology of Vortex Panel Method is simply found in the literature of classical methods for solving the potential flow [4], we have skipped from a detailed discussion on this subject in the present work. Main governing equations of the Vortex Panel Method can be written as:

$$u_{\infty} y - v_{\infty} x - \frac{1}{2\pi} \oint \gamma_0 \ln\left(\left|\bar{r} - \bar{r}_0\right|\right) ds_o = C$$

$$\gamma_{Upper} = -\gamma_{lower}$$

In which Eq. 1 is for the cumulative effect of other panel's vortexes on a specific panel and Eq. 2 stands as the Kutta condition. A self- developed Panel Method Code was employed for solving the potential flow over the airfoil. In this work, this code is applied for NACA0006 after that the code has been validated for the flow around the circle (the flow around the circle can be solved analytically by the assumptions of potential flows and so it is widely found in the literature of Fluid Mechanics). Fig. 1 shows the mentioned validation. Fig. 2 indicates the geometry of NACA0006 and in Fig. 3 as the result of Vortex Panel Method, the pressure coefficient around this airfoil in six different angles of attack (zero, 1^o, 2^o, 3^o, 4^o and 5^o) is shown. In Fig. 4, the acquired lift coefficients are compared to that of thin airfoil theory.

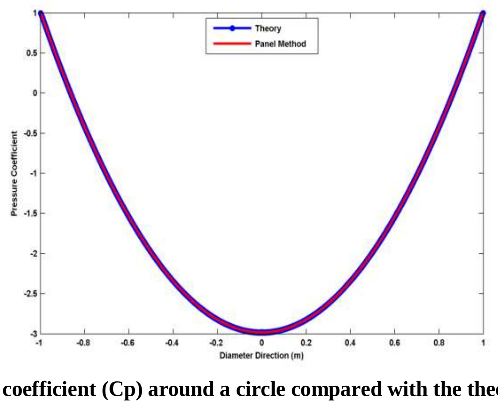


Fig. 1 Pressure coefficient (Cp) around a circle compared with the theoretical solution

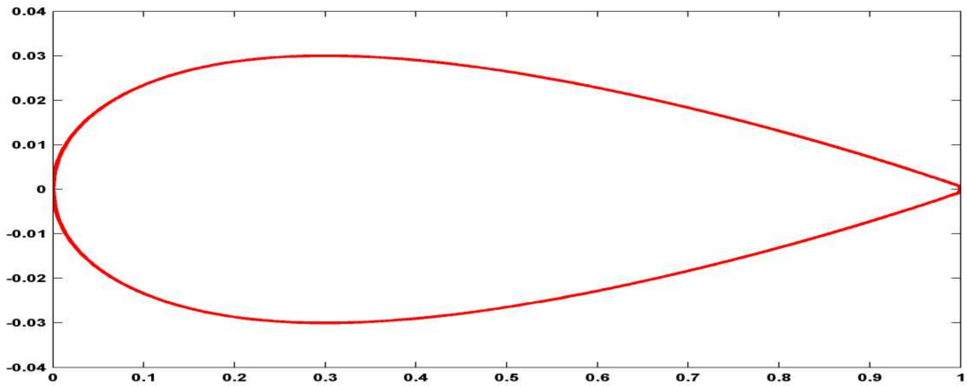


Fig. 2 NACA0006

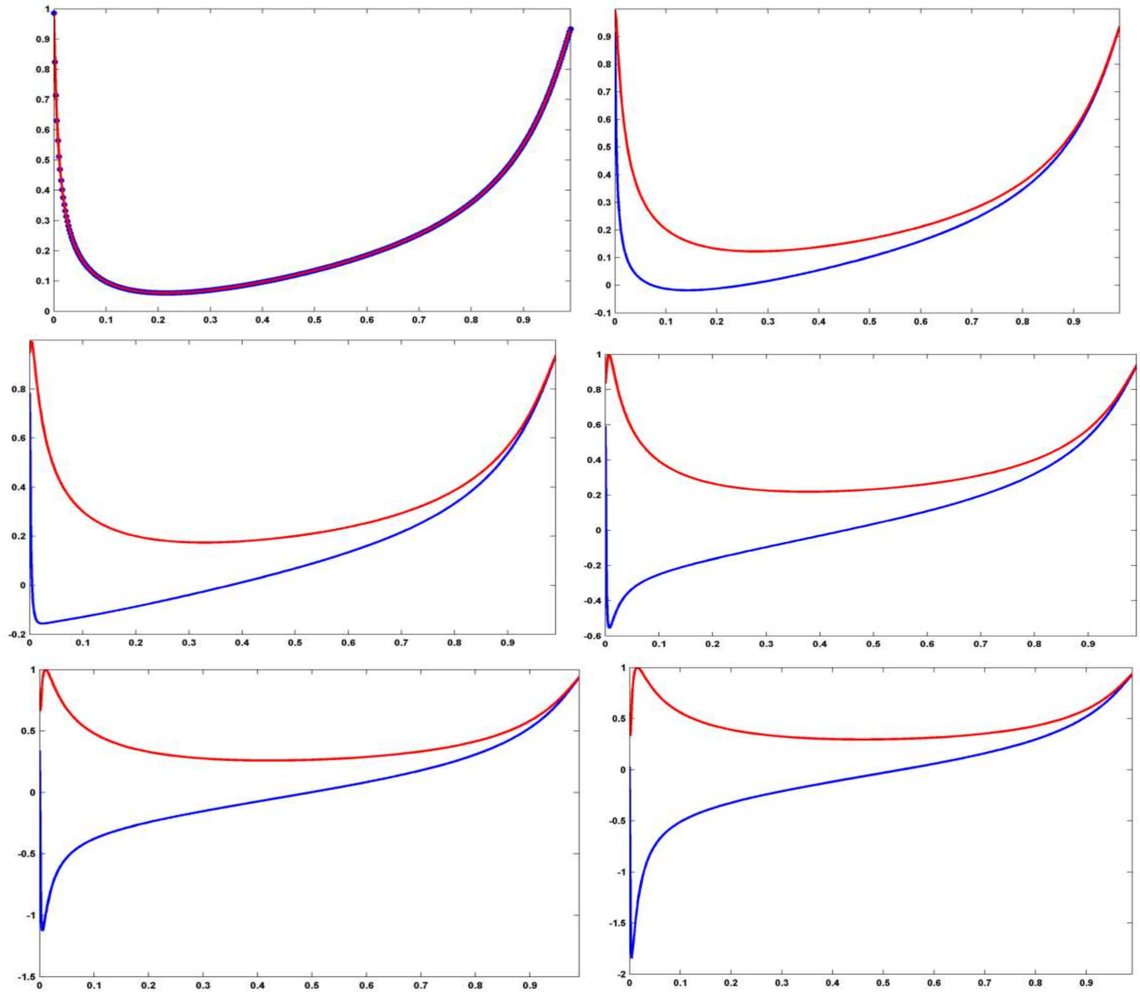


Fig. 3 Pressure distribution over NACA0006; First row: zero and 1º angle of attack; Second row: 2º and 3º angle of attack; Third row: 4º and 5º angle of attack: Red line is for the lower surface and Blue line is for the upper surface

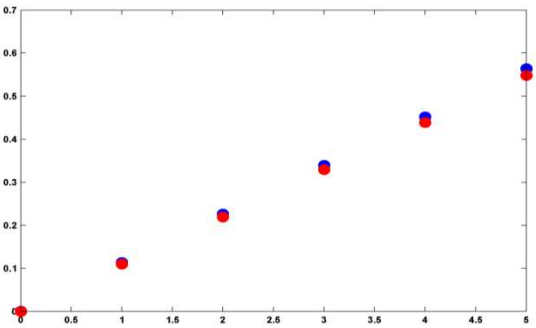


Fig. 4 Lift Coefficient (vertical axis) vs. angle of attack (horizontal axis); Red is for analytical solution of thin airfoils and Blue is the solution of Panel Method

A brief introduction on Falkner Skan Method, Solver Methodology and Results

Blasius developed a similarity solution for boundary layer over a semi- infinite plate. After that Falkner and Skan used this methodology for solving the boundary layer over a wedge. They developed a new transformation for x-momentum equation. In this transformation equation, the effect of pressure gradient was also included (by using order of magnitude technique it can be shown that the pressure gradients happen just for curved and/ or wedged plates). Since as well as Panel Method, this method can be also widely found in the literature of Fluid Mechanics [6]. This procedure is only briefly explained in this paper:

Navier Stokes equations for an incompressible steady state flow are as follow:

$$\begin{aligned} &\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \\ &u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial x}=-\frac{1}{\rho}\frac{\partial p}{\partial x}+\upsilon(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}) \\ &u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial x}=-\frac{1}{\rho}\frac{\partial p}{\partial y}+\upsilon(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}) \end{aligned} \tag{3}$$

Using the order of magnitude technique, the boundary layer thickness over a flat plate can be obtained as:

$$\frac{\delta}{x} \sim \frac{1}{\sqrt{\mathrm{Re}_x}} \tag{4}$$

In Eq. (4), the boundary layer thickness is a function of x direction.

Moreover, order of magnetite reveals that y- momentum equation can be simply ignored. Furthermore, second derivative of x- component velocity in x- momentum equation would be eliminated. So, for a flat plate we only deal with transforming the following PDE equations:

$$\begin{aligned} &\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \\ &u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial x}=\upsilon\frac{\partial^2 u}{\partial y^2} \end{aligned} \tag{5}$$

The only difference between Blasius transformation with Falkner Skan technique is to substitute the x- derivative of pressure with the derivate of velocity over the boundary layer where the potential flow dominates (note that for potential flows, this velocity derivative exists only if the geometry of bluff or streamline body enforces the curvature of stream lines in that region). This substitution is based on this fact that in potential flow, the governing momentum equation is defined by Bernoulli equation. Therefore, the final engaged PDEs for the flow over a wedge (or a shape which has curvatures) can be denoted as follow:

$$\begin{aligned} &\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \\ &u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial x}=U_e\frac{dU_e}{dx}+\upsilon\frac{\partial^2 u}{\partial y^2} \end{aligned} \tag{6}$$

By definition of stream function based on the similarity function [6], x and y- component velocity can be achieved as:

$$\begin{aligned} &u=U_e f'(\eta) \\ &v=\frac{f'(\eta)}{2x}U_e y-\frac{f'(\eta)y}{2}\frac{dU_e}{dx}-\frac{1}{2}\sqrt{\frac{\upsilon U_e}{x}}f(\eta) \end{aligned} \tag{7}$$

In which $f(\eta)$ is the similarity function and η is the similarity parameter which is defined as follow:

$$\eta=\frac{y}{\delta} \tag{8}$$

The final Falkner Skan transformation equation can be written as:

$$\frac{dU_e}{dx}(f'^2-1)-\frac{ff''}{2}(\frac{U_e}{x}+\frac{dU_e}{dx})=\frac{U_e}{x}f''' \tag{9}$$

It is worth to site that Eq. (9) stands as x- momentum equation. And therefore, the effect of continuity equation has been considered in this equation by definition of stream function based on the similarity function. Further explanation on Eq. (9) is extensively presented in [6].

Eq. (9) is a third order ODE equation and then needs three initial or boundary conditions. Based on the physics of flow, the appropriate initial/ boundary conditions for this ordinary problem can be assumed to be:

$$\begin{aligned} &at\,\eta = 0, f\ (\eta)\ \&\ f\ '(\eta) = 0 \\ &at\,\eta = \infty, f\ '(\eta) = 1 \end{aligned} \tag{10}$$

$f\ '(\eta)$, merely indicates $u\ /U_e$, so in solving process, the value of η in which $f\ '(\eta)$ Reaches 0.99, can be assumed as the thickness factor of the boundary layer. Furthermore, ∞ is assumed to be 10 in the present work. In addition, the simulation has been continued for 1000 number of panels in which for solving Falkner Skan equation.

Each panel possesses a certain pressure (related to the velocity just above the boundary layer as the result of Vortex Panel Method). Since Eq. (9) is not an initial value ODE problem, it is usually solved by SHOOTING techniques [7]. Considering that an airfoil is not a uniform wedge, it should be divided into lines or segments (panels). Therefore it can be simply assumed that each panel plays the role of a wedge and each wedge requires solving of Eq. (9) by SHOOTING technique. Coming to the SHOOTING method, it must be sited that Eq. (9) is a non- linear equation. Then by interpolating or extrapolating of two assumptions of initial values, the precise solution will not be achieved. So, in the present work for being absolutely sure about the precision of the outcome results of the numerical procedure, the interpolation process was done based on the 3*106 assumptions of extra initial values. The solution of each wedge represents the boundary layer over its length. Accumulating these solutions, the boundary layer shape for the entire airfoil is achieved. By implementing the skin friction coefficient for each section, the total skin friction coefficient of airfoil is obtained. Furthermore, in this work, Eq. (9) was discretized by using a direct forward algorithm. For validating the developed solver code, it was applied for Blasius problem (see Fig. 5). As it can be seen from Eq. (9), Falkner- Skan equation is highly depended on free stream velocity over the boundary layer which is governed by potential flow regime. Using another Falkner- Skan transformation, the free stream velocity and its gradient can be easily related to the angle of wedge by potential flow analysis. The factor of this relation is β which is defined as:

$$\beta = 2m\ /\ (m+1) \tag{11}$$

Or, it is defined as: $\beta = \theta\ /\ \pi$ In which θ is the wedge angle (12)

Douglas Hartree [8] showed that physical solutions can only be found in the range − 0.0905 ≤ m ≤ 2. In which, m < 0 corresponds to an adverse pressure gradient (which is usually happens in boundary layer separation) while m > 0 indicates a favorable pressure gradient. This means that for airfoils which possess sharp negative angles, there is not exact physical solutions (this results cannot be expanded that Navier- Stokes equations fail to present exact solutions for this condition; and it just indicates that Falkner- Skan method has its limit which cannot be applied for all the desired angles of wedge because after the separation, the similarity factor would change). Based on the geometrical data of NACA0006, all the panels angles are in the above- mentioned range. So, in this case, there is not any limitation for applying Falkner- Skan analysis. An optimized self- developed Falkner- Skan code was used in the present work for simulating the flow over NACA0006. Semi- Infinite Plate is simulated by Blasius equation. $f\ ''(\eta)$ and $f\ '(\eta)$ as functions of η are derived in this case. In which, $f\ ''(\eta)$ relates to the y- direction derivative of x- component velocity and $f\ '(\eta)$ stands as the ratio of x- component velocity to the free stream velocity. Based on the simulation results, these two factors fall within about 0% error with the exact solution of Blasius equation (see Fig. 5). For calculating $C_f\ (x)$ as a function of cord direction, we have used the following procedure:

In general, for a straight line (oblique or flat), this factor can be defined as:

$$C_f\ (x) = \frac{k}{\sqrt{\mathrm{Re}_x}} \tag{13}$$

In which, k is a constant related to the angle of the oblique line (this constant is 0.644 for flat plate (angle of 0)). But according to this fact that an airfoil can be divided into finite numbers of attached lines (all of them are almost oblique), k will be then a factor of cord direction. This factor is equal to $2 * f\ ''(\eta)\big|_{\eta=0}$ in each panel. The total skin friction coefficient can be simply derived by integrating the local values of $C_f\ (x)$. For this, $C_f\ (x)$ was fitted by ninth order polynomials for each case study using Least Square technique [9- 13] with R- Square of about 0.99.

Finally, the total skin friction coefficient of C_f was obtained as a function of Reynolds number. For a flat plate, this factor is:

$$C_f = \frac{1.328}{\sqrt{\text{Re}}} \tag{14}$$

Based on the results of the present simulation, the constant of this factor for NACA0006 varied with the variation of angle of attack for both upper and lower surface of the airfoil (Fig. 6). In the case of zero angle of attack, this factor was obtained as follows:

$$C_f = \frac{1.503}{\sqrt{\text{Re}}} \tag{15}$$

The cumulative results for the first and second derivative of similarity function are gathered in Fig. 7 and 8.

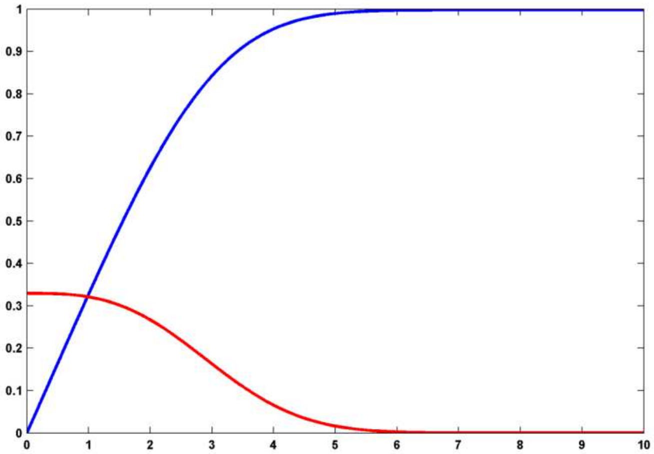


Fig. 5 Semi- Infinite Plate (Blasius Solution); Red Line: $f''(\eta)$ (vertical axis) vs. η (horizontal axis); Blue Line: $f'(\eta)$ (vertical axis) vs. η (horizontal axis)

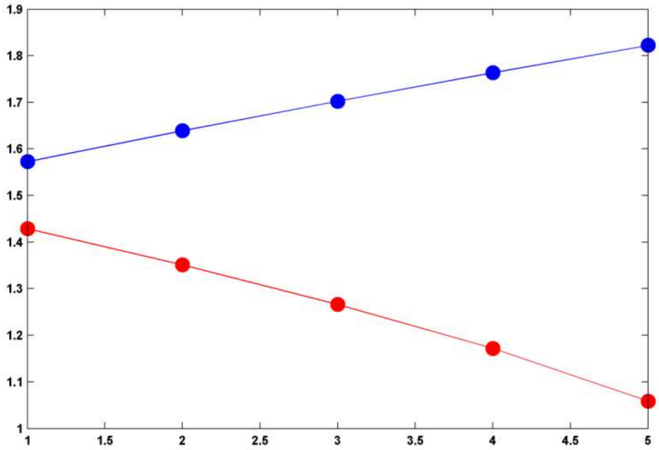


Fig. 6 $C_f\sqrt{\text{Re}}$ (vertical axis) for the upper (blue line) and lower (red line) surface of NACA0006 in different angle of attacks (horizontal axis)

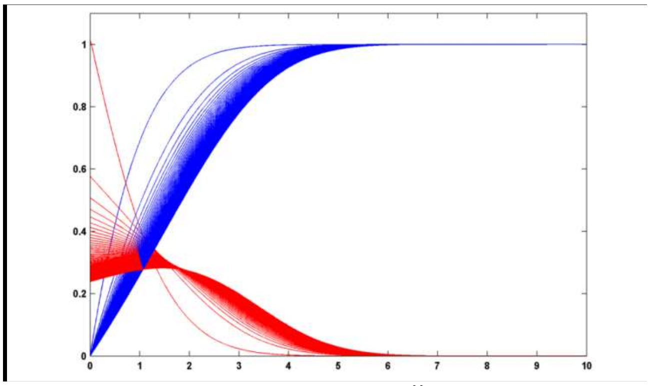
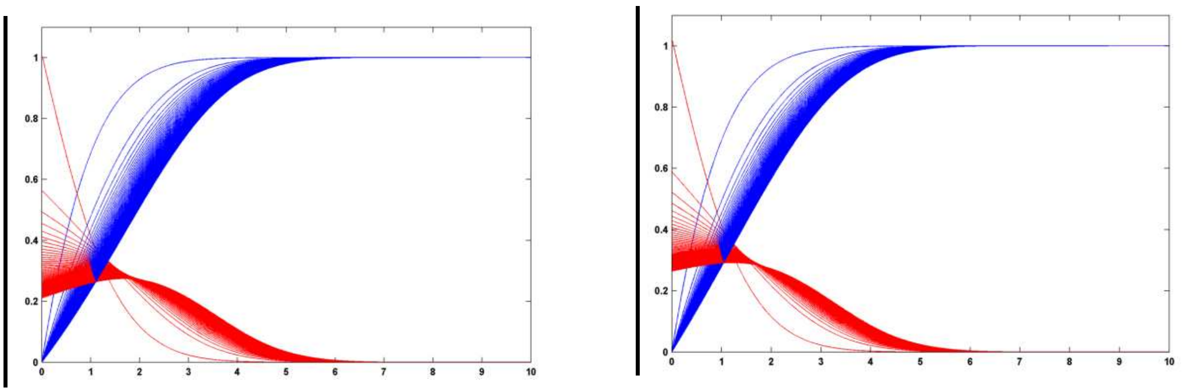


Fig. 7 NACA0006 in zero angle of attack (coupled Solution); Red Line: $f''(\eta)$ (vertical axis) vs. η (horizontal axis); Blue Line: $f'(\eta)$ (vertical axis) vs. η (horizontal axis); Note: The results are shown with the interval of 9 panels from the first panel to the almost last one. Also note that this figure stands for the both lower and upper surface of the airfoil



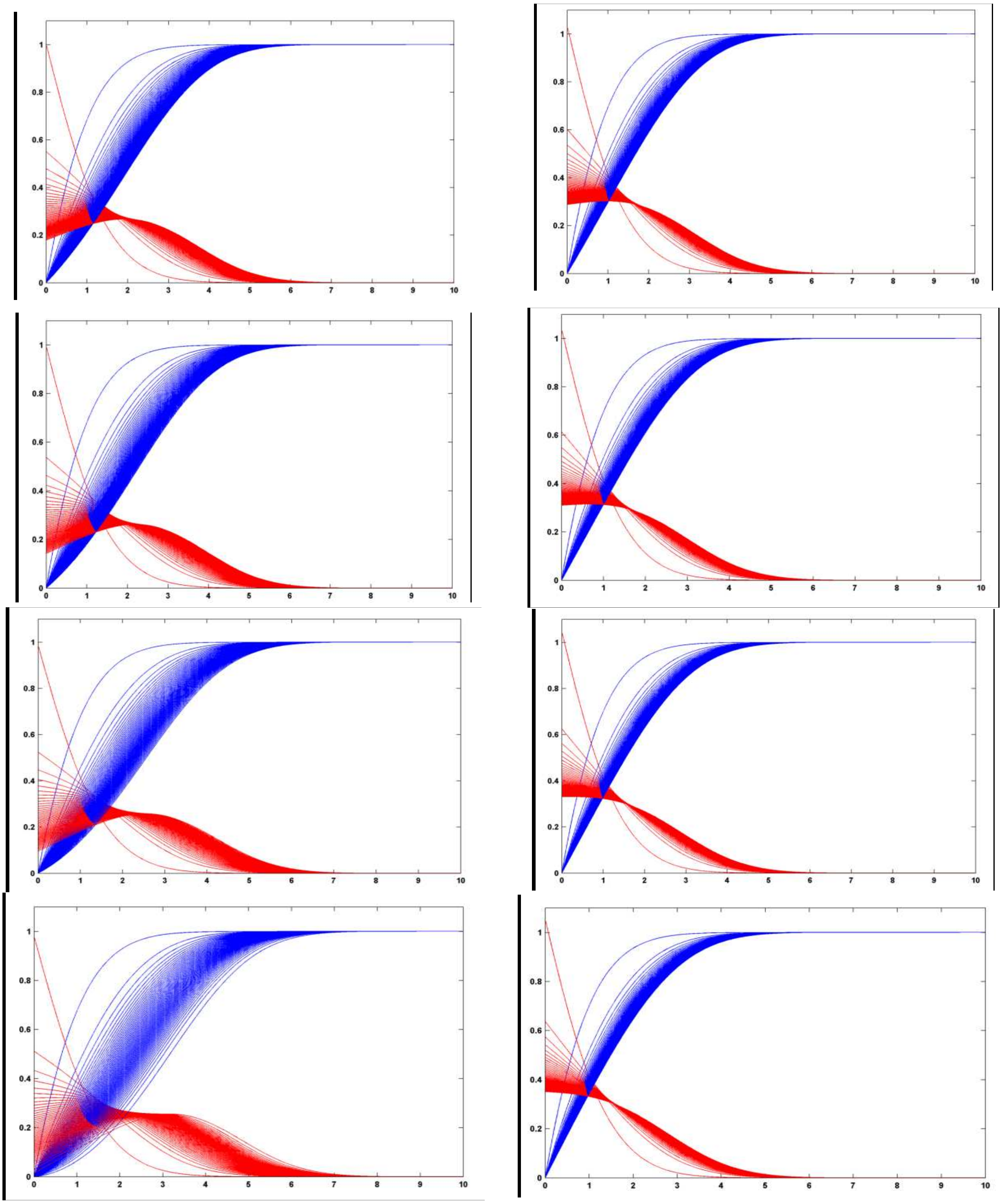


Fig. 8 NACA0006 in different angle of attacks (1º, 2º, 3º, 4º and 5º respectively) for the upper (left) and lower (right) surface of the airfoil (coupled Solution); Red Line: $f''(\eta)$ (vertical axis) vs. η (horizontal axis); Blue Line: $f'(\eta)$ (vertical axis) vs. η (horizontal axis); Note: The results are shown with the interval of 9 panels from the first panel to the almost last one

CONCLUSIONS

Vortex Panel Method and Falkner- Skan Methods were simultaneously used to simulate the flow over NACA0006 in an incompressible steady state flow regime. As it is clear, these methodologies can be only applied for incompressible flows. In this work, the simulation has been conducted in two- dimensional geometry. Since, lift coefficient is highly depended on pressure distribution and besides drag coefficient is related to the skin friction coefficient, an optimized self- developed Vortex Panel- Falkner Skan coupled solver was used for calculating these factors for NACA0006. The first and second derivatives of similarity function are shown for different angle of attacks. Moreover, the lift coefficient for the studied airfoil is shown and compared to that of thin airfoil theory. Finally, it is worth to state that this work stands as a primary attempt for achieving the most efficient airfoil shape for every desired flight condition. In the near future, by expanding the present methodology and including the induced drag effect and other engaged parameters in airfoil performance, we will proceed to reach the above-mentioned goal by assistance of Genetic Algorithm.

Acknowledgement
Authors would like to thank for the great helps of Dr. Mani Fathali in fulfilling the present research.

REFERENCES

[1] JM Anderson, K Streitlien, DS Barrett and MS Triantafyllou, Oscillating Foils of High Propulsive Efficiency, *Journal of Fluid Mechanics*, **1998**, 360(1),41–72.
[2] DA Read, FS Hover and MS Triantafyllou, Forces on Oscillating Foils for Propulsion and Maneuvering, *Journal of Fluids and Structures*, **2003**, 17(1), 163–183.
[3] BC Basu and GJ Hancock, The Unsteady Motion of a Two-Dimensional Aerofoil in Incompressible Inviscid Flow, *Journal of Fluid Mechanics*, **1978**, 87(1), 159–178.
[4] JL Hess, Panel Methods in Computational Fluid Dynamics, *Annual Review in Fluid Mechanics*, 1990, 22, 255–274.
[5] WJ Mc Croskey, Unsteady Airfoils, *Annual Review in Fluid Mechanics*, **1982**, 14, 285–311.
[6] V Ganesh, *Similarity Solution of the Boundary Layer Equation*, VI Indo-German Winter Academy, IIT Guwahati, 2007
[7] David D Morrison, James D Riley, F Zancanaro and F John (December 1962), Multiple Shooting Method for Two-Point Boundary Value Problems, ACM Communication, **1962**, 5 (12), 613–614.
[8] https.//en.wikipedia.org/wiki/Blasius_boundary_layer
[9] A Jafarimoghaddam, S Aberoumand, H Aberoumand and K Javaherdeh, Experimental Study on Cu/Oil Nanofluids through Concentric Annular Tube: A Correlation, *Heat Transfer-Asian Research*, doi, 10.1002/htj.21210, **2016**.
[10] A Jafarimoghaddam and S Aberoumand, An Empirical Investigation on Cu/Ethylene Glycol Nanofluid Through a Concentric Annular Tube and Proposing a Correlation for Predicting Nusselt Number, Alexandria Engineering Journal, **2016**, 55, 1047- 1052.
[11] Sadegh Aberoumand, Amin Jafarimoghaddam, Mojtaba Moravej, Hossein Aberoumand and Kourosh Javaherdeh, Experimental Study on the Rheological Behavior of Silver- Heat Transfer Oil Nanofluid and Suggesting Two Empirical Based Correlations for Thermal Conductivity and Viscosity of Oil Based Nanofluids, *Applied Thermal Engineering*, **2016**, 1,148.
[12] A Jafarimoghaddam and S Aberoumand, On the Implementation of Cu/ Ethylene Glycol Nanolfluids Inside Annular Pipe under Constant Wall Temperature Boundary Condition, *Heat Transfer-Asian Research*, doi, 10.1004/htj.21210, **2015**.
[13] S Aberoumand, K Javaherdeh, A Jafarimoghaddam and H Aberoumand, Synthesis and Rheological Behavior of Novel Ecofriendly Ag–Oil Nanofluid and Introduce General Correlations for Thermal Conductivity and Viscosity of Any Oil-Based Nanofluids, *Heat Transfer-Asian Research*, doi, 10.1002/htj.21193, **2015**.