STUDY OF MESH REFINEMENT ON THE AERODYNAMIC COEFFICIENTS FOR NACA2412 PROFILE WITH DIFFERENT ANGLE OF ATTACK AND $k - \omega$ TURBULENCE MODEL

ESTUDO DO REFINAMENTO DE MALHA NOS COEFICIENTES AERODINÂMICOS PARA O PERFIL NACA2412 COM DIFERENTES ÂNGULOS DE ATAQUE AND MODELO DE TURBULÊNCIA $k - \omega$

Nícolas Lima Oliveira
Eric Vargas Loureiro
Patrícia Habib Hallak

Abstract: This work presents the studies obtained using OpenFOAM OpenSource Computational Fluid Dynamics (CFD) Software. Experiments were performed to predict lift coefficient and drag coefficient curves for the NACA2412 profile. Subsequently, the results obtained were compared with the results of the bibliography and discussed.

Keywords: CFD. OpenFOAM. Airfoil.

Resumo: Este trabalho apresenta os estudos realizados na plataforma de dinâmica dos fluidos computacional OpenFOAM. Foram executados experimentos para prever as curvas de coeficientes de sustentação e arrasto para o aerofólio NACA2412. Em sequência, os resultados obtidos foram comparados aos resultados bibliográficos e discutidos.

Palavras-chave: CFD. OpenFOAM. Perfil Aerodinâmico.

1MSc, Universidade Federal de Juiz de Fora, nicolas_limaoliveira@yahoo.com.br.
2MSc, Universidade Federal de Juiz de Fora, eric.vargas@engenharia.ufjf.br
3DSc, Universidade Federal de Juiz de Fora, patriciahallak@yahoo.com
1 INTRODUCTION

OpenFOAM (GREENSHIELDS, 2015) is an open, free software developed and distributed since 2004. OpenFOAM serves a wide range of areas of science and engineering, for business and academic purposes. Its wide range of tools are able to solve computational fluid dynamics, acoustics, solids mechanics and electromagnetism problems. In this work, OpenFOAM 2.3.1 was used as a tool to study the aerodynamic airfoil NACA2412, which had been extensively studied, both experimentally and computationally (SEDLÁŘ et al., 2015; SEETHARAM; RODGERS; JR, 1997; YLILAMMI; CAVALIERI; SINOINE, 2009). Drag and lift coefficients were obtained for the airfoil using different levels of mesh refinement and two approaches, one laminar and another with k-ω turbulence model. Results from both the panel method and wind tunnel experiments were used as reference values (SAXENA; KUMAR, 2015).

2 AERODYNAMICS PARAMETERS AND TURBULENCE MODEL

It is necessary for the understanding of this work a simple description of some important parameters. NACA nomenclature, nondimensional coefficients and the k-ω turbulence models are presented in this section.

2.1 NACA Profiles

The NACA (National Advisory Committee for Aeronautics), NASA’s predecessor, has created a four-digit system to describe a large number of aerodynamic profiles. Subsequently, a five-digit system was created. The dimensions used to define a profile can be seen in the Figure 1 (JR, 2001).

Figure 1: Dimensions used for 4-digit NACA nomenclature - Obtained from <http://www.akiti.ca/NACA4Bkgrnd.html>
The mean camber line is the average between the upper and lower surfaces of the airfoil. The forwardmost point is known as the leading edge, while the opposite point is known as the trailing edge. The straight line between these two points is the chord line. The length of the chord line is known as chord, $c$. The maximum distance between the chord line and the camber line is known as the maximum camber ($C_{\text{max}}$) and the position where it is located is known as $X_{C_{\text{max}}}$. The thickness of the profile is the maximum distance between the lower and upper surfaces measured perpendicular to the chord line.

In this way, the four digits of the aerodynamic profile name are:

- 1st digit: maximum camber ($C_{\text{max}}$) in percentage terms.
- 2nd digit: position of the maximum camber ($X_{C_{\text{max}}}$) in percentage terms.
- 3rd and 4th digits: thickness in percentage terms.

The aerodynamic profile studied in this work is NACA2412, which is, therefore, a maximum camber profile of 20% at the 40% position of the chord and with a thickness of 12%.

### 2.2 Adimensionalization of Forces Coefficients

For the study of the forces acting on an aerodynamic profile it is common to use coefficients (JR, 2001). The lift and drag forces are, respectively, denoted by $L$ and $D$. The lift coefficient is given by:

$$C_L = \frac{2L}{\rho v^2 S}$$  \hspace{1cm} (1)

And the drag coefficient is given by:

$$C_D = \frac{2D}{\rho v^2 S}$$  \hspace{1cm} (2)

where $\rho$ is the density of air, $v$ is the velocity of the fluid, and $S$ is the reference area. The airfoil was modeled so that the value of its reference area was 1.

### 2.3 Turbulence Model

According to Versteeg & Malalasekera, 2007 and Wilcox, 1998, all the actual flows become turbulent from a certain value of the Reynolds number. In this case, a chaotic and random state of motion is observed in which speeds and pressures change continuously with
time. The Reynolds number provides a measure of relative importance of the inertial forces (associated to the convective term) and the forces viscous:

\[ Re = \frac{vc}{\nu} \]  

where \( \nu \) is the viscosity of the fluid.

In many experiments it is observed that for values below called Re critical (Re\( _c \)) the flow is laminar and layers adjacent the fluid slides between them are in an orderly manner. If the boundary condition does not change over time the flow is stable. This regime is called a laminar regime. Laminar flows are completely described by the Navier Stokes equation. For laminar flow, the effects of turbulence are ignored.

For Reynolds number values greater than Re\( _c \), complex events arise that alter the nature of the flow, making it chaotic and random. The movement, called turbulent, becomes unstable even if constant contour conditions are imposed on it.

### 2.4 Turbulence Model k-\( \omega \)

The k-\( \omega \) model utilizes the hypothesis that the diffusion gradient is related to the Reynolds stress tensor, the mean velocity gradients and the turbulent viscosity. The turbulent viscosity, in turn, is modeled as the product of a turbulent velocity and turbulent-length scale. In the two equation models, the turbulent velocity scale is calculated by the turbulent kinetic energy, which is obtained from the solution of its transport equations. The turbulent length scale is estimated from two properties of the turbulent field, usually the turbulent kinetic energy and its dissipation rate, which is obtained from the solution of its transport equations (WILCOX et al., 1998).

The turbulence models of two equations are widely used, since they offer a good balance between computational cost and accuracy. In these models, velocity and length scale are solved through separate transport equations.

One of the advantages of the formulation of the k-\( \omega \) model is the near-wall treatment for problems with low Reynolds numbers (WILCOX et al., 1998). The model does not involve complex damping functions required by the k-\( \varepsilon \) model and therefore is more accurate and robust.

The k-\( \omega \) model considers that the turbulent viscosity \( \mu_t \), turbulent kinetic energy \( k \) and turbulence frequency \( \omega \) are connected through the ratio:

\[ \mu_t = \frac{k}{\omega} \]  

Revista Mundi, Engenharia e Gestão, Paranaguá, PR, v. 5, n. 2, p. 216-01, 216-12, 2020

DOI: 10.21575/25254782rmetg2020vol5n21141
2.5 Model k-ω of Wilcox

The first formulation of the k-ω model was developed by Wilcox. The formulation solves two transport equations (FLUENT et al., 2011):

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial}{\partial x_j} (\rho U_j k) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_k}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + P_k - \beta' \rho k \omega + P_{kb} \tag{5}
\]

\[
\frac{\partial (\rho \omega)}{\partial t} + \frac{\partial}{\partial x_j} (\rho U_j \omega) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_w} \right) \frac{\partial \omega}{\partial x_j} \right] + \alpha \omega \frac{P_k}{k} - \beta \rho \omega^2 + P_w b \tag{6}
\]

The density, \( \rho \), vector of velocities, \( U \), are obtained through the Navier-Stokes method. \( k \) is the turbulent kinetic energy, \( \omega \) is the specific dissipation rate, \( P_k \) is calculated in the same way as in the model k-ε. The constants of the model are given by:

\[\beta' = 0.09\] (7)

\[\alpha = 5/9\] (8)

\[\beta = 0.075\] (9)

\[\sigma_k = 2\] (10)

\[\sigma_\omega = 2\] (11)

The Reynolds stress tensor is calculated from:

\[-\rho \bar{u_i} \bar{u_j} = \mu_t \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} (\rho k + \mu_t \frac{\partial U_k}{\partial x_k})\] (12)

where \( \mu_t \) is the turbulent viscosity. \( \delta_{ij} \) is 1 when \( i = j \) and 0 when \( i \neq j \).

3 EXPERIMENTS

The simulation domain for the experiment (Figure 2) consists of the following boundary conditions (FLUENT et al., 2011):

- Inlet: A normal speed is defined at the input surfaces of the domain. The magnitude of the velocity was chosen so as to obtain the desired Reynolds number.

- Outlet: is set on the domain output surface. The reference pressure was set to 1atm.
• No-Slip Wall: the wall condition is defined at the lower and upper boundaries of the airfoil.

• Slip Wall: the wall condition is defined at the lower and upper boundaries of the wall.

The aerodynamic profile chord length was defined as unitary. The size of the domain was defined according to the bibliography (FLUENT et al., 2011).

The angle of attack used in the simulations were: 0°, 5°, 9°, 12°, 15° and 18°.

Three different levels of refinement were used for the meshes of each of the angles of attack For 0° angle of attack meshes (3), we list:

• Coarse Mesh: 299382 nodes, 720082 faces and 213728 volumes;

• Mid-mesh: 700705 nodes, 1688713 faces and 501656 volumes;

• Fine Mesh: 1356145 nodes, 3251662 faces and 961508 volumes.

For the experiments, parameters were defined so that the results obtained with the reference database (AIRFOIL..., ) could be compared:

\[ v = 7,2 m/s \]  
\[ \rho = 1,1965 kg/m^3 \]  
\[ Re = 5 \times 10^5 \] 

The angle of attack used in the simulations were: 0°, 5°, 9°, 12°, 15° and 18°.
For the other angles of attack small variations were observed, once we kept the same level of refinement. For the simulation with laminar flow, the most refined mesh was used, while for the simulations with turbulence model $k-\omega$, all the meshes with different levels of refinement were used.

For the pressure-velocity coupling were used the SIMPLE (Semi-Implicit Method for Pressure Linked Equations) and for discretization method were used the linear upwind scheme, both standard algorithms in OpenFOAM. Also, a steady state solver was used.

4 RESULTS

The results obtained from the simulations can be visualized in the Table 1 and graphically in the Figures 4 and 5:
| Angle (°) | L (N) | D (N) | Cl    | Cd    | L (N) | D (N) | Cl    | Cd    |
|----------|-------|-------|-------|-------|-------|-------|-------|-------|
| 0        | 4.29  | 0.576 | 0.138328 | 0.018573 | 7.21  | 0.466 | 0.232481 | 0.015026 |
| 5        | 23.56 | 1.04  | 0.759675 | 0.033534 | 24.16 | 1.01  | 0.779021 | 0.032567 |
| 9        | 35.05 | 2.86  | 1.130161 | 0.092219 | 35.32 | 2.12  | 1.138867 | 0.068358 |
| 12       | 41.32 | 4.73  | 1.332332 | 0.152515 | 39.83 | 3.12  | 1.284289 | 0.100602 |
| 15       | 34.15 | 6.55  | 1.101141 | 0.2112  | 45.98 | 4.87  | 1.482591 | 0.15703  |
| 18       | 34.67 | 9.93  | 1.117908 | 0.320185 | 36.2  | 7.05  | 1.167242 | 0.227322 |

Table 1: Results obtained

Figure 4: Lift Coefficient Curves by Attack Angle - The values of the reference curve were obtained in: <http://airfoiltools.com/airfoil/details?airfoil=naca2412-il#polars>
Figure 5: Drag coefficient curves by angle of attack - The values of the reference curve were obtained in: <http://airfoiltools.com/airfoil/details?airfoil=naca2412-il#polars>

The Figures 6 and 7 show the pressure distribution for the flows of the simulations of the 0° and 12° angles, respectively.

Figure 6: Pressure distribution to 0° - Fine mesh
XFOIL is an interactive program for the design and analysis of subsonic isolated airfoils. It consists of a collection of menu-driven routines which perform various useful functions. It was developed by Professor Mark Drela (DRELA; YOUNGREN, 2001) and applies the panel method to airfoils. The results obtained in XFOIL considered viscous effect and were used as reference in this present work.

For the coarse mesh, it was observed that for both the coefficients there were the worst results. For the coefficient of lift curve (Figure 4), there was a sharp drop for the 15° angle. This can be attributed to the stall phenomenon that happened in this angle of attack. For the drag coefficient curve (Figure 5), it is noted that all other meshes were closer to the reference curve. Such behavior can be attributed to the number of nodes, faces and volumes insufficient to describe the physical phenomenon with greater precision. It is also observed that the greater the refinement of the mesh, the closer the respective curve approaches the expected behavior.

By the analysis of the results, it is also possible to conclude that the turbulent model k-ω and the laminar model, both applied to the most refined meshes, presented slight differences. The largest difference is only 4%, for the angle of 18° in the coefficient of lift analysis. It is concluded from this observation that the viscous effects are not preponderant in the simulated model.

From the Figures 6 and 7, we notice that the pressure distribution around the airfoil is more pronounced for the angle of 12°. Such behavior is expected and compatible with reality.
The sum of the high pressure on the lower surface and the low pressure on the upper surface are the natural factors that cause the forces of lift and drag.

Due to the small difference between the laminar model and the k-ω model it is possible to assume that the flow is almost laminar for the Reynolds number used. This observation is supported by the work of Yousefi, 2018.

5 CONCLUSIONS

The present work presented comparisons between drag and lift coefficient values obtained through computational fluid dynamics using OpenFOAM. The results obtained were compared with values obtained by the panel method.

It was observed that the mesh refinement made the results more compatible with the reference, while the k-ω turbulence model did not present large differences when compared to the laminar model.

For future work we intend to expand the study in relation to the refinement of the mesh. It is also an important idea to use larger Reynolds numbers for simulations to enable comparison with experimental wind tunnel results. It is also necessary to add new simulation angles in the range between 12° and 18°, where it was observed the greatest discrepancies between the results obtained and the reference values.

ACKNOWLEDGMENTS

This work was conducted during a scholarship supported at the Universidade Federal de Juiz de Fora - UFJF. Financed by CAPES – Brazilian Federal Agency for Support and Evaluation of Graduate Education within the Ministry of Education of Brazil.

REFERENCES

AIRFOIL Tools, 2019. Accessed: 2019-01-17.

DRELA, M.; YOUNGREN, H. XFOIL 6.94 user guide. [S.l.]: MIT Aero & Astro, 2001.

FLUENT, A. et al. Ansys fluent theory guide. ANSYS Inc., USA, v. 15317, p. 724–746, 2011.

GREENSHIELDS, C. J. Openfoam user guide. OpenFOAM Foundation Ltd, version, v. 3, n. 1, p. e2888, 2015.

JR, J. D. A. Fundamentals of Aerodynamics. [S.l.]: McGraw-Hill Book Company, 2001.
SAXENA, E. S.; KUMAR, M. R. Design of naca 2412 and its analysis at different angle of attacks, reynolds numbers, and a wind tunnel test. *International Journal of Engineering Research and General Science*, v. 3, n. 2, p. 193–200, 2015.

SEDLÁŘ, M. et al. Numerical and experimental research on unsteady cavitating flow around naca 2412 hydrofoil. In: IOP PUBLISHING. *IOP Conference Series: Materials Science and Engineering*. [S.l.], 2015. v. 72, n. 2, p. 022014.

SEETHARAM, H.; RODGERS, E.; JR, W. W. Experimental studies of flow separation of the naca 2412 airfoil at low speeds. 1997.

VERSTEEG, H. K.; MALALASEKERA, W. *An introduction to computational fluid dynamics: the finite volume method*. [S.l.]: Pearson education, 2007.

WILCOX, D. C. et al. *Turbulence modeling for CFD*. [S.l.]: DCW industries La Canada, CA, 1998. v. 2.

YLILAMMI, N.; CAVALIERI, A. V. G.; SOINNE, E. *Experimental and computational study of two flapped airfoils at low Reynolds Numbers*. Tese (Doutorado) — Dissertação (Mestrado)—Helsinki University of Technology, 2009.