24th COBEM - 2017

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

COBEM-2017-1470
METHOD OF VOLUMES AND FINITE DISCRETIZATION STRATEGIES APPLIED THE AERODYNAMIC PROFILES

Miriam Lys de Almeida Gemaque
Lucas Vinicius Fernandes Coelho
Universidade Federal do Pará, Rua Augusto Correa, 01. Belém, PA. Brasil
lysgmak@hotmail.com
lucasvfc@gmail.com
Marilza dos Santos Viana
Universidade Federal do Pará, Rua Augusto Correa, 01. Belém, PA. Brasil
marilza.viana@ymail.com

Erb Ferreira Lins
Universidade Federal do Pará, Rua Augusto Correa, 01. Belém, PA. Brasil
erb@ufpa.br
Abstract. *The analysis of flow around airfoil is of great importance in various sectors of practical interest, for example, tha aviation industry, where appropriate projects are directly related to the concepts of efficiency and safety. This study aims to develop a methodology for proper discretization and analysis of flow in airfoils, using the software package ANSYS Inc. For this analysis, drag and lift coefficients are compared to results of experimental data (Selig et al., 1995) of Eppler 423, Selig 1223 and NACA 2414 profiles for high Reynolds number. The turbulence models used in the numerical study are standard κ-ε model with an enhanced wall treatment and the κ-ω model SST also in order to evaluate the effectiveness of each model in calculating a turbulent flow. Also used if the turbulence model Reynolds Stress model followed by Sparlat Allmaras model and finally the presentation of the results obtained by comparing the models adopted.*

Keywords:*Computational Fluid Dynamics, Aerodynamic Profiles, Turbulence.*

1. INTRODUCTION

The results of Computational Fluid Dynamics are directly analogous to the wind tunnel results obtained in the laboratory - both represent datasets for flows with different Mach numbers, Reynolds numbers etc. However, unlike a wind tunnel, which is usually a heavy, tricky device, a computer program is something you can use with relative ease (Anderson, 2012). In this work, a discretization strategy is developed for the simulation of aerodynamic profiles, using FLUENT software, which is part of the ANSYS software package. The mesh structure is constructed in MATLAB software, with the aim of improving the quality of the mesh on the surface in which the profiles have a more pronounced curvature, its leading edge. The turbulence models used for such analysis are the κ-ε Standard and κ-ω SST models, for the profiles Eppler 423, Selig 1223 and Naca 2414 and for validation of the results, the data are compared with the results obtained in the literature (Selig Et al., 1995).

2. FUNDAMENTALS

2.1 Aerodynamic Parameters Drag and lift

The flow of fluids on rigid bodies often occurs in practice and is responsible for numerous physical phenomena such as drag and lift force, these parameters are of great importance for determining the efficiency of an aerodynamic profile. The drag force is the component that acts parallel to the direction of relative motion of the fluid (fox, 2006). The force of lift is the component that will act parallel to the action of the movement relative to the flow, as it is represented in Figure 1:

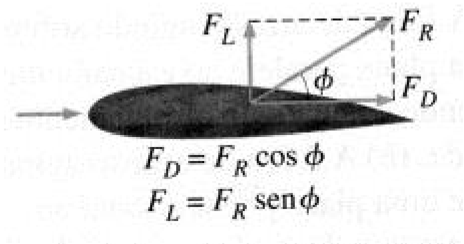


Figure 1. Action of drag forces and lift in an aerodynamic profile; Çengel, 2007

The drag (F_D) and lift (F_L) forces depend on several factors as data concerning the work fluid and the body, however it is more usual for reasons of simplifying the study to work with dimensionless values as the coefficients of the respective forces, thus we have:

$$C_D = \frac{F_D}{\frac{1}{2}\rho v^2 A} \tag{1}$$

$$C_L = \frac{F_L}{\frac{1}{2}\rho v^2 A} \tag{2}$$

2.2 Equations by Navier Stokes

The Navier-Stokes equations are partial differential equations that describe the flow of fluids; are used in order to reduce the difficulties inherent in the numerical solution.The equations that govern the flow of the fluid represent the mathematical statements of the Physical Conservation Laws:
• The mass of the fluid is preserved;
• The rate of change of momentum is equal to the sum of the forces on a particle of the fluid (Second Law of Newton);
• The rate of change of energy is equal to the sum of the added heat rate and the rate of work performed on a particle of the fluid (First Law of Thermodynamics).

3. Turbulence models and wall law

Turbulence is a manifestation of the chaotic space and time behavior presented by fluid flows with high values of Reynolds numbers, ie, it is a dissipative system with an extremely large number of degrees of freedom (most likely) described by equations of Navier-Stokes (MANDUS et al. 2010).

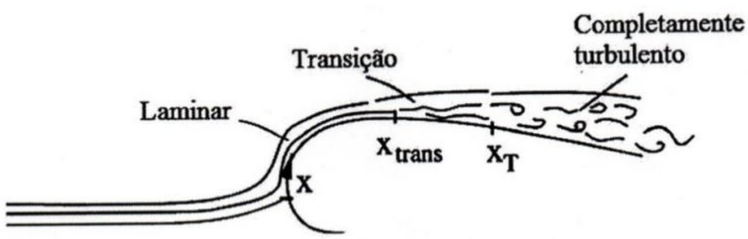


Figure 2. Schematic view of the flow in an aerodynamic profile. HANSEN, 2008

As for wall laws, they represent an important relation in the boundary layer theory with which the velocity and temperature fields are normally calculated for regions close to the wall in addition to other quantities. By modeling the turbulent quantities especially in the innermost region of the boundary layer, in the so-called viscous sublayer and transition region where the viscous dissipation rate takes order of magnitude equal to or greater than the turbulent dissipation. The smaller the loss of information regarding the simplifications of these more appropriate quantities will be this law (ALMEIDA, 2014). The solution to the near-wall region was first introduced by Taylor (1916), and later completed by Prandtl (1925) and Von Karman (1930) (apud Cruz and Freire 2002), where the average velocity profile from distance from the wall is given by:

$$\frac{u}{u_\tau} = \frac{1}{0.41} \ln\left(\frac{u_\tau \gamma}{\upsilon}\right) \tag{3}$$

Eq. (3) represents classical wall law, which determines that the mean velocity of a point of the flow is proportional to the logarithm of the distance from this point to the wall. Another way of expressing the Law of the classical wall is through the dimensional analysis of Buckingham's π theorem, where substitution of the original variables pertinent to the problem of interest by dimensionless groups occurs, thus reducing the number of variables.

3.1 Turbulence model RANS (Reynolds averaged Navier-Stokes)

This model offers the most economical approach for the resolution of turbulent flows for industrial simulations. Typical examples of these models are the κ-ε or the κ-ω models in their different forms (WILCOX, 2010). In the RANS turbulence models attention is focused on the mean flow and the effects of turbulence on the mean flow properties. The solution of two additional transport equations is simplified by these models besides introducing a turbulent viscosity to compute the Reynolds tensors. The basic tool for the derivation of the equations of the RANS model is the Reynolds decomposition.

3.2 Standard Model

The standard κ-ε model is the simplest turbulence model in which only the boundary conditions have to be provided, as well as being the best-established model and delivering excellent performance for industrially relevant flows. However, according to Shinomiya (2013), the limitation of some κ-ε models is their instability for adverse pressure gradients and boundary layer separation. They usually anticipate a delayed and reduced separation from those obtained experimentally. This can result in overly optimistic project evaluations for flows that separate from smooth surfaces (airfoils, diffusers, etc.). Therefore, the κ-ε model is not so used in aerodynamic external flows. This model is based on the transport equations for turbulent kinetic energy (κ) and its dissipation rate (ε), the standard κ-ε model is semi-empirical. In it, the turbulent kinetic energy and its dissipation rate are obtained, respectively, by the transport equation for turbulent kinetic energy κ and by the equation for the viscous dissipation rate ε. The standard κ-ε model is only valid for fully turbulent flows.

3.3 Model SST (shear-stress transport)

The κ-ω SST (shear-stress transport) model was developed by Menter to effectively combine the robust and faithful formulation of the κ-ω model in the near-free-wall region with the independence of the κ-ε model in the distant field. To achieve this goal the κ-ε model is converted into the κ-ω model formulation (ANSYS Inc., 2013). The κ-ω SST model is more accurate and shows greater confidence for a wider class of flows (eg, adverse pressure gradient flow, airfoils) than the standard κ-ε model (Veersteg and Malalasekera, 2007).

3.4 Reynolds Stress Turbulence Model

This model tries to remedy some deficiencies of the RANS model, in addition to attenuating some of the limitations of the Boussinesq hypothesis. It is a way of solving Navier-Stokes average equations by means of the direct acquisition of equations referring to Reynolds transport or Reynolds stress transport.

3.5 Sparlat Almaras Turbulence Model

This model is usual in space applications, where flows are restricted by walls, gradients and pressure variations. In it the differential equations are found through dimensional analysis and through data referring to molecular viscosity

4. NUMERICAL AND COMPUTATIONAL ASPECTS

4.1 Generation of Geometry

For the generation of the geometry of the profile, a subroutine generated in the MATLAB program was used, whose objective is to generate, its geometry, as well as the geometry of the mesh and save it to a text file. The mesh geometry adopted has the objective of better discretizing the elements close to the profile, mainly on surfaces where there is greater deformation of the profile. With the generated text file, in the ANSYS WORKBENCH 16.2 software, several procedures were executed in the FLUENT program.

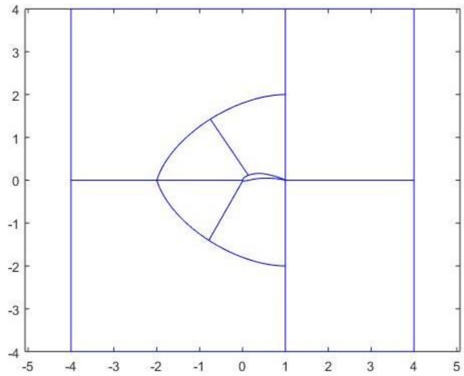


Figure 3. Geometry model

4.2 Mesh Generation and Fluent Setup

Two types of mesh were generated, one for each model of turbulence, however for both meshes the method selected was automatic, the mapping was done on the faces involved in the profile and on the faces behind the profile, while on both faces in front of the profile the "Face Sizing" option was used, with elements of size 0.005. For the model κ-ε Standard, the condition considered for the validation of the mesh was the analysis of the $y^+ \approx 1$ near the profile wall, with that was generated a mesh with 93746 elements using "bias" for the refinement of the elements nearby profile. In the denser

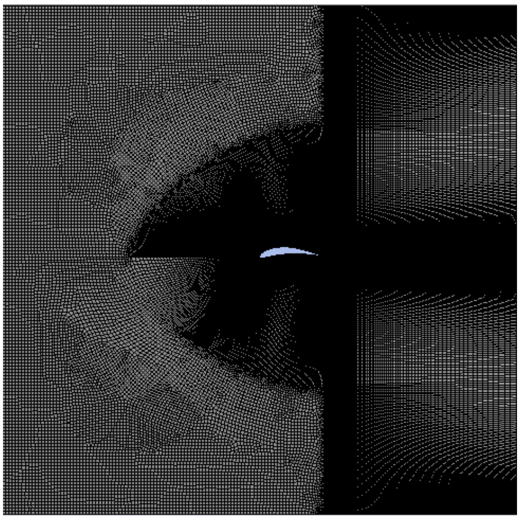


Figure 4. Mesh for the E423 profile.

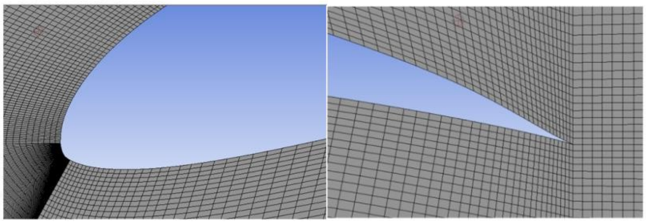


Figure 5. Mesh close to the profile.

region of the mesh, on the leading edge (Figure 5) a bias was created so that the elements did not undergo a very large deformation in relation to their size and angle, which would impair the simulation.

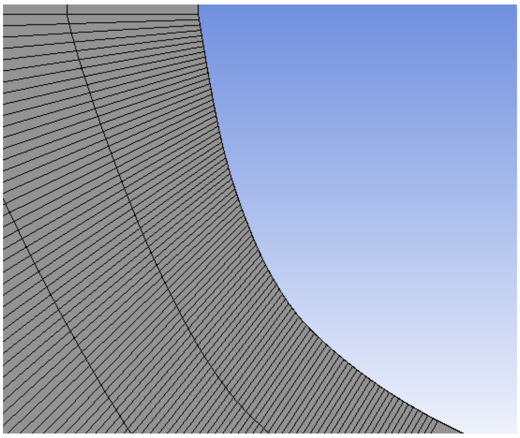


Figure 6. Mesh in the denser region on the leading edge.

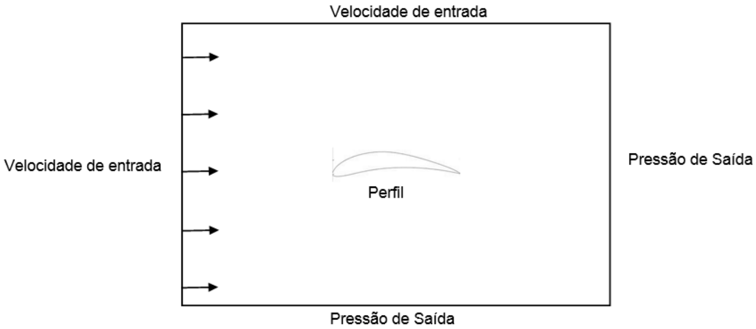


Figure 7. Contour condition for the profile.

In order to check the validity of the mesh, the FLUENT Setup was performed, where the standard κ-εturbulence model was selected with the Enhanced Wall Treatment. The contour conditions were defined according to Fig. 7. For the input velocity, as there is the change of angle in the vector, we chose to use the velocity components in FLUENT itself. 2000 interactions were performed, and after completion the y^+ was close to the profile. To validate the mesh we used the "Adapt" feature, which adapts the elements near the wall to the selected range of y^+ . In the present work the interval $0 < y^+ < 1$.was adopted. The procedure must be repeated until the interval is reached. The criterion used for the validation of the mesh for the κ-ω SST model was the comparison with the experimental data (SELIG et al., 1995), for the forces acting on drag. 6 meshes were created as shown in the following table. The mesh selected was mesh 4 because it presented a smaller error for the coefficient of support (Cl), although it did not present the slightest error for the drag coefficient (Cd). For the FLUENT Setup, the κ-ω SST model was used, with the "low Reynolds Curvature" option activated, the same contour conditions were used as the standard κ-ε model, and the same number of interactions.

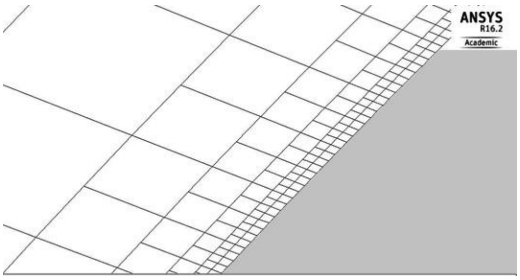


Figure 8. Mesh close to the profile with the "Adapt".

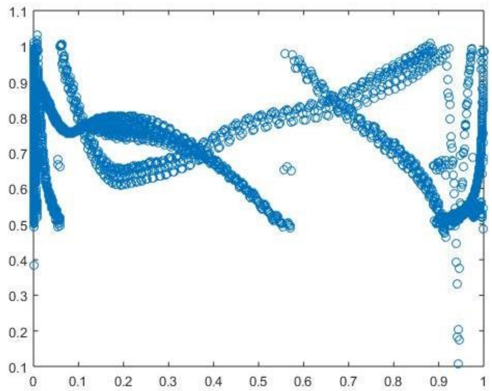


Figure 9. Graph for mesh validation.

Table 1. Comparison between the proposed meshes for the κ-ω SST model.

E423 (α=0.20)	Number of Cells	Cl	Error Cl (%)	Cd	Error Cd(%)
Experimental		0,852		0,0364	
Mesh 1	33035	0,62528	26,6103	0,056402	54,9505
Mesh 2	42146	0,75088	11,8685	0,041129	12,9918
Mesh 3	52873	0,82099	3,6397	0,04374	20,1648
Mesh 4	65230	0,8234	3,3568	0,0431	18,4066
Mesh 5	79889	0,81565	4,2664	0,042418	16,5330
Mesh 6	95348	0,8089	5,0587	0,057585	58,2005

5. RESULTS AND DISCUSSIONS

Using the ANSYS FLUENT software, the flow flows were obtained on the profiles Eppler 423, Selig 1223 and Naca 2414 which can be visualized in "Figure 10". The simulations were performed at three different angles of attack for each profile, obtaining the drag and lift coefficients, besides the graphs, contour for pressure, velocity and turbulent kinetic energy for each profile. The experimental data were obtained from (Selig et al.,1995). Three different Reynolds numbers were used, one for each profile, according to the experimental data.

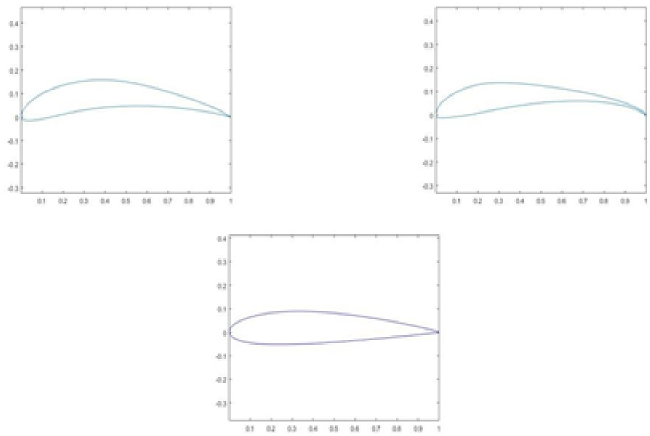


Figure 10. Aerodynamic profiles used in the simulations. COELHO, 2016.

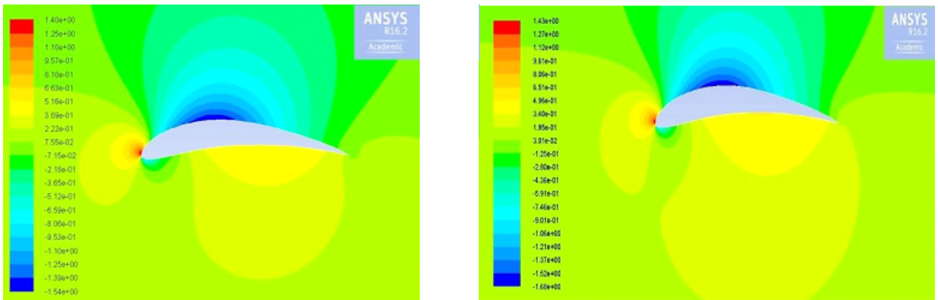
5.1 Result for EPPLER 423 profile

For the Eppler 423 profile, a Reynolds of 100800 was used for the angles of $0.2°$, $4.82°$ and $10.66°$. The results and errors in relation to the experimental data are described in the following table.

Table 2. lift and drag coefficients and errors for profile E423 and Re = 100800.

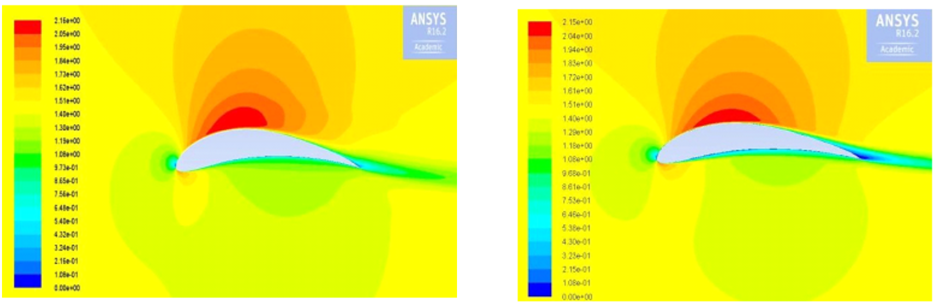
E423 (Re=100800)			
α (?)	0,2	4,82	10,66
Coefficient of lift (Cl)			
Selig et al.	0,852	1,1239	0,981
κ-? Standard	0,9273	1,1056	1,3186
Error (%)	8,8380	1,6283	34,4139
κ-ω SST	0,08234	1,1446	1,5293
Error (%)	3,3568	1,8418	55,8919
Coefficient of drag (Cd)			
Selig et al.	0,0364	0,045	0,1992
κ-? Standard	0,04966	0,09589	0,1607
Error (%)	36,4286	113,0889	19,3273
κ-ω SST	0,0431	0,06285	0,142852
Error (%)	18,4066	39,6667	28,2871

Analyzing the table it is observed that for the angles of $0.2°$ and $4.82°$ the κ-ω SST model presents a smaller or very close error, relative to the experimental data, to the standard κ-ε model, while for $(10.66°)$ the standard κ-ε model was more accurate. The contour charts for pressure, velocity and turbulent kinetic energy for each model are presented below.



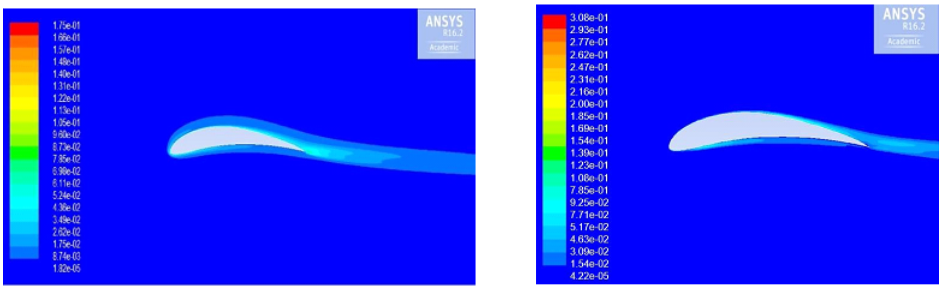
(a) Model κ-ε Standard (b) Model κ-ω SST

Figure 11. Pressure profile contours E423 for the angle of attack of $0.2°$ in Pascal.



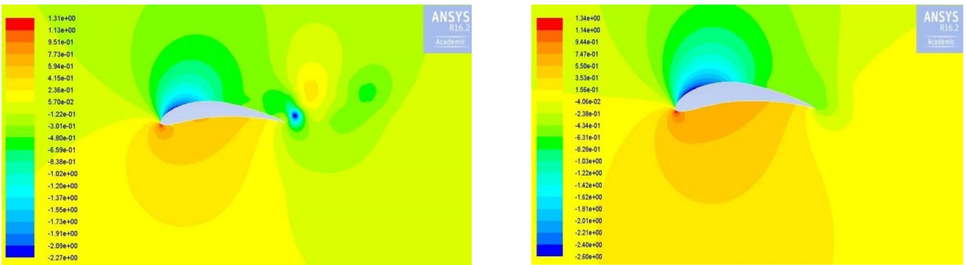
(a) Model κ-ε Standard (b) Model κ-ω SST

Figure 12. Profile contour E423 for the angle of attack of $0.2°$ in m / s.



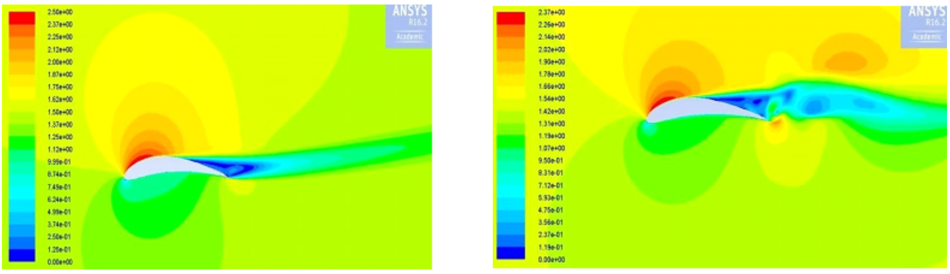
(a) Model κ-ε Standard (b) Model κ-ω SST

Figure 13. Contour of turbulent kinetic energy of profile E423 for the angle of attack of $0.2°$ in m^2/s^2.



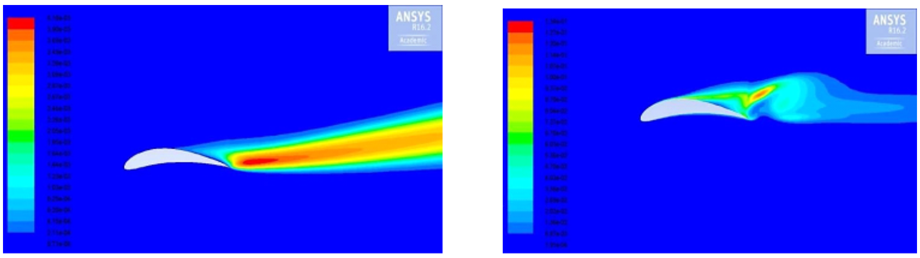
(a) Model Sparlat Allmaras (b) Model Reynolds Stress

Figure 14. Profile pressure contour E423.



(a) Model Sparlat Allmaras (b) Model Reynolds Stress

Figure 15. Profile speed contour E423.



(a) Model Sparlat Allmaras (b) Model Reynolds Stress

Figure 16. Turbulent kinetic energy contour of profile E423.

Analyzing the contours of Figures 11, 12, 14 and 15 for the four models of turbulence, one can observe one of the physical principles of the sustentation, where the increase of the velocity in the superior surface of the profile causes a pressure drop in the same, and the low speed on the lower surface generates an increase in pressure. This pressure difference is responsible for the lift. Figures 13 and 16 show that mainly at the trailing edge of the profile, the turbulent kinetic energy is higher.

5.2 Results for SELIG profile 1223

For the Selig profile 1223, a Reynolds of 102100 was used for the angles of $0.89°$, $4.82°$ and $6.68°$. The results and errors in relation to the experimental data are described in table 3. he results obtained for the Selig profile 1223 show that only for the angle of attack of $0.89°$ the turbulence model κ-ω SST was more accurate than the standard κ-ε model for the

Table 3. Support and drag coefficients and errors for profile S1223 and Re = 102100.

S1223 (Re=102100)			
α (°)	0,89	4,82	6,68
Coefficient of lift (Cl)			
Selig et al.	1,12	1,423	1,536
κ-ϵ Standard	1,2561	1,4559	1,6004
Erro (%)	12,1518	2,3120	4,1927
κ-ω SST	1,0108	1,30623	1,43934
Erro (%)	9,7500	8,2059	6,2930
Coefficient of drag (Cd)			
Selig et al.	0,0311	0,0475	0,0548
κ-ϵ Standard	0,04826	0,07352	0,08358
Erro (%)	55,1768	54,7789	52,5182
κ-ω SST	0,0459	0,071033	0,07778
Erro (%)	47,5884	49,5432	41,9343

coefficient of sustentation, in compensation in the prediction of the Despite the large errors found, the κ-ω SST model was more efficient, however, for the drag coefficient, the prediction given by the ANSYS FLUENT software is very unstable, so it is a coefficient that is not taken into account for profile analysis.

5.3 Results for NACA 2414 profile

For the Naca 2414 profile, a Reynolds of 100800 was used for the angles of $1.31°$; $5.87°$ and $10.45°$. The results and errors with respect to the experimental data are described in tab 4. Table 4 shows that for the Naca profile 2414, in

Table 4. lift and drag coefficients and errors for profile N2414 and Re = 100800.

N2414 (Re=100800)			
α (°)	1,31	5,87	10,45
Coefficient of lift (Cl)			
Selig et al.	0,362	0,718	1,049
κ-ϵ Standard	0,26735	0,6431	1,005
Erro (%)	26,1464	10,4318	4,1945
κ-ω SST	0,2213	0,54265	0,8207
Erro (%)	38,8674	24,4220	21,7636
Coefficient of drag (Cd)			
Selig et al.	0,021	0,0207	0,0333
κ-ϵ Standard	0,02494	0,03767	0,06149
Erro (%)	18,7619	81,9807	84,6547
κ-ω SST	0,02354	0,0378	0,05558
Erro (%)	12,0952	82,6087	66,9069

the prediction of the coefficient of lift, the standard κ-ε turbulence model is more accurate for all the analyzed angles of attack, however, since it is a symmetrical profile, a more accurate response of the software was expected.

6. CONCLUSION

The methodology adopted in the present article was mesh development, where the domain discretization was sought in a structured way, using the standard κ-ε turbulence models with the improved wall treatment, and κ-ω SST, for the analysis of the lift and drag coefficients in aerodynamic profiles. As each model requires its own criterion for the convergence of the mesh, two meshes were created, varying the number of elements and refinement near the boundary layer.The most reliable results were expected to be from the κ-ω SST model over the standard κ-ε, but for higher attack angles, the standard κ-ε model presented a higher accuracy for the coefficient of lift. The high errors presented for the drag coefficient for both models are due to the instability of the software in regard to their calculation. It is interesting to note that for the symmetric profile Naca 2414, errors were higher for both models. Although they do not accurately represent the behavior of velocity and pressure, the contours demonstrate aerodynamic principles since for pressure to lift, the pressure at the bottom of the profile must be greater than at the top and the point of stagnation where the speed is zero in the profile.

7. REFERENCES

ALMEIDA, V.F, 2014. Implementation of a Fluid Dynamic Wall Law Applicable to Flow with Adverse Pressure Gradient. Federal University of Pará, Institute of Technology, PPGEM.

ANDERSON, J.D, 2012. Computational Fluid Dynamics – The basics with Applications. McGrawhill.

ANSYS. 2013. Ansys Fluent Theory Guide 15.0. Canonsburg, PA.: Ansys Inc.

ÇENGEL, Y. A.; CIMBALA, J. M, 2007. Fluid Mechanics - Fundamentals and Applications. 2. ed., McGrawhill.

COELHO, L.V. F, 2016. Finite Volumes Method and Discretization Strategies Applied to Aerodynamic Profiles. Federal University of Pará, Institute of Technology, Faculty of Mechanical Engineering.

CRUZ, D.O.; FREIRE, A. P. S, 2002. Equation of motion and asymptotic results applied to boundary layer theory. In Book of the Third Spring School Transition and Turbulence, volume 1, pages 49-97. Book of the Third Spring School Transition and Turbulence.

FOX, R.W. McDONALD, A.T., 2006 Introduction to Fluid Mechanics 6 ed. LTC.

HENSEN, M. 2008. Aerodynamics of winds turbines. 2.ed., Earthscan.

HIRSCH, C. 2007Numerical Computation of Internal and External Flows – Fundamentals of Computational Fluid Dynamics. 2.ed., Butterworth-Heinemann.

MANSUR, S. S., VIEIRA, E. D. R., NETO, A. S, 2002. Turbulence. 1 ed., Vol. 7, Rio de Janeiro, ABMC, 2010.

SELIG, M.S.; GUIGLIELMO, J.J.; BROEREN, A.P.; GUIGUERE, P. 1995. Summary of low-speed Airfoil data.Volume 1 e 2,Virginia: SoarTech Publications.

SHINOMIYA, D. L. , 2013Numerical study of the drag and drag coefficients of aerodynamic profiles applied to the horizontal axis wind turbine design. Federal University of Pará, Institute of Technology, Faculty of Mechanical Engineering.

TANNEHILL, J.C., ANDERSON, D.A., PLETCHER, R.H. 1997.Computational Fluid Mechanics and Heat Transfer. 2.ed., Tayloy Francis,.

VERSTEEG, H.K., MALALASEKERA, W. 2007. An Introduction to Computational Fluid Dynamics – The Finite Volume Method. 2. ed., Pearson,.

WILCOX, D. C. 2010. Turbulence modeling for CFD, DCW Industries, 3.ed.

8. RESPONSIBILITY NOTICE

The author(s) is (are) the only responsible for the printed material included in this paper.