UNIVERSIDADE FEDERAL DA GRANDE DOURADOS
Faculdade de engenharia
Curso de ENGENHARIA MECÂNICA - FAEN

ALEXANDRE DE ALMEIDA ALVES

EPPLER 387 AIRFOIL LIFT AND DRAG AERODYNAMIC FORCES ANALYSIS BY A BIDIMENSIONAL NUMERICAL MODEL IN OPENFOAM

Dourados - MS
2021

ALEXANDRE DE ALMEIDA ALVES

EPPLER 387 AIRFOIL LIFT AND DRAG AERODYNAMIC FORCES ANILYSIS BY A BIDIMENSIONAL NUMERICAL MODEL IN OPENFOAM

Trabalho de Conclusão de Curso apresentado à Banca Examinadora da Universidade Federal da Grande Dourados, como pré-requisito para obtenção do título de Bacharel em Engenharia Mecânica, sob a orientação do Prof. Dr. Reginaldo Ribeiro de Sousa.

Área de concentração: 3.05.01.02-4 Mecânica dos Fluidos

Dourados - MS
2021

MINISTÉRIO DA EDUCAÇÃO
FUNDAÇÃO UNIVERSIDADE FEDERAL DA GRANDE DOURADOS

ANEXO D - AVALIAÇÃO FINAL DO TRABALHO DE CONCLUSÃO DE CURSO

Aluno: **ALEXANDRE DE ALMEIDA ALVES**

Título do trabalho: **EPPLER 387 AIRFOIL LIFT AND DRAG AERODYNAMIC FORCES ANALYSIS BY A BIDIMENSIONAL NUMERICAL MODEL IN OPENFOAM.**

BANCA EXAMINADORA

1. **Presidente (orientador):**
Prof. Dr. Reginaldo Ribeiro de Sousa, Universidade Federal da Grande Dourados - UFGD

2. **Membro:**
Prof. Dr. Rafael Ferreira Gregolin, Universidade Federal da Grande Dourados – UFGD

3. **Membro:**
Prof. Dr. Rodrigo Borges dos Santos, Universidade Federal da Grande Dourados - UFGD

De acordo com o grau final obtido pelo aluno, nós da banca examinadora, declaramos **Aprovado** o aluno acima identificado, na componente curricular Trabalho de Conclusão de Curso (TCC-II) de Graduação no Curso de Engenharia Mecânica da Universidade Federal da Grande Dourados.

Dourados, 30 de dezembro de 2021.

____
Prof. Dr. Reginaldo Ribeiro de Sousa

____
Prof. Dr. Rafael Ferreira Gregolin

Rodrigo Borges Santos
____
Prof. Dr. Rodrigo Borges dos Santos

RESUMO

Este trabalho tem como objetivo, avaliar as caraterísticas aerodinâmicas de um perfil de asa do tipo Eppler 387. Um modelo bidimensional, de regime permanente, incompressível, isotérmico e turbulento, foi simulado numericamente através do software OpenFOAM 7.0. Os modelos turbulentos Spallart Allmaras e K-omega SST foram testados. As forças aerodinâmicas obtidas dos ângulos de -2 até 14 graus, tiveram boa concordância com autores renomados na literatura. O modelo K-omega SST forneceu melhores resultados com base nos dados experimentais da NASA, em comparação com o modelo Spallart Allmaras. O modelo Spallart Allmaras superestimou a força de sustentação e subestimou as forças de arrasto para ângulos acima de 8 graus.

Palavras-chave: Eppler 387, k omega SST, Spalart Allmaras, OpenFOAM

ABSTRACT

This work aims to evaluate the aerodynamic behavior of the Eppler 387 airfoil. A bidimensional, steady state, incompressible, isothermal and turbulent model was numerically simulated using OpenFOAM 7.0 software. Spallart Allmaras and K-omega SST turbulence models were tested. The resulting aerodynamic forces, for angles of attack from -2 to 14 degrees, were compared with available data in the literature with good agreement. K-omega SST model provided better agreement with NASA experimental results whereas Spallart Allmaras overestimated the lift force and underestimated the drag force for the angle of attack greater than 8 degrees.

Keywords: Eppler 387, k omega SST, Spalart Allmaras, OpenFOAM

COBEM 2021 26th International Congress of Mechanical Engineering

ABCM Associação Brasileira de Engenharia e Ciências Mecânicas
November 22-26, 2021, Virtual Congress, Brazil

COB-2021
EPPLER 387 AIRFOIL LIFT AND DRAG AERODYNAMIC FORCES ANALYSIS BY A BIDIMENSIONAL NUMERICAL MODEL IN OPENFOAM

Alexandre de Almeida Alves
Reginaldo Ribeiro de Souza
Universidade Federal da Grande Dourados, Dourados - MS

alexandrextz124@hotmail.com
reginaldosouza@ufgd.edu.br

Abstract.*This work aims to evaluate the aerodynamic behavior of the Eppler 387 airfoil. A bidimensional, steady state, incompressible, isothermal and turbulent model was numerically simulated using OpenFOAM 7.0 software. Spallart Allmaras and K-omega SST turbulence models were tested. The resulting aerodynamic forces, for angles of attack from -2 to 14 degrees, were compared with available data in the literature with good agreement. K-omega SST model provided better agreement with NASA experimental results whereas Spallart Allmaras overestimated the lift force and underestimated the drag force for the angle of attack greater than 8 degrees.*

Keywords: *Eppler 387, k omega SST, Spalart Allmaras, OpenFOAM*

1. INTRODUCTION

Aerodynamics is a vast and varied field present in our everyday life. This field, in which air is the working fluid, includes studies of buildings and bridges stability (Cheng *et al.* (2002)), polluent dispersion (Bottema (1997)), medical devices for assisted breathing (Zachow *et al.* (2006)), and it is also found in sports, like the design of a new golf ball (Kato *et al.* (2001)). However, the aerodynamics as a specific science field, became to gain deep importance only with the emerging need of decreasing transportation facilities' drag. The need of creating airfoils which would allow upper velocities and less air friction, led to a shift in the studies of aerodynamics Anderson (2001). The airfoils are intended to reduce drag and increase lift force. They are also responsable for the structure stability which leads to the specific field of aeroelasticity.

During the last few years, unmanned aerial vehicles (UAVs) and micro-air vehicles (MAV) have became important in both civilian and military applications. Due these vehicles' low-speed working, with reduced sizes or in rarefied atmospheres, as in the case of UAVs designed to Mars exploration, the use of low Reynolds number profiles became extremely important to aeronautics field.

However, vast majority of studies conducted in aeronautics are based on medium-to-high Reynolds number flow regimes, which has a range between 4e5 to 1e8, that creates the need for aerodynamic studies in low Reynolds number airfoils (bellow Re=4e5) (Martínez-Aranda *et al.* (2016)).

The Eppler airfoil type was used as a study base for the current work, and it is usually employed in low Reynolds Numbers (around Re=2e5) applications. Following on Fig 1., the graphic representation obtained through Paraview software 5.8.0, of Eppler 387 profile.

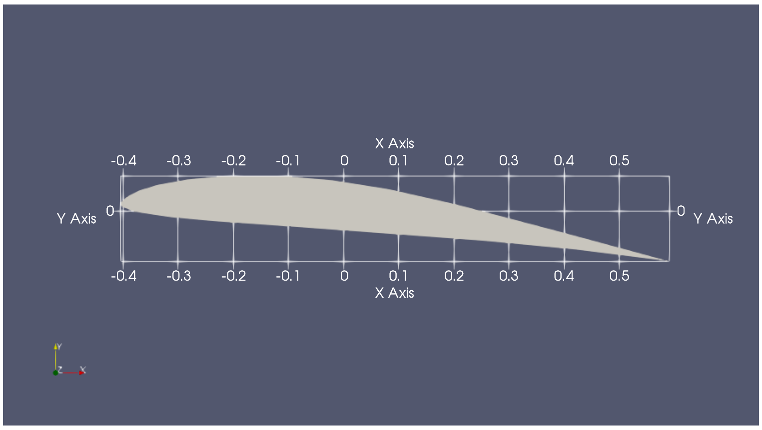

Figure 1. Eppler 387 airfoil profile at 6 degrees of angle of attack. Reference: The author.

The Reynolds number, is defined in Eq. (1).

$$Re = \frac{VL}{\nu} \tag{1}$$

As showned, small wing profiles (short L), reduced velocities (small V), and low density atmospheres (high ν, where $\nu = \frac{\mu}{\rho}$), generates a low Re. Thereby, the commerce becomes to need profiles that deals better with this request and accomplishes its designed functions succesfully.
Where:

V is the free stream velocity $[m/s]$

ν is the kinematic viscosity $[m^2/s]$

L is the chord length of the airfoil [m]

Much has been analyzed in this subject, using both numerical and experimental methods. Numerical simulations using CFD (Computational Fluid Dynamics) have the advantage of delivering results quickly. In order to obtain accurate results, however, one should know how to select the adequate numerical model, especially when boundary layer transition is present.

Yet, before the computational processing application in engineering, the aerodynamic' studies were made by empirical tests on its fullness. This kind of study requires a high finantial and human investment to be accomplished, increasing, this way, the cost and time for developing the project.

With computational processing use, some, but not all, experimental study design steps were replaced. This led not only to cost and final time design reduction, but also to engineering design optimization. Moreover, it provided technological development in this field (Research (2020)). Nevertheless, the computational simulation requires a detailed study of boundary conditions and parameters used to ensure that the results obtained agrees to phisycal reallity that encompasses all the phenomenon (Fernandes (2010)).

Due to the fact that most of external flow in engineering are turbulent, this kind of computational model gained a wide development in late years. However, the turbulence is characterized by being unpredictable, tridimensional and having severals energy scales, what makes harder its modeling. Although, there are methods that solve all turbulence scales accurately, named Direct Numerical Simulation, or DNS Schumann (1974), Coleman and Sandberg (2010), but their high computational cost inhibits their use in industrial applications.

Even so, in great majority of industrial cases, the engineer's interest is focused in averaged values (for head loss, heat transfer rate, drag, lift, etc.) obtained quickly (limited by the computational power available). Thus, Reynolds Averaged Navier-Stokes, or RANS, methods become a suitable choice and are widely used in different engineering fields, where relatively fast and low-cost predictions are required (Brown *et al.* (2018)).

1.1 Objectives

The objective of this paper is to develop a computational model that makes the aerodynamic' study of a Eppler type wing profile, precisely the Eppler 387, represented on fig. 2 previously.

There will be analysed the lifting and drag profile coefficients. It will be subject to a flow with the Reynolds number of 2.0e5 due its application. Will be presented the computational studies of aerospatial engineering department of UIUC (Univesity of Illinois in Urbana and Champaign) and the Xfoil model performed by airfoiltools site. The experimental validation was accomplished by Robert J. McGhee, at NASA's Langley reserch center. The studies quoted above were compiled in order to co-create a substancial bibliographical base to this model.

2. MODELLING AND SIMULATION

The Reynolds-Averaged Navier Stokes (RANS) approach solves the Reynolds equations to determine the mean velocity field U (Menter *et al.* (2003)). Reynolds stresses can be calculated from a turbulent-viscosity model, which can be obtained from an algebraic relation (e.g. mixing-length model) or it can be from turbulence quantities such as k and ω for which modelled transport equations are solved Pope (2004).
Which is:

k is the turbulence kinetic energy $[m^2/s^2]$

ω is the rate of specific dissipation of turbulent kinetic energy $[m^2/s^3]$

In order to solve the RANS equations, it is common to discretize them using the Finite Volume Method (FVM). In this method, flow variables are calculated at each boundary of each control volume by surface integrals. As a result, FVM is called conservative.

The computational modeling software in (FMV) utilized was the OpenFOAM, due being a free computational program about fluid mechanics. It possess an easy access and modification code, what facilitates the physical and numerical understanding regarding the issue. Other advantage is that it can be easily formulated for uses in unstructured meshes (Maliska (2004)). Also accomplishes the requested task with major eficience and robustness.

2.1 Geometry

The first step to create the numerical model is to create airfoil geometry. From airfoiltools Tools (2020), the Eppler 387 airfoil geometric points were saved in a dat file. FreeCAD software (version 0.18.4) recognizes the dat file as a profile by just opening the dat file. The number of points is usually not enough to create a smooth surface, notaceably at the airfoil leading edge. Then, in the draft module, it is recomended to apply the spline function to the imported geometry before extruding it. After extrusion, the final geometry is exported as a *.stl* file.

Additional surface refinement was made using the surfaceRefineRedGreen function from OpenFOAM 7.0.

The airfoil center of mass (CoM) can be identified by the surfaceInertia function. The geometry was translated so that its CoM matches the domains origin at (0, 0, 0), in a cartesian coordinate system. The CoM information is used in the controlDict function, responsable for calculating the aerodynamic forces, so matching it with the coordinate system origin is not necessarily relevant for that purpose. It is relevant when using the functionality that rotates the airfoil for the different angles of attack.

2.2 Mesh domain

The fluid flow numerical domain was firstly defined in the blockMeshDict. Different domain sizes were evaluated in order to eliminate the domain's size influence. The airfoil geometry had to be rotated, at the origin, from -2 to 14 degrees, for every 2 degrees. Thus, a bash script was written to automate the procedure described next.

For each angle of attack simulated, a case directory was built. In each case, two subcases were defined; the first for the snappyHexMesh application and the second for the simpleFoam application. The former is intended to create the mesh for each angle of attack and the later, to the incompressible, steady state simpleFoam solver.

Except the first angle of attack (zero), all the following cases had their airfoil geometry rotated, from the original case, using the surfaceTransformPoints OpenFOAM funcionality. The surfaces geometric information were extracted using the surfaceFeatures OpenFOAM functionality. This is a relative new dictionary in the system case folder. The representation of an airfoil and its characteristics are below on Fig. 2.

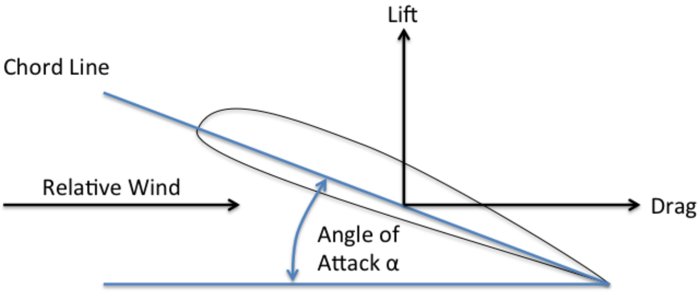

Figure 2. Attack angle, chord length, lift and drag of an airfoil. Reference: http://www.aviationchief.com/.

The background mesh, for each angle of attack, is constructed using the blockMesh application. The element size, at level 0 refinement, was 0.41875 [m]. The tridimensional mesher, the snappyHexMesh, is then used to take into account the airfoil profile. The refinement level around the airfoil was set to level 7, then, the average element size in that region was 0.0032714 [m]. Four layers with 1.5 compression rate were added from a 0.0009 [m] first layer in order to promote y+ less than 1.

The snappyHexMesh generated an intermediate mesh containing 3.5 M cells. This is a very fine mesh that do not requires further refinement. Since it is tridimensional, most of its cells are used to represent the region close to the airfoil surface. To minimize the computaional cost, the bidimensional mesh was derived from the tridimensional using the extrudeMesh funcionality, at the simpleFoam case. This procedure is detailed described in the Allrun file from the OpenFOAM wingMotion tutorial (Foundation (2020)). The resulting bidimensional mesh had 31,000 cells, ilustrated in Fig. 3.

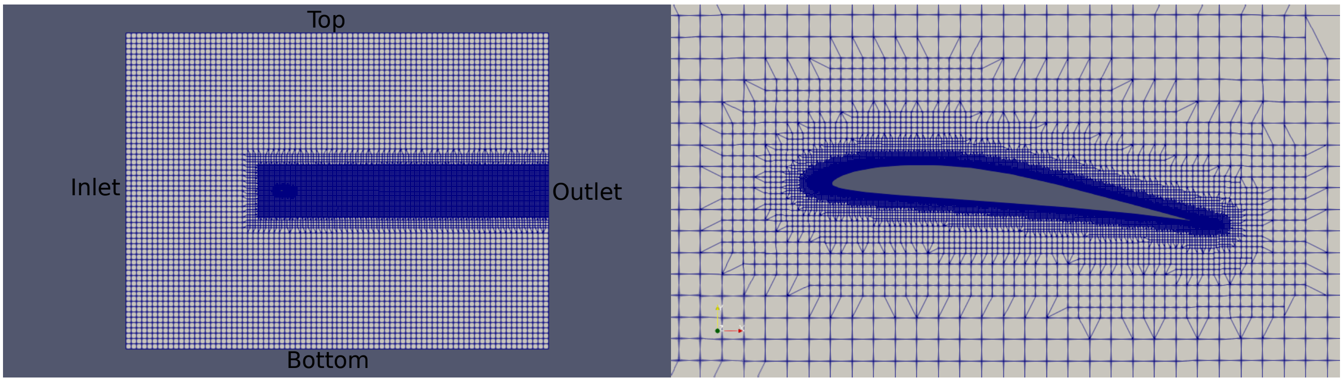

Figure 3. Domain discretization. Reference: The authors.

2.3 Solver

The fluid's physical phenomenom was assumed as a steady state and incompressible, with a turbulent flow. These simplifications are widely utilized in industry, with minimum of error.

The simplified physical phenomenom governing equations are the Reynolds-Averaged-Navier–Stokes equations (RANS), as meant previously. They form a group with approaches based in the turbulent fluid properties knowledge, simplifying, thereby, the Navier-Stokes equations solutions. The results are based in averaged time of flow assumed as newtonian fluid (Menter *et al.* (2003)).

OpenFOAM contains a variety of RANS models to predict turbulent flows. These are called turbulence models and each of them is more appropriate to a certain type of flow.

A turbulence model is defined as a set of equations (algebraic or differential) which determines the turbulent transport terms in the mean flow equations and thus close the system of equations. Turbulence models are based on hypotheses about the turbulent processes and requires empirical input in the form of model constants or functions; they do not simulate the details of the turbulent motion, but only the effect of turbulence on the mean flow behavior. The concept of Reynolds averaging and the averaged conservation equations are some of the main concepts that form the basis of turbulence modeling (Celik (1999)). Follows two of the models used in this work.

2.3.1 SST k-omega

Regards a semi-empirical turbulent model created by Menter (1992), which attaches two other computational fluid mechanics classicals. One of them, k-omega, is utilized to the boundary layer internal regions because it connects turbulent viscosity, cinetic energy and frequency, improving, therefore, the flow behaviour prediction. In addition, it does not involves complex nonlinear damping equations. However, it shows strong sensibility to free stream condition variations, what makes the model deficiet. The k-epsilon model is used to this kind of problem correction on turbulent regions (free shear layer). The SST k-omega contains a great behaviour for adverse pression gradient and boundary layer detachment regions.

2.3.2 Spalart-AllMaras

This is a model of only one transport equation, which determines the turbulence viscosity. Initially developed for modeling flow restricted to walls and with time, it also started presenting good results to boundary layers subjected to adverse pressure gradients (Spalart (1992)).

2.4 SIMPLE Algorithm

For RANS solutions, it was ultilized a well-known algorithm in specialized bibliography, called Semi-Implicit Method for Pressure Linked Equations (SIMPLE). This method is used for solving Navier-Stokes equations with simplifications mentioned above (Maliska (2004)). Also, its code is provided by OpenFOAM.

For angles of attack above 10 degrees, convergence was more difficult and required about 50 thousand iterations, while normally, 5 thousand were enough for an attack angle of zero.

The applied boundary conditions, present in the computational domain as illustrated in Fig. 3, were imposed inlet velocity of 3.01 [m/s] with turbulent intensity of 0.11%, inlet at the outlet, slip boundary condition at the top and bottom surfaces of the domain and zero wall velocity at the airfoil surface. The initial internal flow velocity considered was the inlet velocity. This setup simulates: Re = 2e5.

The advective terms were discretized using the bounded Gauss linearUpwind. The SIMPLE algorithm is employed for the pressure-velocity coupling. Relaxation factor of 0.5 was set. For the sake of simplicity, stop criterion was 10^{-7} for pressure and 10^{-8} for velocity fields.

3. RESULTS AND DISCUSSIONS

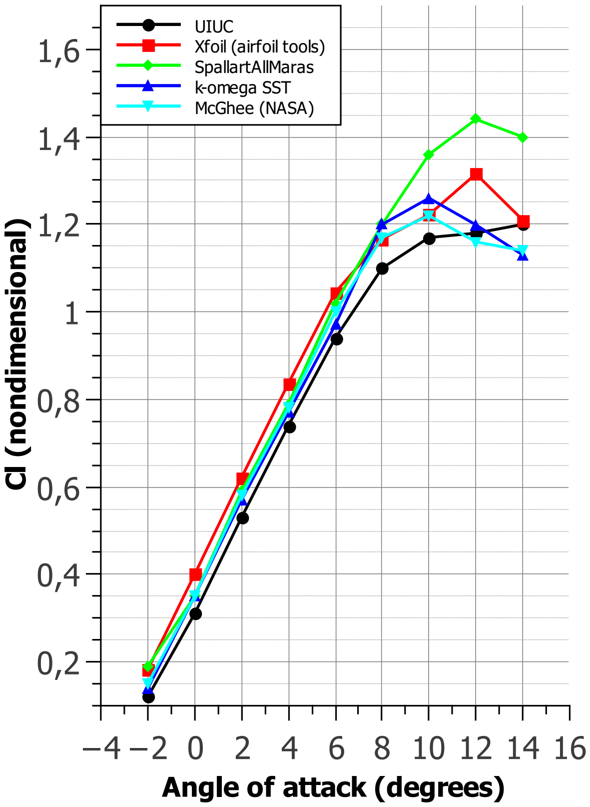

Figure 4. Lift coefficient for different angles of attack. Reference: The authors.

In Fig. 4 it is presented the evolution of the lift coefficient against the angle of attack. For the same Reynolds Number, there is the comparative with other authors (Drela (1989); McGhee (1988)). From -2 to 8 degrees, the lift coefficient presents linear behavior. From 8 degrees, there is a very different behavior (stall) where the increase in lift is minimal and further increase in the angle of attack leads to a decrease of the lift coefficient.

The stall represents a phenomenon in which aerodynamic profiles sustentation drops sharply when subject to elevated angles of attack. It occurs due both substantial adverse pressures and considerable boundary layer displacement rate at the top of the wing. This kind of graphic shows a range of angles in which the airplane must remain in flight, avoiding, therefore, the phenomenon. It is possible to see that, near stall angles, computational models started to diverge from the experimental value and from themselves. It occurs due to the difficulty of RANS models prediction in regions with elevated displacement of boundary layers.

The SST k-omega ended up oversizing in an acceptable way only sustentation coeficientes (Cl) values near to the stall. The authors believe that this phenomenon happens due turbulent boundary layers displacement´s delay. The turbulence levels (eddy viscosities, therefore) on boundary layers are very elevated, increasing moment transferences in regions nearby the wall. It helps the boundary layers to cross more easier regions of adverse pressure gradient. These elevated turbulence levels are caused by the model assuming totally turbulent flows, what ends to retard this displacement in the region. Areas with lower pressure are spread over a larger region of the wing, escaping, this way, from a reliable representation of the physical phenomenon. This delay cannot be seem on experiments and is quite well-known in literature (Matyushenko, 2017).

The use of SpallartAllMaras turbulence model increased the necessary angle for reaching the stall and also oversized lift values close to the phenomenon.

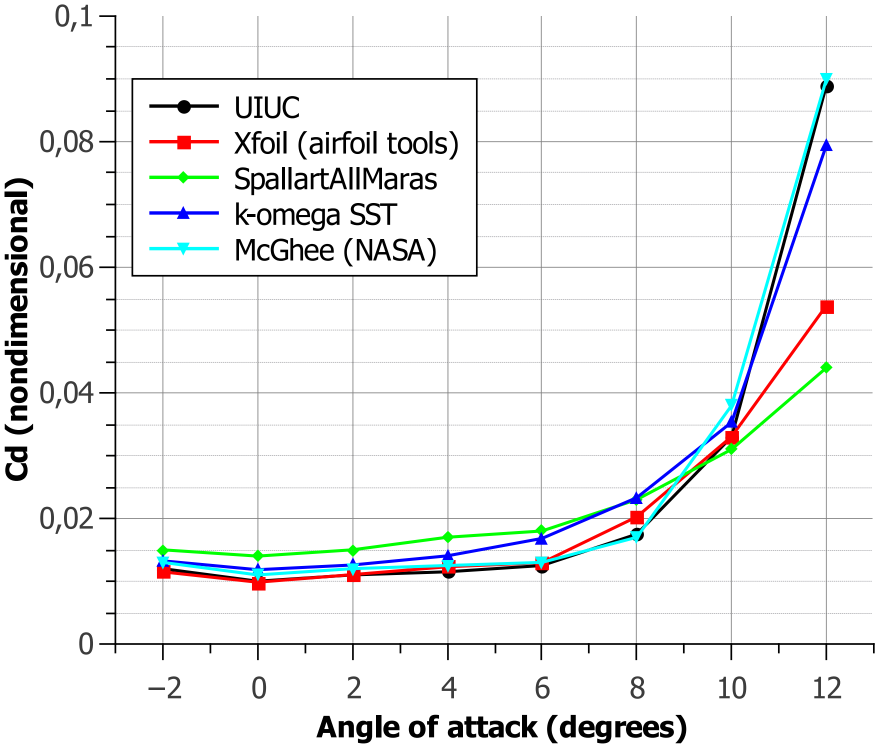

Figure 5. Drag coefficient for different angles of attack. Reference: The authors.

Analysing the drag coefficient, the stall owns a different manner of being studied. The drag by friction ends up having a abrupt increase. As in the previously case, it is observed that its considerable elevation occurs to angles above 8 degrees, corroborating the stall in this angulation.

Another evident characteristic is the experimental data having a lower drag when comparing to many models between the angles from -1 to 8 degrees. A good agreement found in bibliography about this issue is related to laminar boundary layers presence over a considerable part of the profile, which reduces friction drags. This physical phenomenon is widely studied with the application of sensible to regim transition flow models. But for angles above 8 degrees, the turbulence boundary layer prevails and computational models predicts better the physical phenomenon Fig. 5.

The Spallart Allmaras provided subestimated values in relation to the experimental data. The k-omega SST model better predicted the experimental results.

4. CONCLUSIONS

This study evaluated the k-omega SST and the Spallart Allmaras turbulent models to evaluate the aerodynamic forces in the Eppler 387 aerofoil. The k-omega SST provided better predictions when compared to the Spallart Allmaras.

As seen on graphics above, semi-empirical solution models based on RANS equations – SpalartAllMaras and SST k-omega, for example – are not good prediction models for aerodynamic characteristics in angle of attack near the stall. Being defective in many aspects, its shown the need of transient models with equations solutions of higher order and better computational processing for studies that emphasizes this phenomenon.

Also, it was seen the laminar boundary layer effect over the profile, and the difficulty of RANS models to predict some physical flow phenomena, making, therefore, necessary the use of models containing a better sensibility to flow regimes transition for a better accuracy of results.

It is evident the importance of using several computational modeling when the phenomenon to be studied is of high complexity. Each one has its own characteristics that answers better to certain issue solicitations. Thus, it is necessary a set of them to better pedict a physical phenoma.

5. ACKNOWLEDGEMENTS

The authors are gratefull to UFGD support to the development of the current study.

6. REFERENCES

Anderson, 2001. *Fundamentals of Aerodynamics.*, Vol. Third Edition.

Bottema, M., 1997. “Urban roughness modelling in relation to pollutant dispersion”. *Atmospheric Environment*, Vol. 31, No. 18, pp. 3059–3075.

Brown, G.J., Fletcher, D.F., Leggoe, J.W. and Whyte, D.S., 2018. “Investigation of turbulence model selection on the predicted flow behaviour in an industrial crystalliser—rans and urans approaches”. *Chemical Engineering Research and Design*, Vol. 140, pp. 205–220.

Celik, 1999. *Introductory Turbulence Modeling.*, Vol. 1nd ed.

Cheng, C.M., Lu, P.C. and Tsai, M.S., 2002. “Acrosswind aerodynamic damping of isolated square-shaped buildings”. *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 90, No. 12-15, pp. 1743–1756.

Coleman and Sandberg, 2010. “A primer on direct numerical simulation of turbulence methods, procedures and guidelines.” *University of South Hampton. South Hampton,*, p. 21.

Drela, 1989. “Xfoil: An analysis and design system for low reynolds number airfoils”. In *Low Reynolds number aerodynamics*, Springer, pp. 1–12.

Fernandes, 2010. “Refrigeração utilizando pastilhas de efeito peltier”. *HOLOS*, Vol. 2, pp. 25–31.

Foundation, T., 2020. *Openfoam v7 user guide*. Master’s thesis, 8 Apr. 2020 <https://cfd.direct/openfoam/user-guide>.

Kato, T., Fujita, T., Ito, Y., Nakahashi, K. and Kohama, Y., 2001. “Computational analysis of separated flow around a golf ball using unstructured grid cfd”. In *15th AIAA Computational Fluid Dynamics Conference*. p. 2569.

Maliska, 2004. *Transferência de Calor e Mecânica dos Fluidos Computacional.*, Vol. 2nd ed.

Martínez-Aranda, S., García-González, A., Parras, L., Velázquez-Navarro, J. and Del Pino, C., 2016. “Comparison of the aerodynamic characteristics of the naca0012 airfoil at low-to-moderate reynolds numbers for any aspect ratio”. *International Journal of Aerospace Sciences*, Vol. 4, No. 1, pp. 1–8.

Matyushenko, 2017. “Calculations of flow around airfoils using two-dimensional rans: an analysis of the reduction in accuracy”. *St. Petersburg Polytechnical University Journal: Physics and Mathematics*, Vol. 3, No. 1, pp. 15–21.

McGhee, 1988. *Experimental results for the Eppler 387 airfoil at low Reynolds numbers in the Langley low-turbulence pressure tunnel*, Vol. 4062. National Aeronautics and Space Administration, Scientific and Technical ....

Menter, F.R., 1992. “Improved two-equation k-omega turbulence models for aerodynamic flows”. *Nasa Sti/recon Technical Report N*, Vol. 93, p. 22809.

Menter, F.R., Kuntz, M. and Langtry, R., 2003. “Ten years of industrial experience with the sst turbulence model”. *Turbulence, heat and mass transfer*, Vol. 4, No. 1, pp. 625–632.

Pope, 2004. *Turbulent Flows.*, Vol. illustrated. ed.

Research, A., 2020. `https://www.aberdeen.com/?s=computational+development`.

Schumann, 1974. “A procedure for the direct numerical simulation of turbulent flows in plate and annular channels and its application in the development of turbulence models.” *NASA, Washington*, p. 321.

Spalart, 1992. “A one-equation turbulence model for aerodynamic flows”.

Tools, a., 2020. “Airfoil tools”. <http://www.airfoiltools.com>.

Zachow, S., Steinmann, A., Hildebrandt, T., Weber, R. and Heppt, W., 2006. “Cfd simulation of nasal airflow: towards treatment planning for functional rhinosurgery”. *International Journal of Computer Assisted Radiology and Surgery*, Vol. 1, No. 7, pp. 165–167.

7. RESPONSIBILITY NOTICE

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