

S809 AIRFOIL: REYNOLDS NUMBER EFFECT ON THE AERODYNAMICS OF WIND TURBINE BLADES

João C. Gaigher Junior

Julio Cesar C. Campos

Álvaro M. Bigonha Tibiriça

Henrique M. Pereira Rosa

Matheus M. Beraldo

Universidade Federal de Viçosa – Departamento de Engenharia de produção e Mecânica. Rua PH Rolfs s/n

joao.gaigher@ufv.br

julio.campos@ufv.br

alvaro.tibirica@ufv.br

henrique.rosa@ufv.br

matheus.beraldo@ufv.br

Antônio Carlos de Andrade

Universidade Federal de Minas Gerais – Departamento de Engenharia Mecânica. Av. Antônio Carlos, 6627

andrade@demec.ufmg.br

Rogério Fernandes Brito

Universidade Federal de Itajubá – Campus Avançado de Itabira. Rua Irmã Ivone Drumond, 200, Distrito Industrial II

rogbrito@unifei.edu.br

Pedro Casanova Treto

Universidad de Costa Rica – Instituto de Investigaciones en Ingeniería – INII. San Pedro de Montes de Oca

pcasanova2000@gmail.com

Abstract. The objective of this work is to study the S809 airfoil profile used in the wind turbine blades of horizontal axis. Wind turbines developed rapidly in technological terms, showing reliability in generating energy from the wind. The methodology employed was the aerodynamics characterization from simulations on CFD to get the parameters such as lift, the drag coefficient, the relationship between lift and drag, speed, and the pressure contours for different angles of attack. Simulation results using ANSYS FLUENT® v16.1 presented important features. The focus was in the study of Reynolds numbers of about 75,400 and 134,500, using the turbulence model κ -omega (κ - ω) Standard and model (κ - ϵ) realizable. Stall has been noticed from 15°, where verify the detachment of the boundary layer, as well as, the effect of pressure through the speed contours and the simulated pressure for this angle of attack. The blade inclination of wind turbines are important when the design of such turbines is analyzed. Concludes that numerical study of the airfoil S809 served to characterize the aerodynamic profile of the blades of horizontal axis wind turbines.

Keywords: aerodynamic coefficients, numerical simulation, turbulence model, S809 airfoil

1. INTRODUCTION

Wind turbines were first used to produce electricity in the twentieth century, in response to accelerated economic growth and increased power consumption. Nowadays wind energy is highly desirable for being a non-polluting and theoretically inexhaustible mode of power generation. Another relevant factor is the strong ecological appeal present in the wind energy generation, often listed as a requirement in current projects.

A class of lower-upper symmetric Gauss-Seidel implicit weighted essentially non oscillatory (WENO) schemes was developed by Huang, *et al.*, 2009 for solving the preconditioned Navier-Stokes equations of primitive variables with Spalart-Allmaras one-equation turbulence model. The computations are performed for the two-dimensional lid driven cavity flow, low subsonic viscous flow over S809 airfoil, three-dimensional low speed viscous flow over 6:1 prolate spheroid, transonic flow over ONERA-M6 wing and hypersonic flow over HB-2 model. The viscous flow over S809 airfoil was considered at moderate angle of attack (AOA). The S809 airfoil was designed specially for horizontal-axis-wind-turbine (HAWT) applications. The thickness ratio of the airfoil is 21%. A 600 mm chord length model of the S809 airfoil has been tested in a 1.8 m x 1.25 m low turbulence wind tunnel (Ghia, *et al.*, 1982). Four grid systems of C-type topology are generated for grid-independency testing. The grid points are 185 x 49, 273 x 73, 409 x 109 and 601 x 145 from coarse to fine grid system, and there are 137, 207, 321 and 481 points on airfoil surface, respectively, and the outer boundary for all four grid systems is about 20 chord lengths away from airfoil. The free stream Mach number is 0.001,

Reynolds number is 2×10^6 and AOA is 9.22° . For all the cases computed, the solutions of the present algorithms are in very good agreement with the available experimental data.

Investigation of the effects of near-wall grid spacing for the SST- κ -omega (κ - ω) model and study of the aerodynamic behavior of a horizontal axis wind turbine are the two goals of the Moshfeghi, *et al.* 2012's paper. The National Renewable Energy Laboratory (NREL) Phase VI is used as the aerodynamic model. Eight different cases are investigated for the near wall grid spacing study. Furthermore, one case is studied in both the SST- κ -omega (κ - ω) and the Langtry-Menter transitional models. For all cases the total number of nodes are fewer than 5,000,000. The results of this paper lead us to a general understanding about the wind turbine blade aerodynamics. Thrust forces, flow patterns and pressure coefficients are compared at different wind speeds. The thrust values of the SST- κ -omega (κ - ω) are not in a good agreement with the test results. The streamlines show that the inboard section of the blade has a severe complex 3D flow which separates at low velocities; the mid-span section stays attached for higher velocities and the outboard part has 2D-like behavior and separates as the last part. Besides, it is observed that Gamma-Theta transitional model behaves differently from the SST- κ -omega (κ - ω), especially at the inner part and the results are closer to the test results.

Airfoil aerodynamics research appears to be more concentrated on conventional aircraft design with Reynolds numbers above 5×10^5 and below stall conditions.

Countless aerodynamic profiles have been proposed as a result of the intense efforts made by the scientific community to improve the efficiency with which wind power is used. However, special attention is required when the S809 airfoil is considered, as it has been designed to operate in turbines at low speeds.

In this context the development of an experimental project on the S809 airfoil is justified. The main purpose is to assess airfoil aerodynamic performance at various angles of attack.

Therefore, justified the development of a project for the numerical study of the S809 airfoil, primarily used in horizontal wind turbine blades to meet the aerodynamic behavior for different angles of attack and Reynolds numbers of about 75,400 and 134,500, using the turbulence model κ -omega (κ - ω) Standard and model (κ - ϵ) realizable.

2. COMPUTATIONAL METHODS

The first step of the numerical solution involves the description of the physical phenomenon, i.e., the physical variables acting on and affecting the physical system (Fortuna, 2000). Then a numerical model has to be designed with equations of conservation of mass, momentum, and energy that are valid within a given domain. The domain must be clearly defined in order for the numerical solution to accurately represent the analyzed flow conditions.

The analysis included variables pressure, velocity, and other parameters related to these variables as observed in an S809 airfoil. Software package ANSYS FLUENT[®] v16.1 was used to produce a physical representation of the flow patterns over an S908 airfoil. Control volumes are cell-centered in this code, whereas on the CFX code control volumes are node-centered.

The computational method used in this code is the finite volume method. This discretization scheme uses conservation equations in integral form. Cebeci, *et al.*, 2005 presented a generic conservation equation in integral form as follows:

$$\frac{\partial}{\partial t} \iiint U d\Omega + \iint \vec{F} \cdot d\vec{S} = \iiint Q_v d\Omega + \iint Q_s dS \quad (1)$$

where $\iiint U d\Omega$ represents the integral form of unknown control volume variable U , $\iint \vec{F} \cdot d\vec{S}$ is the integral form of flow vector $\vec{F}$ on the control surface, $\iiint Q_v d\Omega$ is the integral form of control volume variable Q_v , $\iint Q_s dS$ is the integral form of control surface variable Q_s , $d\Omega$ is the control volume and dS the control surface.

The variables of interest in Equation (1) are mass flow, momentum, and energy inside and outside the control volume. After discretization, it yields the conservation equation for each cell in the physical space, as shown in Eq. (2):

$$\frac{\partial(U_{ij}\Omega_{ij})}{\partial t} + \sum_{sides}(\vec{F} \cdot \vec{S}) = (Q_v)_{ij}\Omega_{ij} + \sum_{sides}(Q_s)_{ij}S \quad (2)$$

Equation (2) is then applied to elicit the equations for continuity, momentum, and energy for each cell in the physical space. Thus, the S809 airfoil domain was discretized for this set of control volumes.

Isothermal flow was analyzed in the simulations carried out on the S809 airfoil, which by its turn called for the analysis of turbulence. Many authors such as Neto (2002) and Freire (2002) have indicated that turbulence is present in most flow patterns. In this study turbulence was described as a highly diffuse, dissipative, rotational, and three-dimensional phenomenon.

The simulation of complex flow patterns requires an understanding of the basic principles of the theory of turbulence and the means used to model it. With these principles in mind, the best turbulence model and near-wall treatment options for the problem at hand were chosen. Flow around the S809 airfoil was thus characterized as external and turbulent.

The dimensionless parameter used in flow characterization was the Reynolds number, as defined in Eq. (3):

$$Re_c = \frac{Vc}{\nu} \quad (3)$$

where Re_c is the Reynolds number for the S809 airfoil chord length c , c is the S809 airfoil chord length in meters, V is the air flow velocity in m/s over the S809 airfoil, and ν is the kinematic viscosity of the fluid over the S809 airfoil in m^2/s .

In this study the turbulence over the S809 airfoil was simulated using the RANS (Reynolds-averaged Navier-Stokes) equations. In this method the equations are obtained from a set of averaged Navier-Stokes and continuity equations. A key element in RANS modeling is the representation of the Reynolds stress, or turbulent stress, used to describe the effects of turbulent fluctuations in pressure and velocities.

This particular method was chosen for the modeling quality it provides and the computational cost it offers to produce turbulence models. Many authors have described the low computational cost and high grade modeling the RANS method offers vis-à-vis other turbulence models. More specifically, the Standard κ -omega (κ - ω) turbulence model – a method known in the literature for its applications in the description of aerodynamic flow – was used in this study, along with the Realizable κ - ϵ model.

2.1 Geometry and Mesh

The domain was discretized, i.e., divided into points for which a solution was obtained. The set of discretized points is referred to as the mesh.

The geometry of the S809 airfoil was designed on software program Solidworks®. The resulting geometry was then exported to the ANSYS® ICEM-CFD® mesh generator. Figure 1 shows the geometry of the S809 airfoil designed on Solidworks®. The mesh defines the cells in the computational domain for which flow variables such as velocity and pressure are calculated.

Figure 1 shows the asymmetric profile of the S809 airfoil, with a convex upper surface, a rounded edge called leading edge, and a sharp edge referred to as the trailing edge. The asymmetry stems from the shape of the lower surface of the S809 airfoil.

Asymmetric S809 airfoils are optimized to generate higher levels of lift when the lower surface of the airfoil is positioned closer to the direction of airflow (Fadigas, 2011).

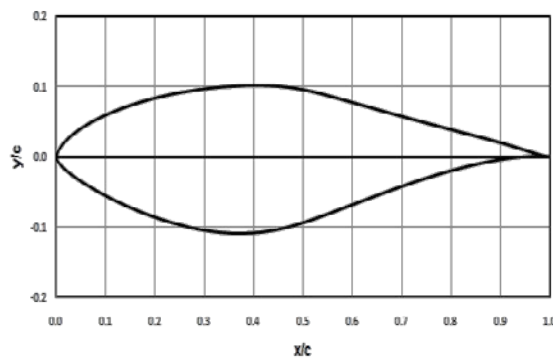


Figure 1. The profile of an S809 airfoil

The ANSYS® ICEM CFD meshing software was used to generate a C-H mesh for the S809 airfoil. This is one of the most critical steps of the simulation effort, once it yields a representation of a continuous domain of interconnected points from which the equations governing the physical phenomenon are solved.

Figure 2 (a, b) shows the respective representations of the 2D mesh and the C-H mesh over the S809 airfoil surface.

The mesh contained 1,528,900 elements. In order to enhance the modeling of drag forces and turbulence, the mesh was extended to cover points near the wall, with the first point on the grid located in the viscous sublayer, $y^+ < 10$. The grid extended in the radial direction for a length equivalent to $20 \cdot c$ from the center of gravity of the S809 airfoil.

The length of $20 \cdot c$ was needed to ensure the computational domain was large enough not to allow reverse flows.

The structured mesh had flat quadrilateral cells around the S809 airfoil as shown in Fig. 2 (b).

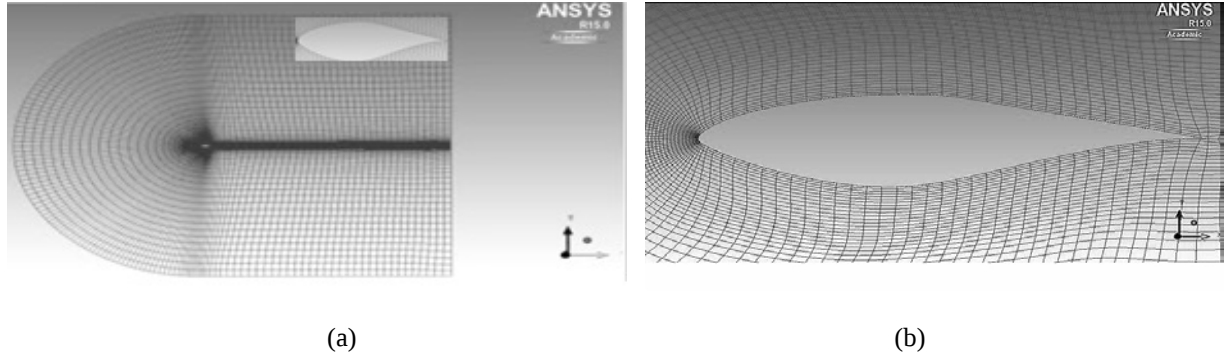


Figure 2. Structured mesh: (a) C-H mesh generated on ICEM-CFD®; (b) mesh over the profile of the S809 airfoil

2.2 Boundary Conditions

Proper boundary conditions have to be set if one wishes to obtain CFD solutions that represent the physical phenomenon more closely. The simpler of the boundary conditions applies to the wall. Once in this case the fluid cannot pass across the wall, the normal velocity component is equal to zero, thus characterizing a prescribed wall boundary condition. Therefore, the walls were considered adiabatic and a no-slip boundary condition was applied to them.

Similarly, boundary conditions were defined at the inlet and exit of the S809 airfoil. In this scenario, the fluid – air – entered and left the computational domain.

In this study, inlet velocity was set to range from 1 to 5 m/s and ambient pressure away from the S809 airfoil was defined as the specified pressure condition. Pressure and velocity close to the S809 airfoil, i.e., in its upper and lower surfaces, were calculated for Reynolds numbers of 75,400 and 135,400, respectively.

3. RESULTS AND DISCUSSIONS

The use of computational fluid dynamics (CFD) allows to analyze a model when exposed to different flow conditions. This technique is commonly used in the analysis of aerodynamic behavior at different types of airfoil. This analysis can be made in two-dimensional models (2-D) and in three-dimensional (3-D).

In some studies, there is no interest in the influences generated by edge effects, being only the results for the mid span sections of the airfoil important. In such cases the 3-D models have similar features to the 2-D (Trolborg, 2013).

According to Logsdon (2006), the use of 2-D and 3-D models under the same simulation conditions showed very similar results. This makes valid the use of 2-D models, since they are more simplified and consumes less time and processing than 3-D models. Therefore, in this study we used the 2-D model to represent the physical phenomenon in airfoil S809.

The mesh has high importance in a FLUENT analysis. The quality of the elements will influence in the accuracy and stability of the simulation. Thus it is essential to check the quality of the mesh used.

In meshes generated by ICEM-CFD, assessing the quality of the elements can be done through the tool "Quality Histogram". There, a scale from 0 to 1 is established, 0 is the worst quality and 1 the best one (FLUENT Database), and the number of elements per quality range is displayed. The minimum quality of a mesh should be greater than 0.3 to perform a simulation. In this study the quality of the elements, as shown in FIG. 2 was 0.9 (Skewness) and an aspect ratio less than 100, which features a mesh with good quality.

The influence of mesh refinement were used as output data C_l , C_d , C_p , residue and the historic convergence of these parameters. The residue scale for the mesh shown in Fig. 2 was of the order of 10^{-6} . Then, they were compared with the data of Fig. 5. Although these results have been presented for $Re = 75,400$ and $134,500$ it was realized a qualitative similarity to the data of Fig. 5.

Figures 3, 4, and 5 show the drag (C_d) and lift coefficients (C_l) derived from Equations (4) and (5), respectively:

$$C_d = \frac{Fd}{0.5\rho V^2 A} \quad (4)$$

where Fd is the drag force in Newtons, ρ is the specific weight of the air in kg/m^3 , and A is the platform area of the S809 airfoil in meters. This area is the product of the chord times the wingspan, as described by (Campos *et al.*, 2014).

$$C_l = \frac{Fl}{0.5\rho V^2 A} \quad (5)$$

where Fl is the lift force in Newtons. Standard κ -omega (κ - ω) and realizable (κ - ϵ) turbulence models.

Figure 3 (a,b) shows the drag (C_d) and lift (C_l) coefficients as a function of the angle of attack α in degrees, for a Reynolds number (Re) of 75,400 using the realizable ($\kappa-\epsilon$) and standard κ -omega ($\kappa-\omega$) turbulence models, respectively.

Figure 3 (a) shows an increase in C_d when α is increased in both models. The maximum and minimum values for C_d in the standard κ -omega ($\kappa-\omega$) model were 0.14 and approximately 0.04, respectively. Likewise, the maximum and minimum values for C_d in the realizable ($\kappa-\epsilon$) model were 0.1 and 0.02, respectively. Differences were generally observed between the analyzed turbulence models.

Figure 3 (b) illustrates a similar pattern, in which the slope of the lift coefficient (C_l) was close to 15° in the standard κ -omega ($\kappa-\omega$) model. The slope in the realizable ($\kappa-\epsilon$) model could not be clearly established. The slope observed in the standard κ -omega ($\kappa-\omega$) model characterizes stall, or loss of lift from the S809 airfoil.

This slope can also be seen in Fig. 5 (b) for experimental results and in the numerical results described by (Gharali and Johansen, 2012, adapted).

The boundary layer separates at an angle of attack $\alpha = 15^\circ$; therefore, the realizable ($\kappa-\epsilon$) model did not suffice to illustrate such slope.

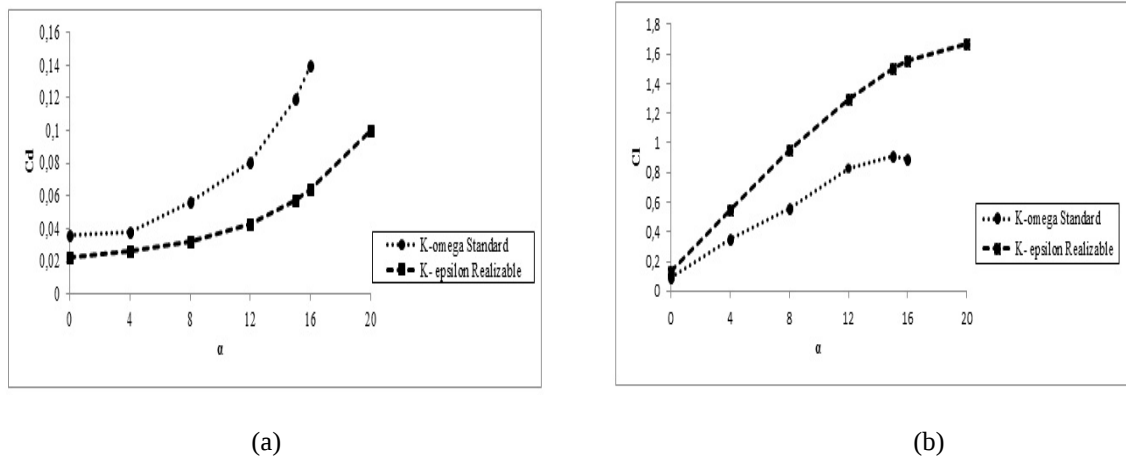


Figure 3. Drag (a) (C_d) and lift (b) (C_l) coefficients for a Reynolds (Re) number of 75,400

Figure 4 (a,b) shows the drag (C_d) and lift (C_l) coefficients as a function of the angle of attack (α) in degrees for a Reynolds (Re) number of 134,500 using the realizable ($\kappa-\epsilon$) and standard κ -omega ($\kappa-\omega$) turbulence models, respectively. The Figure also reveals that closer results were obtained from the studied turbulence models as Re increased. The models also produced closer results for angles ranging between 0 and 8° , but Fig. 4 (b) shows that the realizable ($\kappa-\epsilon$) model failed to elicit the slope at 15° .

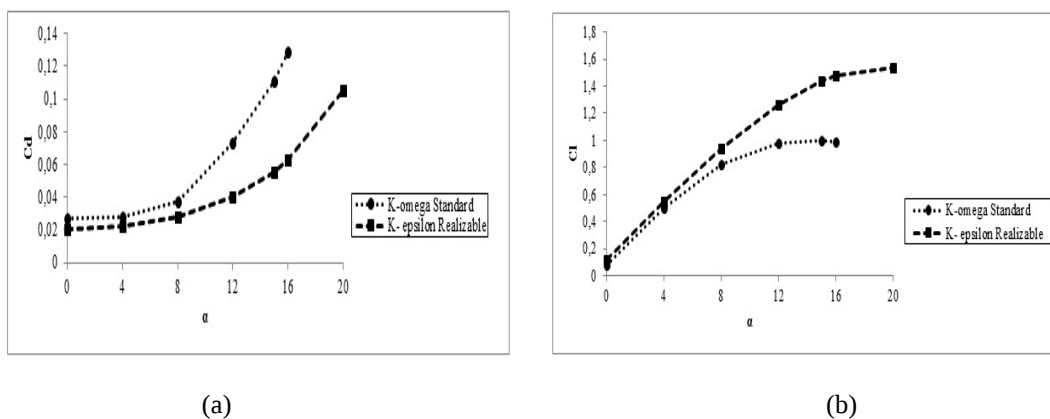


Figure 4. Drag (a) (C_d) and lift (b) (C_l) coefficients for a Reynolds (Re) number of 134,500

Figure 5 (a,b) shows the drag (C_d) and lift (C_l) coefficients as a function of the angle of attack (α) in degrees for a Reynolds (Re) number of 10^6 using the realizable ($\kappa-\epsilon$) and the SST- κ -omega ($\kappa-\omega$) turbulence models, respectively. This Figure shows the experimental and numerical results published by Gharali and Johansen (2012).

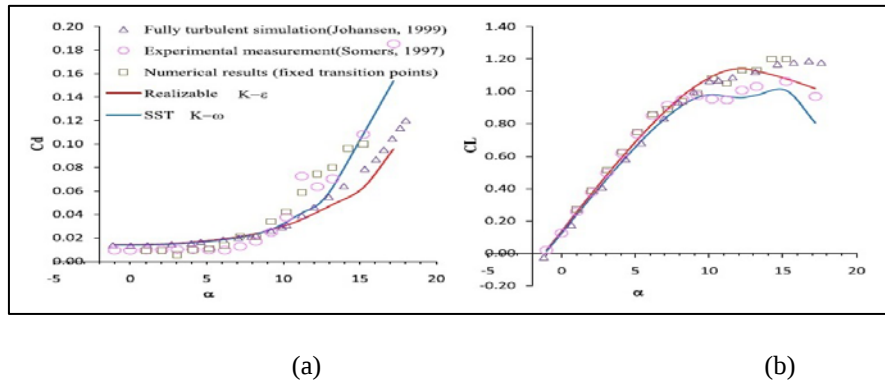


Figure 5. Drag (a) (C_d) and lift (b) (C_l) coefficients for a Reynolds (Re) number of 10^6 , (Gharali and Johansen, 2012, adapted)

A comparison between Fig. 4 (a,b) and Fig. 5 (a,b) reveals that the results of the standard κ -omega (κ - ω) and SST- κ -omega (κ - ω) turbulence models drifted away from the results of the realizable (κ - ϵ) model, while the values for drag and lift coefficients remained quite close to each other. The numerical results seen in Figure 5 (a,b) were close to the observed experimental results.

Figure 6 shows the lift-to-drag ratio (Cl/Cd) as a function of the angle of attack (α) in degrees for Reynolds (Re) numbers of 75,400 and 134,500 using the realizable (κ - ϵ) model and the standard κ -omega (κ - ω) model.

This figure illustrates a design point called the drag polar, a point at which the Cl/Cd ratio reaches its maximum value. It may be referred to as $(Cl/Cd)_{max}$ or maximum efficiency, E_{max} .

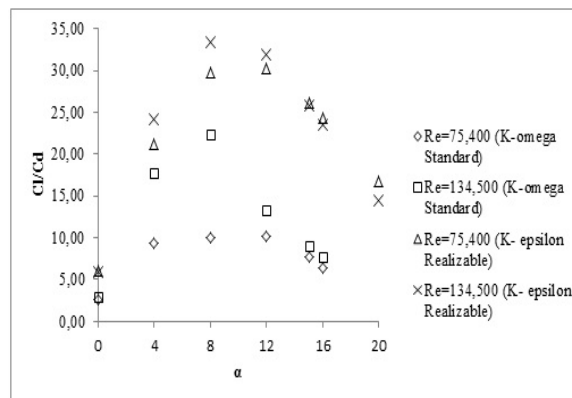


Figure 6. Lift-to-drag ratio (Cl/Cd)

Figure 6 shows that the lift-to-drag ratio reached its peak value when $\alpha = 8^\circ$ for all studied Reynolds numbers and turbulence models. This is the angle at which horizontal axis wind turbines reach their maximum efficiency.

This point represents the angle of attack at which it is possible to maintain maximum lift with minimum drag, and significantly affects airfoil performance and the design of wind turbine blades.

The ratio increased as flow velocity increased, and reached its maximum for $Re = 134,500$ and $Cl/Cd \approx 32$ and 22 in the realizable (κ - ϵ) and the standard (κ - ω) models, respectively.

Figures 7 to 10 show the velocity (m/s) and static pressure (Pa) contours for $Re = 75,400$ and 134,500 at $\alpha = 4^\circ$ and 15° for the realizable κ -epsilon (κ - ϵ) model, respectively.

In profiles inclined at a positive angle in relation to the direction of airflow, air particles reach higher velocities in the upper surface of the profile when compared to the lower surface, thus generating a static pressure differential between the upper and lower surfaces and creating a lift force.

In the figures with $\alpha = 4^\circ$, the boundary layer was kept close to the S809 airfoil wall and tended not to separate from it, thus providing a favorable pressure gradient. For $\alpha = 15^\circ$, the boundary layer separated and vortices formed in the trailing edge, thus producing an adverse pressure gradient. The separation point moved toward the leading edge of the S809 airfoil and the separation bubble covered almost the entire surface of the airfoil. In such conditions stall sets in, lift is lost and aerodynamic drag increases.

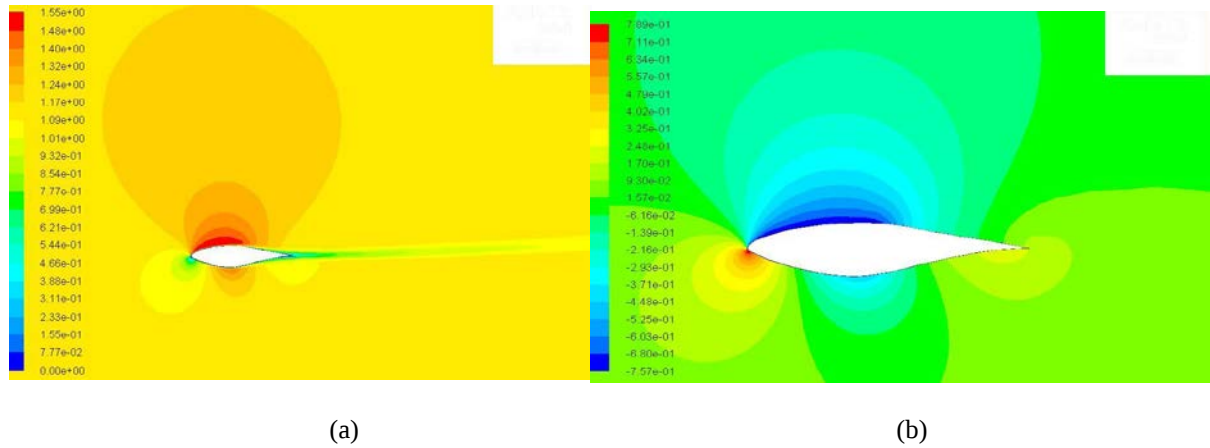


Figure 7. (a) Velocity (m/s) and (b) static pressure (Pa) contours for $\alpha = 4^\circ$; Reynolds number: 75,400; realizable $(\kappa-\epsilon)$ model

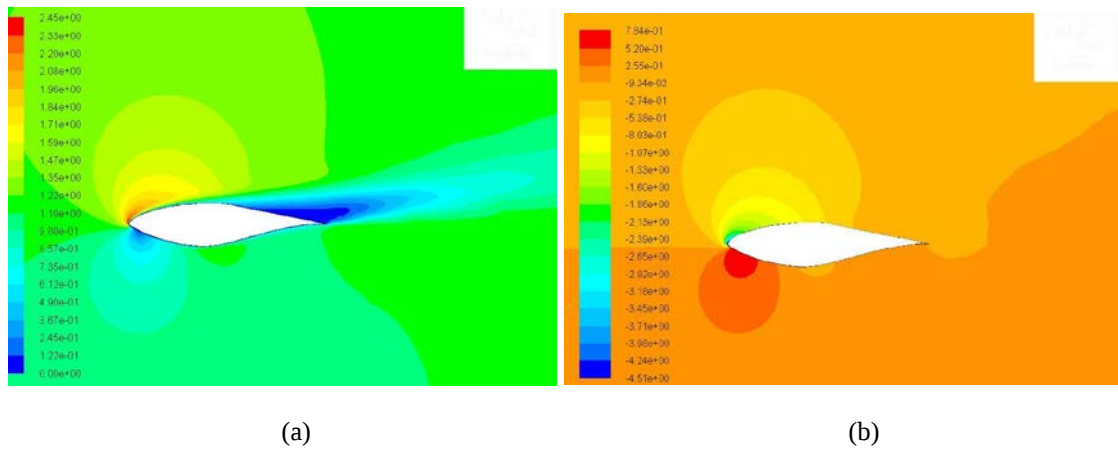


Figure 8. (a) Velocity (m/s) and (b) static pressure (Pa) contours for $\alpha = 15^\circ$, Reynolds number: 75,400; realizable $(\kappa-\epsilon)$ model

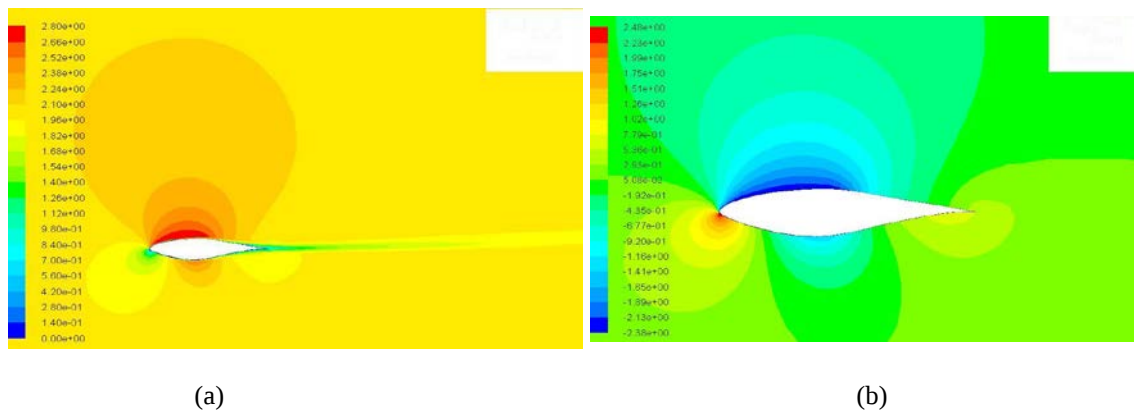


Figure 9. (a) Velocity (m/s) and (b) static pressure (Pa) contours for $\alpha = 4^\circ$, Reynolds number: 134,500; realizable $(\kappa-\epsilon)$ model

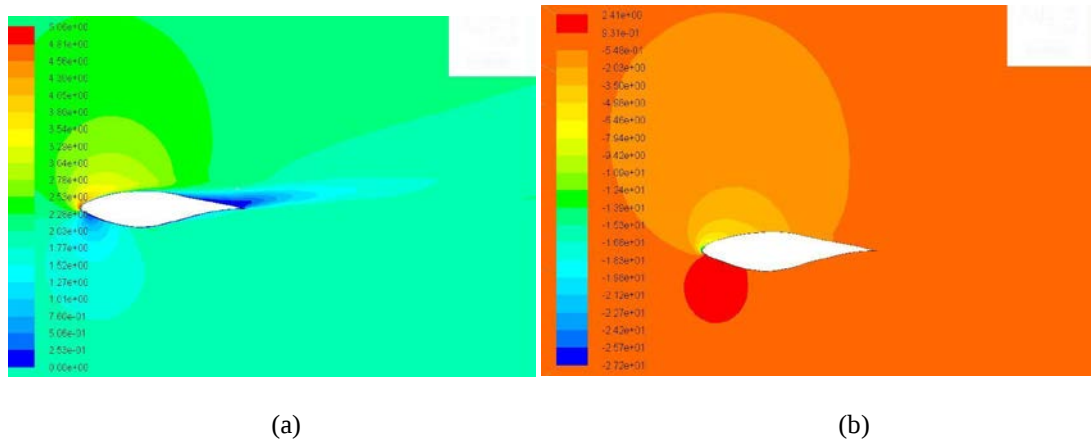


Figure 10. (a) Velocity (m/s) and (b) static pressure (Pa) contours for $\alpha = 15^\circ$, Reynolds number: 134,500; realizable (κ - ϵ) model

4. CONCLUSIONS

The aerodynamic characterization of airfoil profiles should include smaller angles of attack, particularly of 20 degrees. Aerodynamic performance may be significantly affected, as important drag and lift forces act on the airfoil in this angle of attack range. This was described by Zhou, *et al.* 2011.

The point of maximum efficiency of the S809 airfoil used in the blades of horizontal axis wind turbines was reached when $\alpha = 8^\circ$.

The S809 airfoil stalled when the angle of attack was set in the region of 15° to 16° , which is the angle formed between the fluid stream and the chord. Stall reduces the efficiency of wind turbine blades. At this angle of attack turbulence forms and flow separates, leading to the formation of whirling masses of fluid called vortices behind the rotor of the wind turbine. This phenomenon was observed for $Re = 75,400$ and $135,400$ and for $\alpha \approx 15^\circ$ - 16° .

The turbulence models used in this study – the realizable (κ - ϵ) and the standard κ -omega (κ - ω) models – yielded results in agreement with the literature. However, the standard κ -omega (κ - ω) model presented more efficient results near the wall, while the realizable (κ - ϵ) model produced more efficient results far from the wall.

Stall should be avoided in the design of horizontal axis wind turbines. Stalling wind turbines lose torque. However, in high winds this characteristic may be needed to control the velocity and output of a wind turbine. In this context, blades with good stall characteristics should be picked.

5. ACKNOWLEDGEMENTS

The authors would like to thank FAPEMIG for financial support provided.

6. REFERENCES

- Campos, J. C. Costa, Gomes, A.O., Brito, R.F., Rosa, H.M.P., Tibiriça, A.M.B. and Treto, Pedro C., 2014. "Experimental analysis of an S809 airfoil". *Thermal Engineering*, Vol. 13, p. 28-32.
- Cebeci, T., Shao, J.P., Kafyeke, F. and Laurendeau, E., 2005. *Computational Fluid Dynamics for Engineers*. Springer, Long Beach, California.
- Fadigas, E. and Amaral, A. F., 2011. *Energia Eólica*. Manole, São Paulo.
- FLUENT Database, 2011. ANSYS 14.0 Help User's guide.
- Fortuna, A. de O., 2000. *Técnicas Computacionais para Dinâmica dos Fluidos - Conceitos Básicos e Aplicações*. Edusp, S.P.
- Freire, A.P.S., 2002. *Coleção Cadernos de Turbulência*. Vol. 1. ABCM, Rio de Janeiro.
- Ghia, U., Ghia, K.N. and Shin, C.T., 1982, "High-Re solutions for incompressible flow using the Navier-Stokes equations and a multigrid method". *Journal of Computer Physical*, Vol. 48, p. 387-411.
- Gharali, K. and Johnson, D.A., 2012. "Numerical modeling of an S809 airfoil under dynamic stall, erosion and high reduced frequencies". *Applied Energy*, Vol. 93, p. 45-52.
- Huang J.-C, Lin H. and Yang, J.-Y, 2009, "Implicit preconditioned WENO scheme for steady viscous flow computation". *Journal of Computational Physics*, Vol. 228, p. 420-438.
- Logsdon, Nathan. 2006. A procedure for numerically analyzing airfoils and Wing sections. University of Missouri, Columbia.

- Moshfeghi, M., Song, Y.J. and Xie, Y.H., 2012, "Effects of near-wall grid spacing on SST-K- ω model using NREL Phase VI horizontal axis wind turbine". *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 107-108, p. 94-105.
- Neto, A.S., 2002. *Coleção Cadernos de Turbulência*. Vol. 1. ABCM, Rio de Janeiro.
- Troldborg, Niels., 2013. Experimental and numerical investigation of 3D aerofoil characteristics on a MW wind turbine, Roskilde, Denmark.
- Zhou, Y., Md., Mahbub, Alam, Yang, H. X., Guo, H. and Wood, D. H., 2011, "Fluid forces on a very low Reynolds number airfoil and their prediction". In *Proceedings of Schulich School of Engineering*, University of Calgary, Canada.

7. RESPONSIBILITY NOTICE

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