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

# COBEM-2017- 0306
# AN ASSESSMENT OF DIFFERENT TURBULENCE MODELS ON A CFD SIMULATION OF AIR FLOW PAST A S814 AIRFOIL

**Ciro Cordeiro de Araújo Bezerra**
Federal Univesity of Pernambuco, Department of Mechanical Engineering, Prof. Moraes Rêgo Av., 1235 - Cidade Universitária, Recife - PE - Brazil - Zip Code: 50670-901 - Phone: +55 81 2126-8000.
eduardocrfernandes@gmail.com

**Alex Maurício Araújo**
**Augusto Antônio Coutinho da Silva**
**Cristiano Leal dos Santos**
**Eduardo Corte Real Fernandes**
**Eduardo José Novaes Menezes**
**Guilherme José de Arruda Moura Rocha**
**Vitor Fernando de Souza Rodrigues**
**Wênio Fhará Alencar Borges**
Federal Univesity of Pernambuco, Department of Mechanical Engineering, Prof. Moraes Rêgo Av., 1235 - Cidade Universitária, Recife - PE - Brazil - Zip Code: 50670-901 - Phone: +55 81 2126-8000.
ama@ufpe.br, augustocoutinho@gmail.com, ciro.cordeiro1@gmail.com, sholeal@gmail.com, eduardo.novaesmenezes@ufpe.br, guilhermejamr@gmail.com, oymadouglas@gmail.com, wenioborges@hotmail.com

**José Ângelo Peixoto da Costa**
Federal Institute of Pernambuco, Prof. Luís Freire Ave., 500 - Cidade Universitária, Recife - PE - Brazil - Zip Code: 50740-540 - Phone: +55 81 2125-1600.
angelocosta@recife.ifpe.edu.br

***Abstract.*** *Turbulence modelling is a crucial part of any CFD simulation, the selection of an appropriate method allows for precise validation and reliable results. This paper aims to investigate which turbulence model is best suitable for a simulation over a S814 airfoil profile at different angles of attack using CFD software Ansys-Fluent. Literature usually recommends two models for this kind of simulation: $\kappa - \omega$ Shear Stress Transport and Spalart-Allmaras. Both model's strengths and weakness are tested performing simulations at the following angles of attack: 0º,5º,10º, 15º, 20º. Results were validated with experiments reported in literature and showed that Spalart-Allmaras performed better for lower angles of attack, whereas $\kappa - \omega$ Shear Stress Transport presented the best results for higher angles of attack.*

***Keywords:*** *CFD, Turbulence Modelling, Ansys, Spalart-Allmaras, $\kappa - \omega$ SST.*

## 1. INTRODUCTION

Computational Fluid Dynamics (CFD) simulations have been applied to a diversity of problems of different fields of engineering, including wind energy. These simulations can determine the aerodynamic behaviour of different types of turbines in order to select parameters to optimize its efficiency, making the use of CFD software a powerful tool to assist the design process of a real wind turbine (Hernández Gómez, 2014).

Generally, parameter variation is the main factor that increases the complexity of a real CFD modelling (Walshe, 2003). Boundary conditions, mesh size and turbulence model are some of these parameters. The present article deals with the latter, a crucial stage of a simulation of any turbulent flow using CFD software.

The chaotic and random nature of a turbulent flow excludes the possibility of a precise description of the movement of all fluid particles, which causes velocity to be decomposed in a mean value with positive and negative fluctuations (Versteeg and Malalasekera, 2007). Consequently, more equations are added to the problem, reinforcing the importance of CFD, because otherwise it would not be possible to solve the case.

Carrying on the studies of De Magalhães Melo *et al* (2013), which develops a method for a CFD simulation over a S814 airfoil profile demonstrating that good simulation practices should bring consistent results, the present paper aims to apply this method to investigate which turbulence model is best suitable for this simulation at different wind conditions.

A similar study was conducted by Eleni *et al* (2012) on a NACA 0012 profile, using the following turbulence models: Spalart-Allmaras, Realizable κ - ε and κ – ω Shear Stress Transport (SST), with the latter presenting the best results. Rocha *et al* (2014) performed a calibration study of the κ – ω SST turbulence model for small scale wind turbines simulating on both NACA 0012 and NACA 4412 profiles. In general, according to Wang and Xiongwei, (2016), κ – ω SST model is considered to be the most successful for both 2D airfoil and 3D blade CFD modelling, despite the representation of roughness in such model not being entirely consistent or straightforward (Franke *et al*, 2004).

On the other hand, the Spalart-Allmaras is also widely used on this type of simulation. Fernandes *et al* (2010) performed a study in a NACA 2410 airfoil comparing κ - ε, κ – ω SST and Spalart-Allmaras using the opensource CFD software Openfoam finding the latter to be the most successful. A CFD simulation of air flowing past a fixed 2D NACA 0015 airfoil using the Spalart–Allmaras turbulence model combined with a Navier–Stokes solver based on a Chorin projection method was performed by Nordanger *et al* (2015). Likewise, a study developed by Pellerin *et al* (2015) shows that Spalart–Allmaras leads to results that successfully capture the experimental lift and drag patterns reported in the literature for S1223 and E387 airfoils.

The present study will run simulations of a flow over a S814 airfoil profiles at different AoA using the κ – ω SST and Spalart-Allmaras to determine which one is best considering criteria such as results accuracy and computational cost. Experimental results as Jonkman (2014) and Janiszewska *et al* (1996) are used to validate the simulations.

## 2. METHODOLOGY

In order to determine which one is the best turbulence model, steady state CFD simulations of a flow over an S814 airfoil will be held with each model at different angles of attack and a Reynolds number of approximately $1.5\cdot10^6$. Geometry and mesh are created with the assist of ANSYS Workbench together with ANSYS Fluent to solve the case. Results will be compared to Jonkman (2014) and Janiszewska *et al* (1996) to analyse which model best describes the problem.

### 2.1 Computational Domain

Two types of domain are commonly used to this kind of problem. The O-Mesh domain, represented in Ribeiro *et al* (2011) consists of a circle surrounding the airfoil, while the C-mesh is a semicircle coupled with a rectangle. The latter allow for better representation of the airfoil wake (De Magalhães Melo *et al*, 2013) and therefore, was chosen.

The domain must be large enough, so that wall effects will not significantly influence the flow field (Liu *et al*, 2016). Due to this fact, the computational domain was dimensioned with approximately 19c x 40c in which “c” is the chord length (De Magalhães Melo *et al*, 2013). The domain is shown in figure 1.

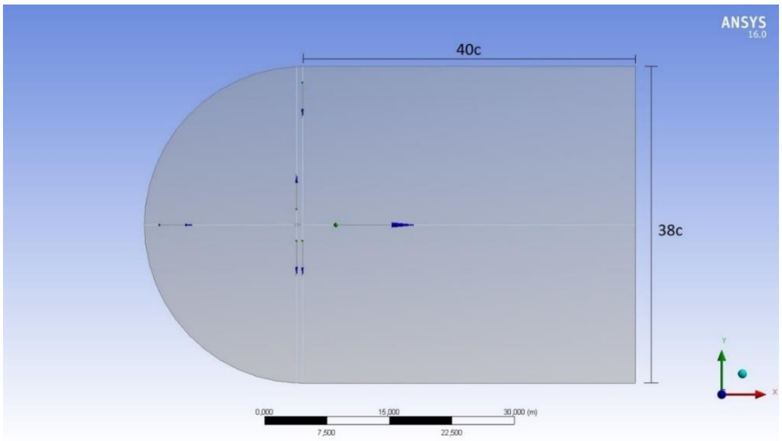


Figure 1: C-mesh domain size.

### 2.2 Boundary Conditions

Boundary conditions are shown in figure 2. The airfoil is set to be wall as if it was an obstacle to the flow. The upper and lower surfaces are also wall, but include the condition non-slip wall which imposes tangential velocity to be zero near wall (Çengel and Cimbala, 2015).

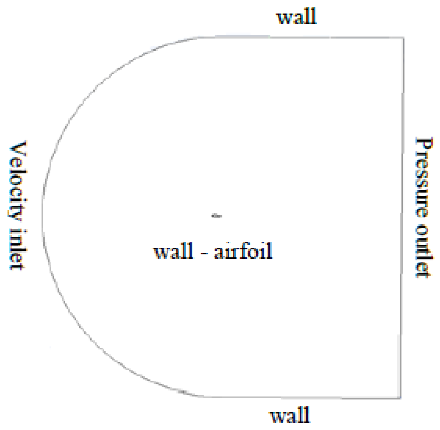


Figure 2: Boundary Conditions used for the software Ansys Fluent.

Inlet velocity was determined so that the Reynolds would be approximately $1.5 \cdot 10^6$. This value was chosen according to Janiszewska *et al* (1996). Using the chord length of 1m and a kinematic viscosity of $1.562 \cdot 10^{-5}$ m²/s for the air, a value of 23.43 m/s was obtained for velocity inlet. Furthermore, the outlet region was set to pressure outlet with gauge pressure equals zero

### 2.3 Wake Refinement

According to Wu and Porté-Agel (2012), turbulence models give two important characteristics of wind turbine wake: 1) the velocity deficit directly associated with the loss of power in large wind farms and 2) dynamic load on the turbine. Thus, a mesh refinement on the wake region is necessary in order to better analyse these factors, as shown in figure 3:

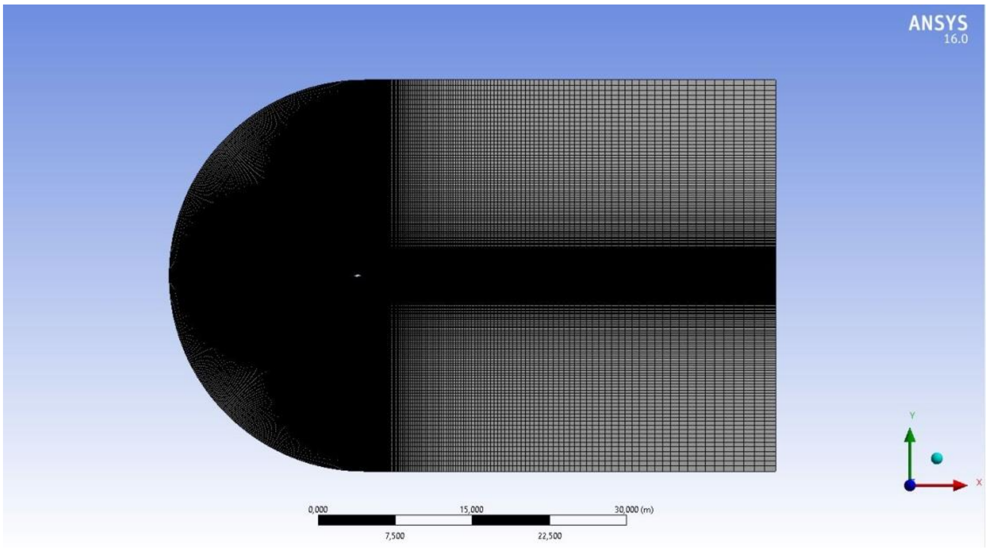


Figure 3: Refinement in the semicircle and, after the trailing edge, refinement until the end of the computational domain.

### 2.4 Mesh Creation

Another important refinement region is the airfoil itself. Refinement techniques were used around it as shown in figure 4, allowing for better visualization of the boundary layer detachment region and vortices formation. In this simulation, 540000 elements were obtained from 541800 nodes.

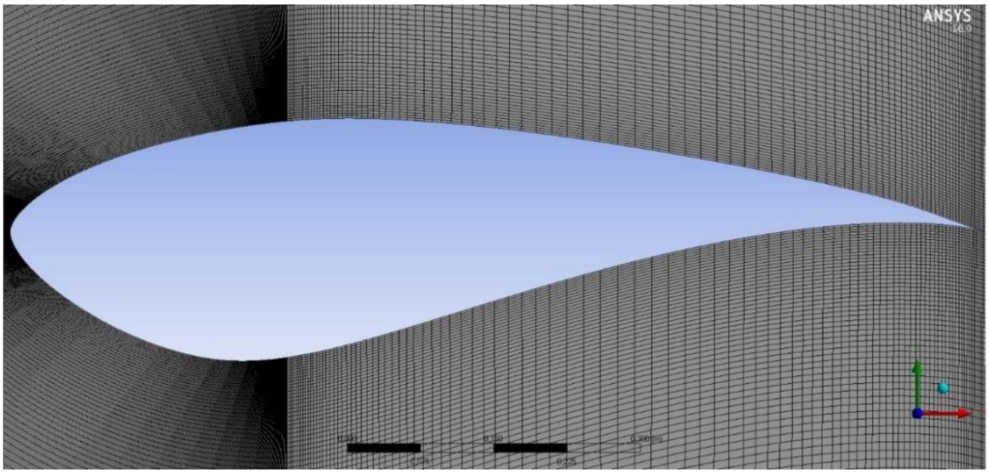


Figure 4: Refinement techniques near the S814 airfoil.

### 2.5 Turbulence Models

Current turbulence models can be grouped in three main categories, Reynolds-Averaged Navier-Stokes (RANS), Large Eddy Simulation (LES) and Direct Numerical Simulation (DNS). The last two, according to Versteeg and Malalasekera (2007) and Smyth (2016) are highly costly in terms of computing resources and will not be addressed in this paper. Both Spalart-Allmaras and $\kappa - \omega$ SST are classified as RANS models. In these models, velocity and pressure are broken into mean and fluctuating components, which are substituted into the original Navier-Stokes equation, reducing computational costs significantly (Smyth, 2016).

#### 2.5.1 Sparlat-Allmaras

The Spalart-Allmaras model is a one-equation model that solves a modelled transport equation for kinematic eddy (turbulent) viscosity parameter $\tilde{\nu}$, (ANSYS, 2013). This model was designed specifically for aerospace applications involving wall-bounded flows and has been shown to give good results for turbulence boundary layers subjected to adverse pressure gradients. The (dynamic) eddy viscosity is related to $\tilde{\nu}$ by:

$$\mu_t = \rho \tilde{\nu} f_{v1} \tag{1}$$

Equation 1 has a wall damping function, given by:

$$f_{v1} = \frac{\chi^3}{\chi^3 + C_{v1}{}^3} \tag{2}$$

Where:

$$\chi = \frac{\tilde{\nu}}{\nu} \tag{3}$$

According to Sparlat and Allmaras (1992), $\tilde{\nu}$ is equals $\nu$ except in the viscous region. Versteeg and Malalasekera, (2007) states that for high Reynolds numbers, $f_{v1}$ tends to one, whereas in the wall region, it tends to zero.

For the Fluent Solver, the input data chosen was the ratio of the turbulent viscosities $\left(\mu_t / \mu\right)$ which was set to be maintained as the standard value of 10, since it produced satisfying results.

#### 2.5.2 κ – ω SST

The Shear-Stress Transport (SST) model was developed by Menter *et al* (2003), to combine the precise and robust formulation of the κ – ω model in the near-wall region with the independence of the free flow of the κ - ε model in a more distant region. For this, a transition occurs from the κ – ω model, close to the wall, to the κ - ε model, at some distance. It uses a blending function to combine the models.

It was used as input data for the Fluent Solver calculated values for turbulent kinetic energy, κ, and for the parameter ω, that is the rate at which the turbulent kinetic energy is converted to thermal energy (Versteeg and Malalasekera, 2007).

The rate that the kinetic energy is converted into thermal energy ω is calculated by the following parameters: turbulent dissipation ε and turbulent kinetic energy κ. Thus:

$$\omega = \frac{\varepsilon}{\kappa} \tag{4}$$

The turbulent kinetic energy is calculated according to Eq. 5, whereas the turbulent dissipation is in accordance with Eq. 6.

$$\kappa = \frac{2}{3}\left(U_{ref} \cdot T_i\right)^2 \tag{5}$$

Where $U_{ref}$ is the reference velocity (velocity inlet) and $T_i$ is the turbulence intensity. In this article, $U_{ref}$ was 23.43 m/s and $T_i$ i was 5%. The value found for $\kappa$ was 0.9149 m²/s². For ε:

$$\varepsilon = \frac{\kappa^{3/2}}{l} \tag{6}$$

According to Versteeg and Malalasekera (2007), l is equivalent to 0.07L, where L is the characteristic length of the experiment, which, for the case of this study, is the chord length, which is 1 meter. Therefore, a value for ε of 125.0236 m²/s³ and ω of 136.6465 $s^{-1}$ were obtained.

### 2.6 Simulation

Final results with both models were obtained and compared to Jonkman (2014) and Janiszewska *et al* (1996), for the S814 airfoil. The most suitable for each situation was determined considering convergence, precision and computational cost criteria.

## 3. RESULTS AND DISCUSSION

Utilizing a chord length of 1 m, Reynolds number of approximately $1.5 \cdot 10^6$ and experimental data provided by Jonkman (2014) and Janiszewska *et al* (1996), the results for the S814 airfoil showed that the S-A turbulence model works best at lower AoA, up to 10°. However, κ – ω SST model performs better at higher AoA, except for 20º where S-A presented a lower error. Figure 5 shows the variation of the lift coefficient versus the angle of attack of the airfoil.

The best performance at lower values AoA can be explained by the fact that the S-A model solves the whole flow field for a refined mesh (ANSYS, 2013).

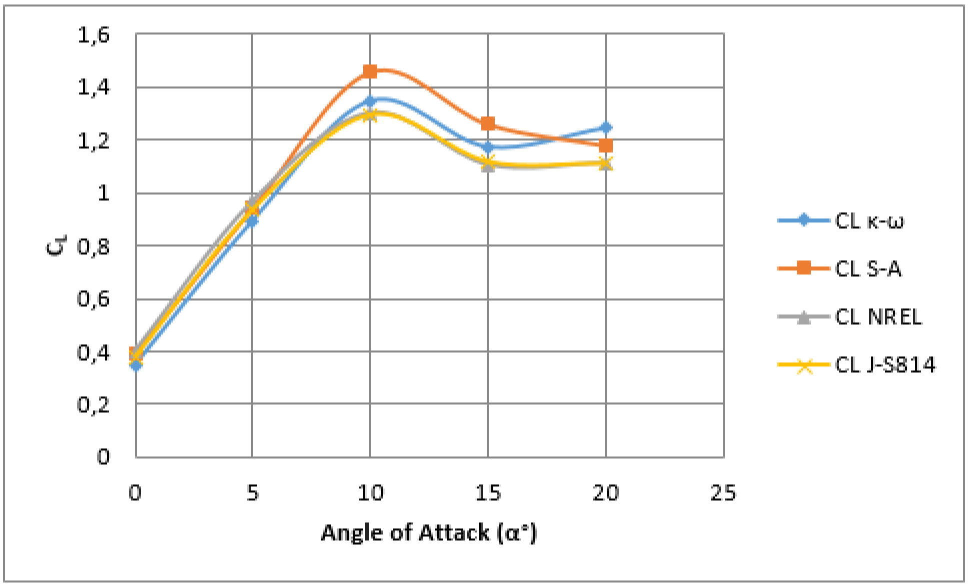


Figure 5: Lift coefficient versus the airfoil angle of attack in degrees.

At higher angles of attack, the airflow detaches faster from the airfoil, requiring a more refined model for turbulent regions such as the κ – ω SST model. This model combines the optimal performance of the κ – ω model for wall-limited and low-Reynolds numbers using wall functions, with the κ - ε model for the fully turbulent regions.

The results agree with Hao *et al* (2014) and Versteeg and Malalasekera (2007). For turbulent regions with a high density of vortices, S-A does not bring satisfying results. However, κ – ω SST performs well at these regions, and thus brings better results at higher AoA.

The error compared to Jonkman (2014) and Janiszewska *et al* (1996) is shown in figure 6. The highest error was observed (around 14%) when the lift coefficient calculated by the S-A model was compared to NREL data. These errors values do not represent a very large discrepancy and therefore the model was validated.

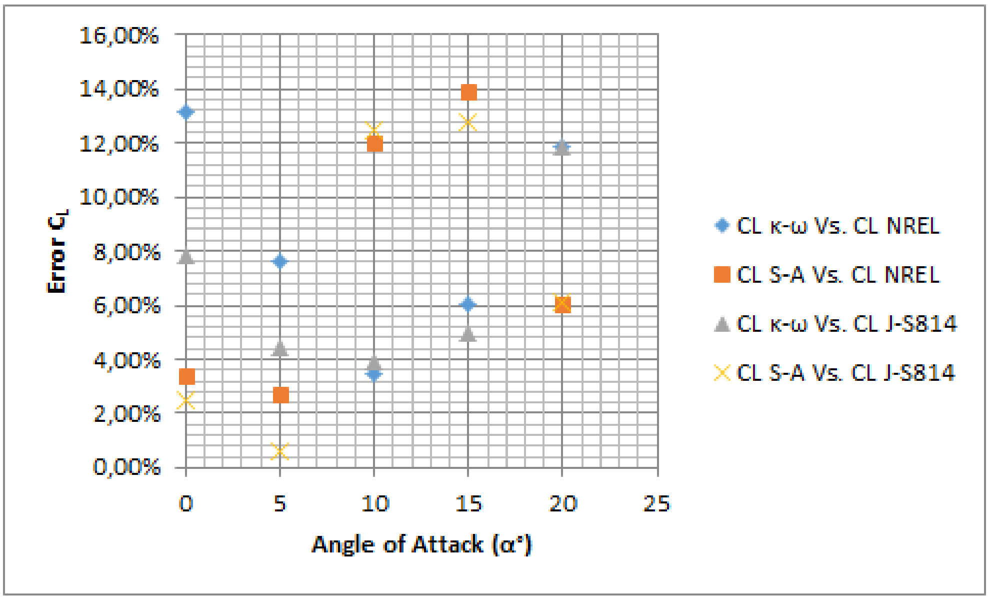


Figure 6: Relative errors of the lift coefficients for several models of turbulence versus angle of attack compared to Jonkman (2014) and Janiszewska *et al* (1996).

The drag coefficient was also studied and analysed. As expected from the work of Fernandes (2010), this coefficient can be super or underestimated at times. This is because RANS turbulence models assume that the whole flow is turbulent on the profile, which is not in accordance with reality, since there is usually a significant extension over the airfoil where the regime can be laminar or transition, leading to errors.

In addition, Bordin (2014) and Anderson Junior (2015) state that all real airfoils of finite extension possess less lift and drag than data from their airfoil sections indicate. The pressure difference between upper and lower blade surfaces causes a flow to occur at its extremity, generating a vortices trail, which decreases pressure difference and, consequently, the lift force. The effective angle of attack decreases with the downward velocities induced by vortices, causing lift to have a component in the flow direction, kwon as induced drag.

According to Wilcox (2006), based on the approach of Boussinesq, there is an increase of turbulent friction on the profile, even in zones where the regime is laminar. Therefore, it can be concluded from this theory that there is an error for the drag coefficient inherent to any model of turbulence chosen. Nevertheless, the present work showed significantly lower errors when compared to that of Fernandes (2010).

Figures 7 and 8 show the behaviour of the drag coefficient for various angles of attack and the respective errors compared to the data of Jonkman (2014) and Janiszewska *et al* (1996).

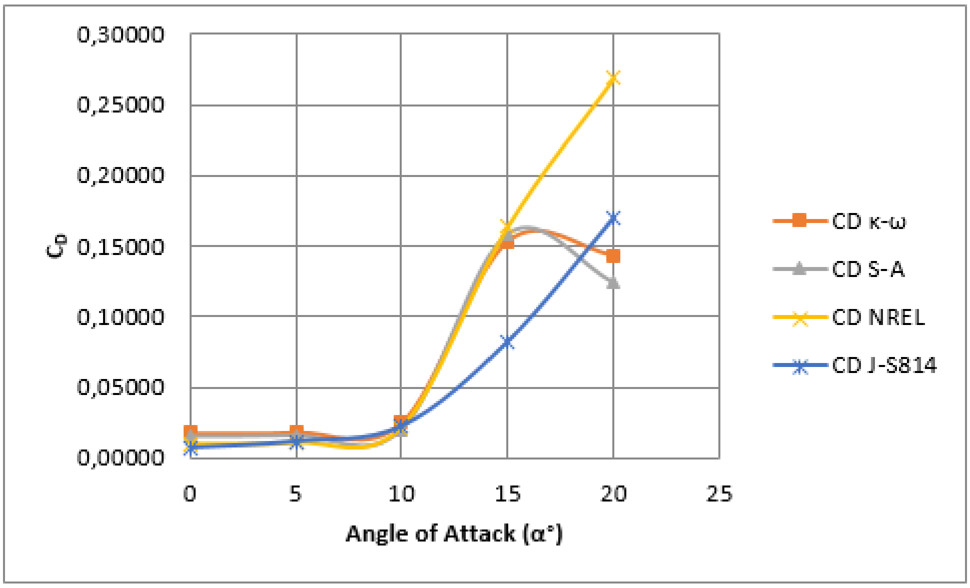


Figure 7: Drag coefficient versus the airfoil angle of attack in degrees.

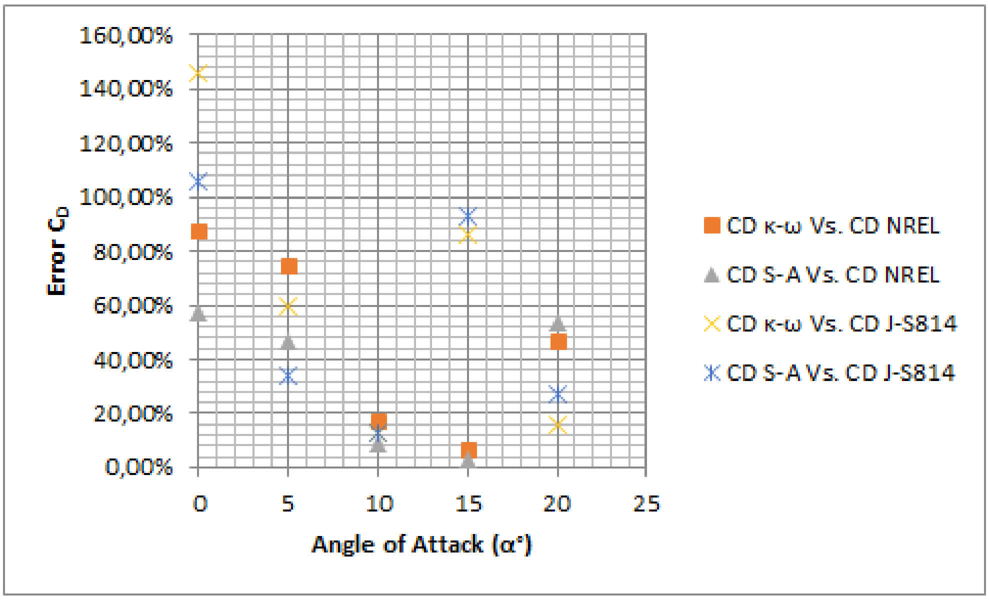


Figure 8: Relative errors of the drag coefficients for several models of turbulence versus angle of attack compared to Jonkman (2014) and Janiszewska *et al* (1996).

## 4. CONCLUSION

According to the presented simulations on steady state, with each angle of attack simulated separately, it was observed that for lower AoA the S-A presents a lower error of lift coefficient compared to the experimental data; however, as AoA increases, this model's accuracy decreases. In contrast, for higher AoA κ – ω SST presents best results when compared to Jonkman (2014) and Janiszewska *et al* (1996). As for the drag coefficient, the error is intrinsic to the turbulence model because there are regions where the flow can be laminar or transition, leading the model to failure.

Thus, when pursuing good engineering practices, and in a certain way computational economy, in steady state simulations of a flow over a S814 airfoil S-A or κ – ω SST can be used depending on the AoA.

## 5. ACKNOWLEDGEMENTS

The authors would like to acknowledge the Graduate Program in Mechanical Engineering at the Federal University of Pernambuco (PPGEM/UFPE) and the Ministry of Education of Brazil through CAPES and CNPq for supporting this work. We thank Ansys for providing the Ansys Fluent student version, which have been used in this study. In addition, we are grateful to the help of all colleagues involved in this work.

## 6. REFERENCES

Anderson Jr, John D. *Fundamentos de Engenharia Aeronáutica*. AMGH Editora, 2015.

ANSYS® Academic Research, Release 15.0, Help System, "*Ansys Fluent Theory Guide***"**, 2013.

Bordin, Franciele Stail. "Análise do efeito da interação fluido-estrutura nas forças fluidodinâmicas em um elemento de pá flexível 3D." (2014).

Cengel, Yunus A., and John M. Cimbala. Mecânica dos fluidos-3. AMGH Editora, 2015.

De Magalhães Melo, G. G..; Oliveira, L. A.; Araújo, A. M.; Asibor, A. I.; De Medeiros, A. L. R.; De Oliveira Filho, O. D. Q.; Espíndola, R. L.; Machado, H. C. M.; Jaouen, P.; Noguerra, R. E. "Application of good CFD simulation practices to s814 airfoil profile". In: International Congress of Mechanical Engineering, 22, 2013, Ribeirão Preto. Anais… Ribeirão Preto, 2013, p 3980 – 3989.

Eleni, Douvi C., Tsavalos I. Athanasios, and Margaris P. Dionissios. "Evaluation of the turbulence models for the simulation of the flow over a National Advisory Committee for Aeronautics (NACA) 0012 airfoil." Journal of Mechanical Engineering Research4.3 (2012): 100-111.

Fernandes, M. P. G.; Rocha, P. A. C.; Carneiro, F. O. M. "Avaliação de resultados de simulação numérica de escoamento sobre o perfil NACA 2410 utilizando o openfoam com diferentes modelos de turbulência". In: Congresso Nacional de Engenharia Mecânica, 2010, Campina Grande. Anais… Campina Grande, 2010, 9p.

Franke, J.; Hirsch, C.; Jensen, A. G.; Krüs, H. W.; Schatzmann, M.; Westbury, P. S.; Wisse, J. A.; Wright, N. G. "Recommendations on the use of CFD in wind engineering." Cost action C. Vol. 14. 2004.

Hao. Y.; Liu, Yangwei.; Le, F.; Lipeng, L. "Modification of Spalart-Allmaras turbulence model for predicting S825 airfoil aerodynamic performance." Applied Mechanics and Materials. Vol. 543. Trans Tech Publications, 2014.

Hernández Gómez, Antonio. "Computational Fluid Dynamics study of 2D vertical axis turbines for application to wind and tidal energy production". BS thesis. Universitat Politècnica de Catalunya, 2014.

Janiszewska, J. M.; Ramsay, R. R.; Hoffmann, M. J.; Gregorek, G. M. 1996. Effects of grit roughness and pitch oscillations on the S814 airfoil. No. NREL/TP--442-8161. National Renewable Energy Lab., Golden, CO (United States).

Jonkman, J., and Marshall L. Buhl Jr. "NWTC information portal (FAST)." *2015-05-15]. https://nwtc. nrel. gov/FAST* (2014).

Liu, Z.; Takeshi, I.; He, X.; Niu, H. "Les study on the turbulent flow fields over complex terrain covered by vegetation canopy." Journal of Wind Engineering and Industrial Aerodynamics 155 (2016): 60-73.

Menter, F. R., M. Kuntz, and R. Langtry. "Ten years of industrial experience with the SST turbulence model." *Turbulence, heat and mass transfer* 4.1 (2003): 625-632.

Nordanger, Knut.; Holdahl, R.; Kvamsdal, T.; Kvarving, A. M.; Rasheed, A. "Simulation of airflow past a 2D NACA0015 airfoil using an isogeometric incompressible Navier–Stokes solver with the Spalart–Allmaras turbulence model." Computer Methods in Applied Mechanics and Engineering 290.

Pellerin, N.; Leclaire, S; Reggio, M. "An implementation of the Spalart–Allmaras turbulence model in a multi-domain lattice Boltzmann method for solving turbulent airfoil flows." Computers & Mathematics with Applications 70.12 (2015): 3001-3018.

Ribeiro, A.F.P., Awruch A.M., Gomes, H.M. 2011. "An airfoil optimization technique for wind turbines." Applied Mathematical Modelling 36.10 (2012): 4898-4907.

Rocha, PA Costa.; Rocha, H. H. B.; Carneiro, F. O. M.; Bueno, A. V. "k–ω SST (shear stress transport) turbulence model calibration: A case study on a small scale horizontal axis wind turbine." Energy 65 (2014): 412-418.

Smyth, Thomas AG. "A review of Computational Fluid Dynamics (CFD) airflow modelling over aeolian landforms." *Aeolian Research* 22 (2016): 153-164.

Spalart, P. and Allmaras, S. A. (1992). "A one equation turbulence model for aerodynamic flows." RECHERCHE AEROSPATIALE-FRENCH EDITION- (1994): 5-5.

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

Walshe, John D. CFD modelling of wind flow over complex and rough terrain. Diss. © John Walshe, 2003.

Wang, Lin.; Liu, Xiongwei.; Kolios, Athanasios. "State of the art in the aeroelasticity of wind turbine blades: Aeroelastic modelling." Renewable and Sustainable Energy Reviews 64 (2016): 195-210.

Wilcox, D. C. Turbulence Modelling for CFD. Third Edition. La Canãda, California: DCW Industry, 2006. 522 p.

Wu, Y. T.; Porté-Agel, F. (2012). "Atmospheric turbulence effects on wind-turbine wakes: An LES study." *Energies* 5.12 (2012): 5340-5362.

## 7. RESPONSIBILITY NOTICE

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