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

COBEM-2019-0264
COMPARATIVE STUDY OF MESH DENSITY, SOLUTION ALGORITHM AND TURBULENCE MODELS IN AERODYNAMIC CHARACTERISTICS OF A NACA 0012 AIRFOIL

Rafael da Silva Rosário
Max William Frasão Reis
School of Mechanical Engineering, University of Campinas
rafael.s.rosario@gmail.com
maxreis@fem.unicamp.br
William D. P. Fonseca
School of Mechanical Engineering, University of Campinas
fonsecawdp@gmail.com

Abstract. *The study of two-dimensional, incompressible, permanent and turbulent flow over a National Advisory Committee for Aeronautics (NACA) 0012 airfoil at various angles of attack and operating at a Reynolds number of the order of 1E6 is presented. The flow was obtained by solving the continuity equations and Navier-Stokes combined with one of the three turbulence models (k-ϵ, k-ω and k-ω SST) and validation of these models were obtained by comparing them with the experimental results presented in the literature. The aim of the work was to show the behavior of the airfoil at these conditions and to establish a solution method. The computational domain was composed of two densities of the computational mesh emerged in a structured way, taking care of the refinement of the grid near the airfoil in order to enclose the boundary layer approach. Two algorithms for the pressure-velocity coupling solution were studied. Calculations were done for constant air velocity altering only the angle of attack for every turbulence model tested. The results showed that for moderate Reynolds number, the models of turbulence can be used with certain accuracy for prediction of aerodynamic characteristics.*

Keywords: *Aerodynamics, Computational Fluid Dynamics, Turbulence models*

1. INTRODUCTION

The rapid evolution of computational fluid dynamics (CFD) has been driven by the need for faster and more accurate methods for flow field calculations around configurations used in real engineering problems. These computational simulations show features and details that are difficult, expensive, or impossible to measure or visualize in wind tunnels. When it is desired to simulate flows on airfoils, the need for preliminary adjustments for numerical parameter evaluates, such as the study of mesh independence and run-time models is essential, since they optimize the quality and simulation time and play an important role in determining the characteristics of the airfoil. Flow and the quantification of airfoil performance, such as drag and lift coefficients.

Because of this important role that optimization simulations play in CFD analysis, many studies are being developed over the years. Bacha and Ghaly (2006) presented a model that combined existing methods to predict the beginning and the extension of the transition, which were compatible with the Spalart-Allmaras turbulence model, where the flow was simulated using the software ANSYS Fluent. The beginning of the transition was based on Michel (1952) method for compressible two-dimensional flow, while extension was quantified through the development of a model for an intermittent function. Badran *et al.* (2008) calculated the turbulence through a two-equation model and Reynolds stress model (RSM) to predict the separation of the boundary layer in NACA 4412 aerofoil in the stall. Yao *et al.* (2012) and Sadikin *et al.* (2018) performed numerical-dimensional simulations for wind turbine aircrafts and a NACA 0012 in high Reynolds numbers respectively using three different turbulence models (Spalart-Allmaras, k-ϵ and k-ω)

Krishnaswamy *et al.* (2014) investigated five different Reynolds Averaged Navier-Stokes (RANS) models at four different mesh densities in terms of aerodynamic characteristics of the NACA 0012 aerofoil in the High Numbers Reynolds. The purpose of the study was to choose the most accurate turbulence model to be used in simulations to determine the effect of the Gurney flap and synthetic jet on aerofoils.Suvanjumrat (2017) used open-source software, OpenFOAM, to simulate the flow around the NACA 0015 aerofoil. Three turbulence models for aerospace applications were used (Spalart-Allmaras, k-ϵ and k-ω SST).

In CFD simulations the first step in modeling a problem involves creating geometry and meshes with a preprocessor. Most of the time spent on an industrial CFD project is usually dedicated to mesh generation, so that the computational domain allows for a desired precision of the results and a relatively low cost of simulation. After the creation of the mesh, a solver is able to solve the ruling equations of the problem.

Finally, the results are examined and saved and, if necessary, revisions in numerical and physical models are made.Based on the premise that numerical optimization simulations with aerodynamic applications are extremely important, this article aims to study the best mesh density, solution algorithm for velocity coupling and turbulence model for a NACA 0012 airfoil. The draft is considered to be incompressible, permanent and in total turbulence regime. The Reynolds number used in the simulation is of the order of 1E6

2. NUMERICAL METHODS

2.1 Mathematical formulation

The flow of fluids are governed by the set of Navier-Stokes equations, these are given in their index form by Kundu *et al.* (2016) as:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u_i)}{\partial x_i} = 0 \tag{1}$$

$$\rho \frac{Du_i}{Dt} = -\frac{\partial p}{\partial x_i} + \rho g_i + \frac{\partial \rho}{\partial x_i}\left[2\mu e_{ij} - \frac{2}{3}\mu(\nabla.\vec{u})\delta_{ij}\right] \tag{2}$$

$$\rho \frac{D}{Dt}\left(e + \frac{1}{2}u_i^2\right) = \rho g_i u_i + \frac{\partial}{\partial x_j}(\tau_{ij}u_i) - \frac{\partial q_i}{\partial x_i} \tag{3}$$

In these equations, ρ represents the density, u_i the velocity, p the static pressure, μ dynamic viscosity, e_{ij} the strain rate tensor, e the kinetic energy and q_i is the heat flux.

The direct resolution of the Navier-Stokes equations using computational methods, although possible, is still not costly in engineering problems. This technique known as Direct Numerical Simulation (DNS), is able to solve the entire spectrum of turbulent scales without any numerical modeling, but the computational cost is still very high Pope and Pope (2000). In order to reduce computational cost, other methods for resolution of turbulent flows are used. Among them is the method known as Large Eddy Simulation (LES). This method is capable of solving the larger turbulent slates, which are more dependent on the geometry of the problem and the boundary conditions, while the smaller scales are modeled because they are more isotropic and therefore more universal. This method uses the Navier-Stokes equations filtered in space, and the filter is defined by the mesh used, and therefore, the smaller structures that filter are modeled. Such a method is still little used, due to the need for a very refined of domain discretization and require a large simulation time to ensure a stable statistical result of the modeled structures. Another approach for solving the Navier-Stokes equations is to use the Reynolds-Averaged Navier-Stokes (RANS), which are this equations taking the time-averaged variables and inserting a term, called Reynolds voltages. Such a model does not solve the turbulent scales, but calculates the temporal mean. This approach is used in several models, such as k-ϵ, k-ω e k-ω SST. Such models have a much lower cost computation than LES models, and are commonly used in engineering problems.

2.2 Turbulence models

For the comparative purposes of the present analysis, the RANS k-ϵ, k-ω e k-ω SST models were used. The turbulence models of two equations, initially proposed by Kolmogorov (1941), depart from the need for two independent scales to represent the effects of turbulence on the mean flow Menter and Egorov (2010). All models of two equations are based on the conservation equation of turbulent kinetic energy. The $k - \epsilon$ model, proposed by Lauder and Sharma (1974), consists of the utilization (4) and (5), the conservation equation of the rate of disembodiment of turbulent kinetic energy, according to Equations (4) and (5).

$$\frac{\partial k}{\partial t} + u_i\frac{\partial k}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\left(\frac{\nu_T}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right] + P_k - \epsilon \tag{4}$$

$$\frac{\partial \epsilon}{\partial t} + u_i\frac{\partial \epsilon}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\left(\frac{\nu_T}{\sigma_\epsilon}\right)\frac{\partial \epsilon}{\partial x_i}\right] + P_\epsilon - D_\epsilon \tag{5}$$

This model is relatively to implement, comparatively, a model of low computational requirements, being widely used. However, the model requires complex damping functions to allow the correct treatment on the walls making it with little robustness and accuracy.

Already k-ω model, proposed by Wilcox (1988), uses kinetic energy specific dissipation rate equation Eqs. (6) and (7)

$$\frac{\partial k}{\partial t} + u_i \frac{\partial k}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\left(\frac{\nu_T}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right] + P_k - \beta^* \rho k \omega \tag{6}$$

$$\frac{\partial \omega}{\partial t} + u_i \frac{\partial \omega}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\left(\frac{\nu_T}{\sigma_\omega}\right)\frac{\partial \omega}{\partial x_i}\right] + \alpha \frac{\omega}{k} P_k - \beta \rho \omega^2 \tag{7}$$

This approach has the advantage of having a simple and robust treatment close to wall, not needing wall functions in its implementation. However, this model is extremely sensitive to the values of ω in the far field field, an effect not present in k-ϵ.

The k-ω SST model, proposed by Menter (1994), appeared as an improvement of the k-ω modelo model using some advantages of the k-model. This model consists of the insertion of a conservation term in the equation of ω next to the competing function, presented in Eqs. (8) and (9). In this way the equation presents a behavior similar to k-ω in the regions near the wall, and similar k-ϵ for the treatment of far-field flow.

$$\frac{\partial k}{\partial t} + u_i \frac{\partial k}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\left(\frac{\nu_T}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right] + P_k - \beta^* \rho k \omega \tag{8}$$

$$\frac{\partial \omega}{\partial t} + u_i \frac{\partial \omega}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\left(\frac{\nu_T}{\sigma_\omega}\right)\frac{\partial \omega}{\partial x_i}\right] + \alpha \frac{\omega}{k} P_k - \beta \rho \omega^2 + \rho \frac{2(1-F)}{\sigma \omega} \nabla k \nabla \omega \tag{9}$$

In Eq. (9) F, the weighting function, defined by Eq. (10), where $CD_{k\omega}$ and turbulent viscosity are defined by Eqs. (11) and (12), respectively.

$$F = \tanh\left(min\left[max\left(\frac{\sqrt{k}}{\beta^* \omega y}, \frac{500\nu}{y^2 \omega}\right), \frac{4\rho\sigma_\omega k}{CD_{k\omega} y^2}\right]^4\right) \tag{10}$$

$$CD_{k\omega} = max\left(2\rho\sigma_\omega \frac{1}{\omega}\nabla k . \nabla \omega, 10^{10}\right) \tag{11}$$

$$\nu = \frac{ak}{max(a\omega, SF)} \tag{12}$$

2.3 Discretization method and problem modeling

The task of a numerical method is to solve one or more differential equations, replacing the existing derivatives in the equation with algebraic expressions involving the unknown function. With the numerical method of finite volume adopted, the simulations were carried out. As presented in Patankar (1980), Versteeg and Malalasekera (2007) and Maliska (2017) the procedure to obtain the discretized equations in the finite volume method consists of integrating, finite control, the differential equation in the conservative form.

As for the pressure-velocity coupling present in the governing equations, the SIMPLE (Semi Implicit Linked Equations) and PISO (Pressure-Implicit with Splitting of Operators) algorithms were used, where the first consists of creating an equation for the pressure, which allows the iterative process to progress, until the moment all the conservation equations involved are satisfied. PISO is an extension of SIMPLE, in which it does not require iterations and is conducive to transient problems.

The equations previously presented were solved through the CFD ANSYS / Fluent commercial software. The second-order UPWIND method was adopted for the treatment of the advective terms, a convergence factor of 0.001 for the variables pressure, velocity and continuity, and relaxation factors of 0.3 and 0.7 for the pressure and momentum, respectively. The contour conditions used were:

- Velocity prescribed at the input of the computational domain;
- Atmospheric pressure at the outlet;
- Wall condition in the airfoil to satisfy the non-slip condition.

3. RESULTS AND DISCUSSIONS

Based on the methodology presented previously, in this part of the work will be presented and discussed the results obtained. All the results were compared with the experimental data presented by Abbott (1934).

Figures 1 and 2 present the lift and drag curves for the study of the independence of the mesh. Two grades of mesh refinement were studied, one with 300,000 nodes the other with 800,000 nodes. It can be noticed, mainly through the lift curve, that the mesh with 800,000 nodes presented the best results, including picking up quite well the stall zone, which according to the literature is still an open problem for numerical aerodynamic analyzes.

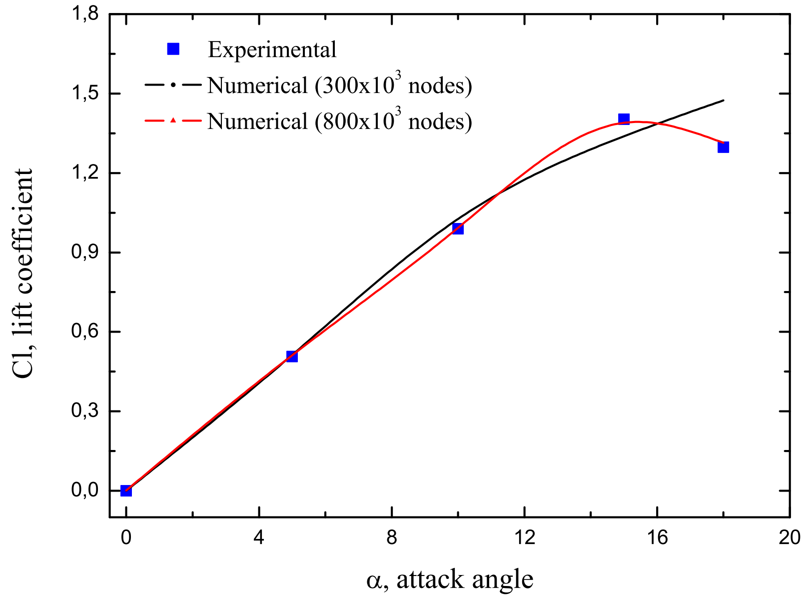


Figure 1: Lift curve for the mesh independence test.

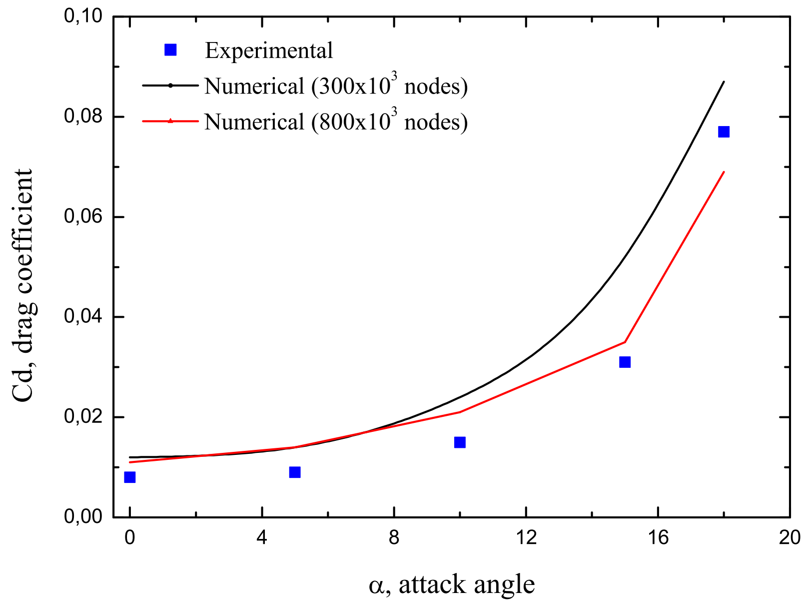


Figure 2: Drag curve for the mesh independence test.

Figure 3 shows the mesh with 800,000 nodes, which is used for the other analyzes. It is noted that there is a high refinement in the regions of the boundary layer and the flow wake, this is due to these regions being sensitively to the solution method.

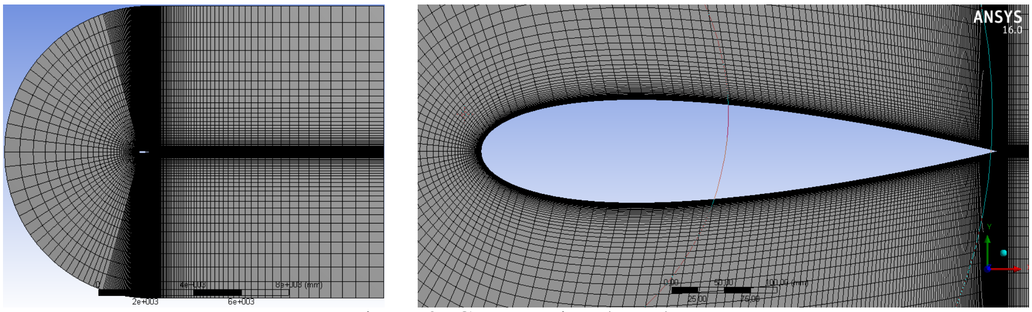


Figure 3: Computational mesh.

After the independence tests of the numerical mesh, analyzes were performed on the solution algorithm of the pressure-velocity coupling present in the Navier-Stokes equation. The SIMPLE and PISO algorithms were analyzed (Figs. 4 and 5), and SIMPLE presented the best results which may be due to the sweeps that the algorithm makes until the convergence of each time-step. It can be seen from Fig. (4) that the PISO diverge algorithm of the solution in the stall region. This can be credited to the permanent flow in which the airfoil is submitted, since the numerical formulation of the algorithm has as a characteristic to analyze transient cases and that could be corrected by adding a new level of the predictor-corrector operation in each time-step.

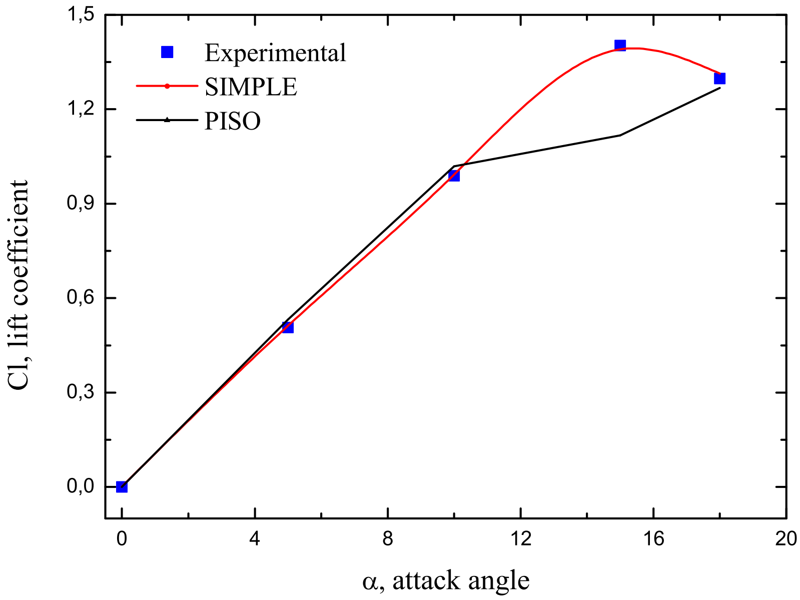


Figure 4: Lift curve for the solution algorithm test.

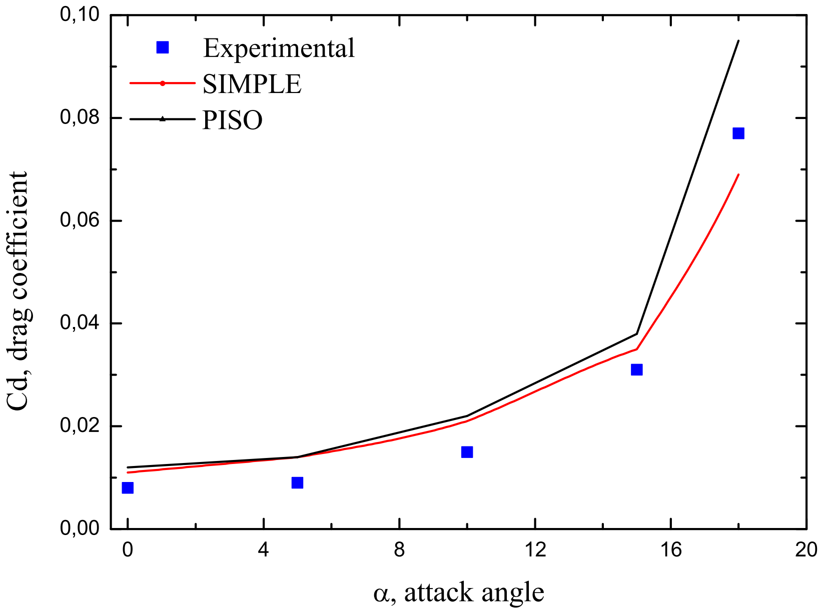


Figure 5: Drag curve for the solution algorithm test.

Finally, it was verified which turbulence model would best fit the solution of the problem. Fig. (6) shows this result, and through it it can be verified that the $k - \epsilon$ model presented the best results comparing with the experimental data, although not predicting the separation of the boundary layer with great precision, this can be credited with the higher smoothness provided by the model on the surface of the airfoil, contributing to reduction of vortex in the region of the wake and can also be credited the use of an algorithm based on the eddy viscosity. The $k - \omega$ model may not have presented a good result due to its formulation being very sensitive to the values of free-stream of k and ω outside the shear layer. Whereas this model does not employ damping functions. This can be corrected improving the mesh resolution near the all or using a variant of the model, the $SST - k - \omega$ combining the original Wilcox $k - \omega$ model for use near walls and the standard $k - \epsilon$ model away from walls using a blending function, which generated better results and that can be improved by increasing the resolution of the mesh near the wall.

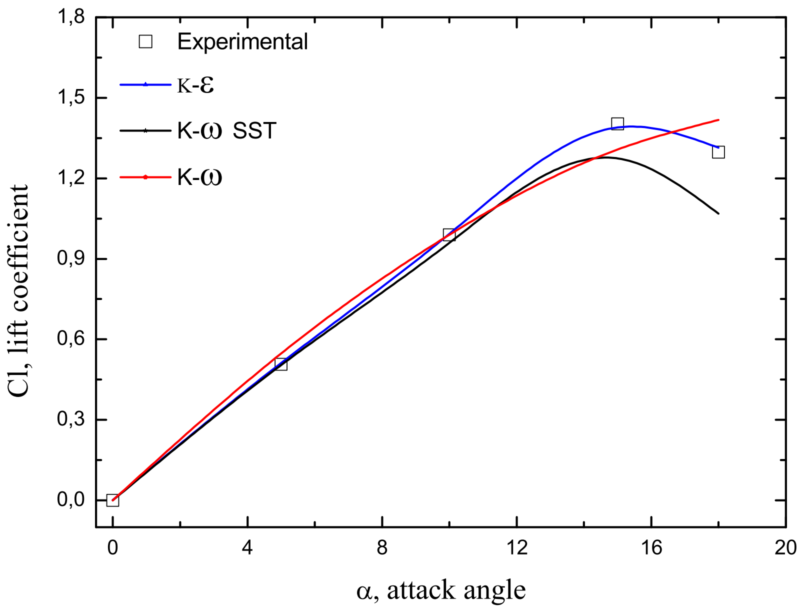


Figure 6: Lift curve for turbulence model verification.

With the 800,000 nodes mesh, the SIMPLE algorithm established for the pressure-velocity coupling solution and the $k - \epsilon$ model for turbulence modeling, the velocity (Tab. 1) and pressure fields (Tab. 2) were compared with literature, more precisely the theory presented by Anderson Jr (2007). These analyzes were performed with angles of attack equal to $0, 10, 13$ e $14°$, respectively, and showed to be in agreement with what is presented by the aeronautical literature with an acceptable accuracy.

Table 1: Velocity fields.

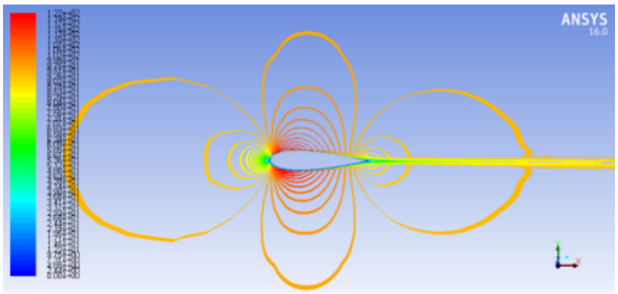
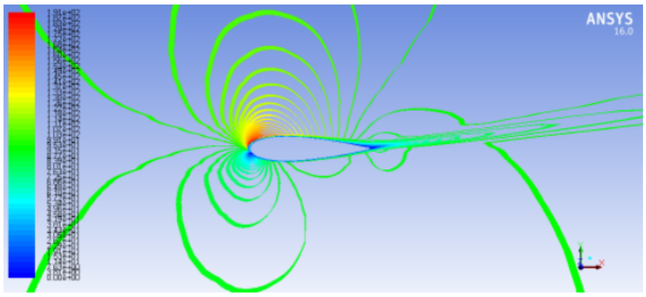
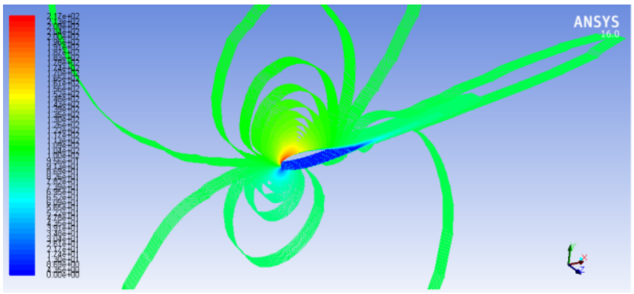
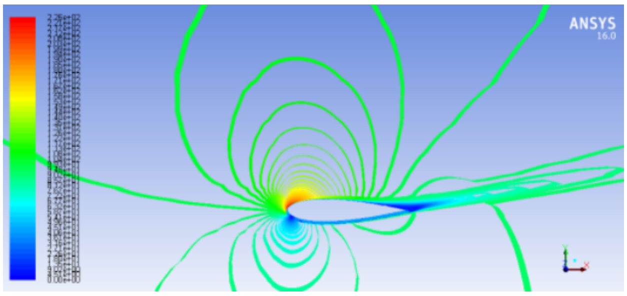
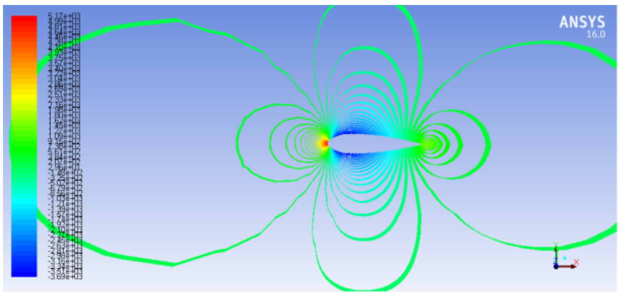
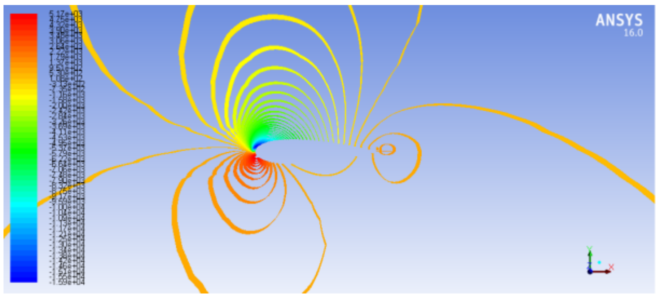
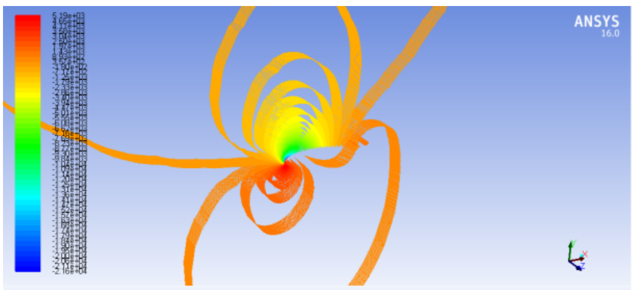
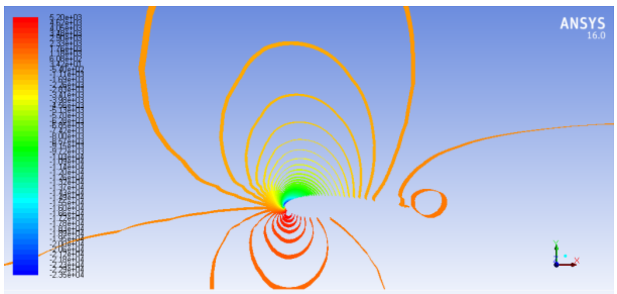
 α=0°	 α=10°
 α=13°	 α=14°

Table 2: Pressure fields.

 α=0°	 α=10°
 α=13°	 α=14°

4. CONCLUSIONS

From the simulations, it can be observed that if we consider a two-dimensional, incompressible, permanent regime and turbulent flow, with Re = 1E6, the SIMPLE method for the solution of the pressure-velocity coupling,due the sweeps that the algorithm makes until the convergence of each time-step, and the k-ϵ for turbulence modeling, due the algorithm based on the eddy viscosity and the damping in the turbulence, are the ones with the best results. Even though not so good in predicting the separation of the boundary layer with great precision, in most cases.

Future work will be developed to test more parameters, such as, more refined meshes. Since the Mach number is low, around 0.042, no need to consider the fluid as being ideal gas. Furthermore more models of turbulence, including the LES, could be used to obtain greater comparisons between different aerodynamic flow solution methods.

5. REFERENCES

Abbott, I.H., 1934. "The drag of two streamline bodies as affected by protuberances and appendages".

Anderson Jr, J.D., 2007. *Fundamentals of aerodynamics*. Tata McGraw-Hill Education.

Bacha, W. and Ghaly, W., 2006. "Drag prediction in transitional flow over two-dimensional airfoils". In *44th AIAA AeroSpace Sciences Meeting and Exhibit*. p. 248.

Badran, O., Aldudak, R.Q.F. and und Aerodynamik, F.S., 2008. "Two-equation turbulence models for turbulent flow over a naca 4412 airfoil at angle of attack 15 degree". *Mechanical Engineering Department Faculty of Engineering technology, Al-BalqaApplied University, PO Box*, Vol. 331006.

Kolmogorov, A.N., 1941. "Equations of turbulent motion in an incompressible fluid". In *Dokl. Akad. Nauk SSSR*. Vol. 30, pp. 299–303.

Krishnaswamy, S., Jain, S. and Sitaram, N., 2014. "Grid and turbulence model based exhaustive analysis of naca 0012 airfoil". *Journal of Advanced Research in Applied Mechanics & Computational Fluid Dynamics*, Vol. 1, No. 1, pp. 13–18.

Kundu, P.K., Cohen, I.M. and Dowling, D.R., 2016. "Fluid mechanics".

Maliska, C.R., 2017. *Transferência de calor e mecânica dos fluidos computacional* . Grupo Gen-LTC.

Menter, F.R., 1994. "Two-equation eddy-viscosity turbulence models for engineering applications". *AIAA journal*, Vol. 32, No. 8, pp. 1598–1605.

Menter, F. and Egorov, Y., 2010. "The scale-adaptive simulation method for unsteady turbulent flow predictions. part 1: theory and model description". *Flow, Turbulence and Combustion*, Vol. 85, No. 1, pp. 113–138.

Michel, R., 1952. "Determination of transition point and calculation of drag of wing sections in incompressible flow". *ONERA Publication*, Vol. 58.

Patankar, S., 1980. *Numerical heat transfer and fluid flow*. CRC press.

Sadikin, A., Yunus, N.A.M., Hamid, S.A.A., Salleh, S.M., Rahman, M.N.A., Mahzan, S., Ayop, S.S. *et al.*, 2018. "A comparative study of turbulence models on aerodynamics characteristics of a naca0012 airfoil". *International Journal of Integrated Engineering*, Vol. 10, No. 1.

Suvanjumrat, C., 2017. "Comparison of turbulence models for flow past naca0015 airfoil using openfoam". *Engineering Journal*, Vol. 21, No. 3, pp. 207–221.

Versteeg, H.K. and Malalasekera, W., 2007. *An introduction to computational fluid dynamics: the finite volume method*. Pearson Education.

Wilcox, D.C., 1988. "Reassessment of the scale-determining equation for advanced turbulence models". *AIAA journal*, Vol. 26, No. 11, pp. 1299–1310.

Yao, J., Yuan, W., Xie, J., Zhou, H., Peng, M., Sun, Y. *et al.*, 2012. "Numerical simulation of aerodynamic performance for two dimensional wind turbine airfoils". *Procedia Engineering*, Vol. 31, pp. 80–86.

6. RESPONSIBILITY NOTICE

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