The Eurasia Proceedings of Science, Technology, Engineering & Mathematics (EPSTEM)
ISSN: 2602-3199

The Eurasia Proceedings of Science, Technology, Engineering & Mathematics (EPSTEM), 2019

Volume 7, Pages 76-86

IConTES 2019: International Conference on Technology, Engineering and Science

Evaluation of Rans Turbulence Models for Turbulent Flow over Nrel S809 Airfoil

Bouabdellah ABED
Université des Sciences et de la Technologie

Abdelouahab BENZERDJEB
Université des Sciences et de la Technologie

Habib ACHACHE
Université Mohamed Benahamed Oran 2

Abiddine DEBAB
Université des Sciences et de la Technologie

Mohamed-Kamel HAMIDOU
Université des Sciences et de la Technologie

Abstract: In this presented work, a numerical investigation was conducted to evaluate the performance of RANS turbulence models implemented in ANSYS CFX to predict flow field and aerodynamic characteristics of the NREL S809 Airfoil. The simulations were carried out with ANSYS CFX R18.2 (academic version). The blade geometry was simplified into a 2D airfoil and the flow analysis around the profile was conducted at various angles of attack equal to 0°, 5.13°, 9.22°, 11.24° and 20.15°, at a constant and same Reynolds number equal to $2x10^6$. In this paper, the turbulence flow patterns and separation around the profile were represented using the most turbulence models used in CFD codes, the standard k-ε and its modifications RNG k-ε. The moment coefficient (C_m), the lift coefficient (C_l) and the drag coefficient (C_d) were determined and plotted as a function of the angle of attack. The pressure distributions around this profile, for each angle of attack, were also presented. The results and the plots obtained have shown good agreements with published experimental results. From the simulations, the two turbulence models provided near prediction to the experimental values at low angle of attacks, but over predict the Cd values. Calculations have also shown that the standard k-ε model and RNG k-ε model produced similarly results but with partly differences in pressure magnitude.

Keywords: NREL S809 Airfoil, CFD simulations, RANS turbulence models, Aerodynamic characteristics

Introduction

For the last three decades, fluid dynamics (CFD) approach has been the most appropriate method used almost exclusively by researchers to evaluate the hydrodynamic properties of the flow field over wind. This method provides a detailed description of a phenomenon and consequently gives a good description of flow fields around wind turbine blades. With the recent advances in computing power, together with advanced solvers that enable to give robust solutions of the flow field in a reasonable time, CFD approach provides a practical tool to model and simulate the aerodynamic performances of wind turbine airfoils.

Some of the interesting work concerning CFD analyses of the flow field around S809 airfoil are presented below:
- Wolfe and Ochs (1997) studied the aerodynamic characteristics of S809 and compared the numerical results with experimental data. They used the standard k-ε turbulence model, a fully turbulent and a mixed

- Selection and peer-review under responsibility of the Organizing Committee of the Conference

laminar/turbulent flow. Their calculated results agree well with the experimental data for smaller angles but failed for 14.24° and 20.15°;
- Guerri et al. (2006) employed a 2D Navier–Stokes simulation of steady and unsteady flow for the S809 airfoil. They studied different turbulence models, SST k-ω and the RNG k-ε model. They establish that k-ω model is better to simulate flow instability region than the k-ε model;
- David Hartwanger et al. (2008) conducted a CFD analysis for NREL S809 airfoil using the various codes, 2D panel code, X-Foil ANSYS CFX 2D code. They concluded that using a high-resolution mesh with advanced turbulence models provide an excellent match with experimental;
- Haipeng et al. (2017) investigated by the fluid dynamic method the aerodynamic performance of the airfoil S809 without and with the use of vortex generators. They discussed the effects of vortex generators on the airfoil S809 boundary layer and analyzed quantitatively the lift coefficient, drag coefficient, x-velocity gradient vorticity.

The main objective of this work is to evaluate the performance of different RANS turbulence models implemented in ANSYS CFX solver code. There are several RANS models available in ANSYS CFX and each model has been developed with different objectives, strategies, and measures in mind. In this work, two Linear Eddy Viscosity models have been tested, the standard k-ε model (Launder and Spalding, 1974) and it variants, RNG k-ε model (Yakhot wet al. 1992). The objective is to evaluate the ability of these two models to determine the hydrodynamics coefficients, pressure coefficient (C_p), moment coefficient (C_m), lift coefficient (C_l), and drag coefficient (C_d), of NREL S809 airfoil at fixed Reynolds number of $2.0\times10^{+06}$. The simulated results will be compared to wind tunnel test results of the wind turbine dedicated airfoils made by Delft University of Technology (DUT) (Somers, 1997).

We will not try to explain in depth why models fails or succeed in approximating these quantities, it will simply state that a model gives good results or failed results.

For the evaluation, the software ANSYS CFX R18.2 version academic (ANSYS, Inc, 2018) is used for calculation of the flow field over the NREL S809 airfoil.

Method

Turbulent flow calculations have been performed in two-dimensional steady state flow over a S809 NREL airfoil at different angles of attack. In a steady flow, hydrodynamics cooficients and mean velocity field of the airfoil have been calculated by two turbulence models, k-ε model and RNG k-ε model.

Turbulence Modelling

The governing equations used in the simulations are the RANS equations. The RANS Eddy Viscosity Models are commonly used in the engineering applications for their robustness, low computation costs and providing good results. In order to model and simulate the turbulent flow in the present study we have selected both k-ε turbulence and RNG k−epsilon models. The simulations were done for a Reynolds number of $2.0\times10^{+06}$ with three different grids.

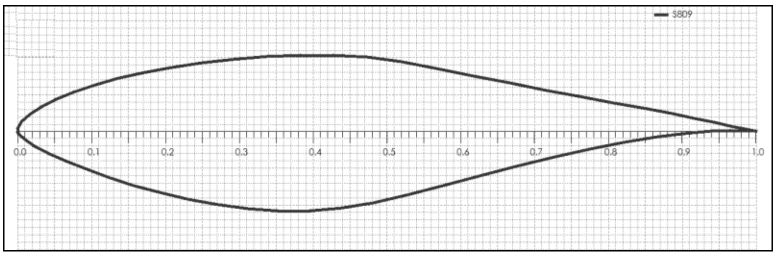


Figure 1 . Schematic of the NREL S809 airfoil

The NREL S809 airfoil is a laminar flow aerodynamic profile 21% thick (Figure 1), specifically designed for horizontal axis wind turbine (HAWT) applications (Somers, 1997). This profile is justified by the existence of a very large number of studies carried out in the field of wind turbines, which makes it easier to obtain experimental data (Butterfield et al., 1992) and (Somers, 1997) and numerical results from other CFD software. This will allow more comparisons and gives a better evaluation of the models proposed by CFX solver.

Computational Mesh

The computational domain is defined as 2-D; extend to a distance of 5C (C = airfoil cord) long in all directions from the airfoil aerodynamic center and 20C long at the wake. The two-dimensional C-H structured mesh around the S809 airfoil shape is generated using ICEM CFD code. The numerical domain is divided into 22 blocks with 4 blocks lying very close to the airfoil, which have very refined mesh size when compared with the remaining blocks far from the airfoil (Figures 2-a and 2-b).

Three mesh sizes are done, namely coarse, medium and fine mesh, as shown in the Table 1. A boundary layer mesh is used at the wall of the airfoil, with the first near-wall node placed at $Y_{Plus} \approx 60$. Figures 3-a and 3-b show, respectively, the whole mesh of the domain and the refinements near the airfoil with a coarse mesh (Figure 3-b).

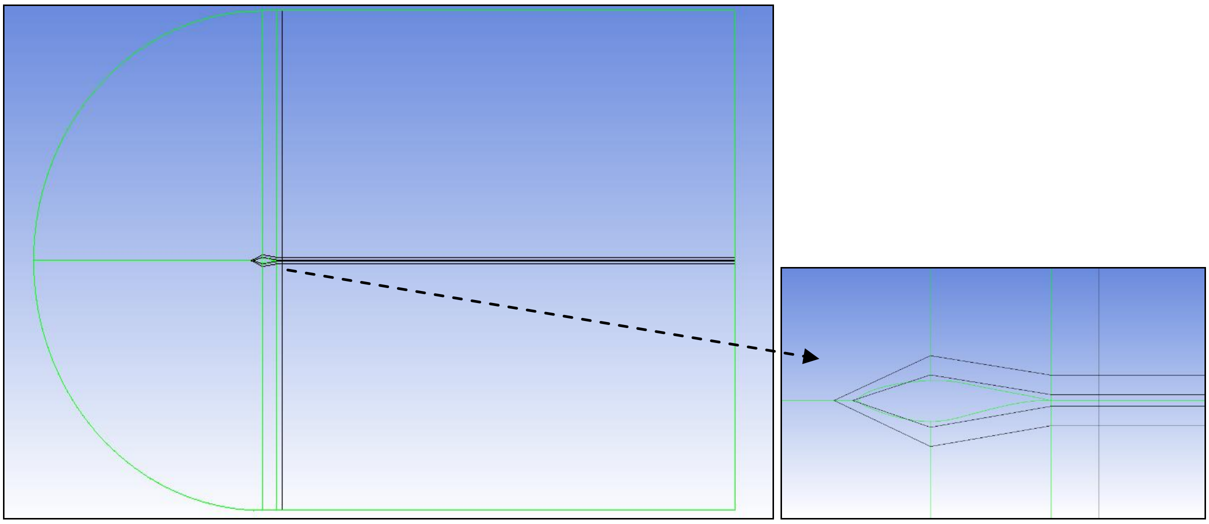
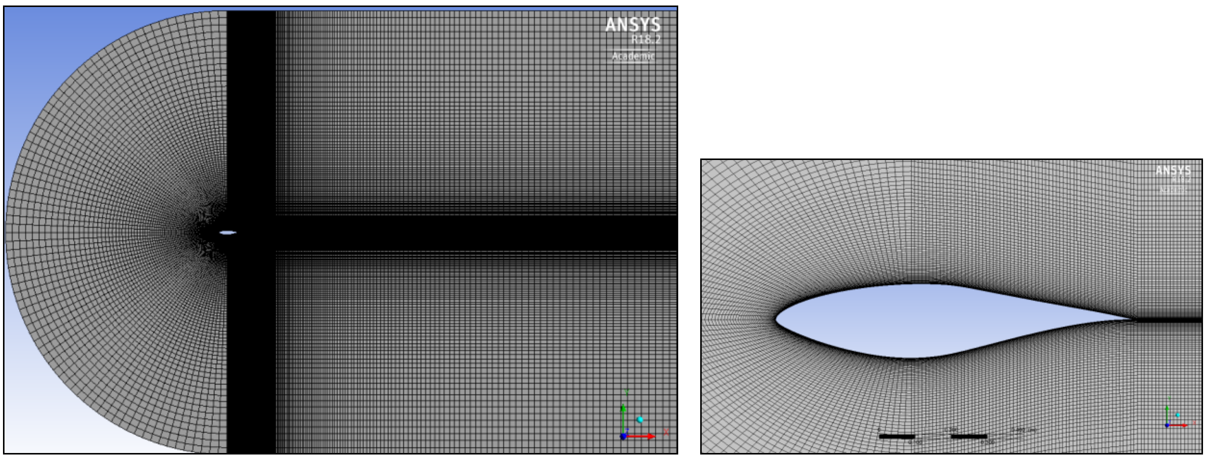


Figure 2. C-H grid topology around the NERL s809 airfoil



(a) (b)

Figure 3. Numerical Domain and Mesh
(a) Domain with Coarse Mesh ; (b) Mesh detail around S809 airfoil shape

Table 1. Mesh Information

Mesh	Local Mesh Size on Airfoil shape	Number of Hexahedra Elements
Coarse	0.010 C	71 484
medium	0.005 C	41 514
Fine	0.001 C	302 697

Boundary Conditions and solver setting

Velocity components are specified at the inflow boundary of the computational domain as follow: $U_x = U_\infty \cos(AOA)$ and $U_y = U_\infty \sin(AOA)$, where (*AOA*) is the angle of attack. The free stream velocity is $U_\infty = 1\ [m/s]$ and the dynamic viscosity μ is calculated based on the fixed Reynolds number, $\mathrm{Re} = 2.0 \times 10^{+06}$ chord length, $C = 1[m]$, and the Density $\rho = 1[kg/m^3]$.

The reference pressure is set to 1 atmosphere and the relative pressure is zero at the outflow boundary placed at the right side of the numerical domain. Along the airfoil surface, no-slip boundary conditions are imposed, while the top and the bottom of the numerical domain are considered as inflow boundary conditions. Considering the RANS, simulations are carried out on two-dimensional meshes, the two sides of the domain in the z-direction are set to symmetry boundary condition. Finally, the value of turbulence intensity at the inflow boundary is set to 5%.

The analysis is performed using steady state RANS models. ANSYS CFX use pseudo-transient solution approach for steady-state computations. Here the timescale is specified to local timescale with factor 5. The high-resolution scheme is used to evaluate advection terms that are second-order accurate and bounded. The solver stops iterating when RMS residual error values are below to 10^{-06} for all variables or when achieving a maximum number of 10.000 iterations. Initial conditions for simulations are set to the free stream velocity $U_\infty = 1.\ [m/s]$ and turbulence values to 5%.

Results and Discussion

Post processing has been performed for different parameters such as velocity and aerodynamics coeficients for both turbulence models and also for different meshes.

The lift, drag and pitch moment coefficients are dimensionless numbers used to measure the aerodynamic characteristics of the airfoil. These coefficients are expressed as follows:

Lift coefficient : $$C_L = \frac{L}{\frac{1}{2}\rho U_\infty^2 A} \quad (1)$$

Drag coefficient : $$C_D = \frac{D}{\frac{1}{2}\rho U_\infty^2 A} \quad (2)$$

Pitch moment coefficient (calculed about 0.25 chord) : $$C_m = \frac{M}{\frac{1}{2}\rho U_\infty^2 C A} \quad (3)$$

Where: U_∞ is the freestream velocity, A is a reference area and C is a reference length, D is the drag force, L is the drag force, M is the pitching moment and ρ is the mass density of the fluid.

Grid resolution analysis

The graphs below (Figure 4, 5 and 6) show the lift, drag and moment coefficients as a function of the angle of attack, obtained by k-ε and RNG k-ε models.

The values of the coefficients are almost identical for angles of attack less than or equal to 9.22 degrees. Indeed, we observe the same shape and almost the same values obtained by three different meshes. There is little effect of the mesh density on the values of the lift, drag and moment coefficients calculated by the two turbulence models for the small angles of attack.

For the values of the lift at the angle of attack equal to 14.24°, the numerical results obtained by both turbulence models are larger than the experimental results except for the case of the k-ε model for the coarse mesh. Same results are observed for 20.15°, except for this case the k-ε model underestimates the lift for the average mesh.

For the numerical results of the drag coefficient, the curves look the same except for the results obtained for the coarse mesh. The drag values are underestimated for the fine mesh at angles of attack greater than 9.22 degrees, and are closer to the experimental values for the values calculated for the mean mesh.

For the moment coefficient, the curves obtained by the simulations and the experimental curves have the same shape. A slight difference is observed at the 20.15° angle for the results obtained for the three different meshes.

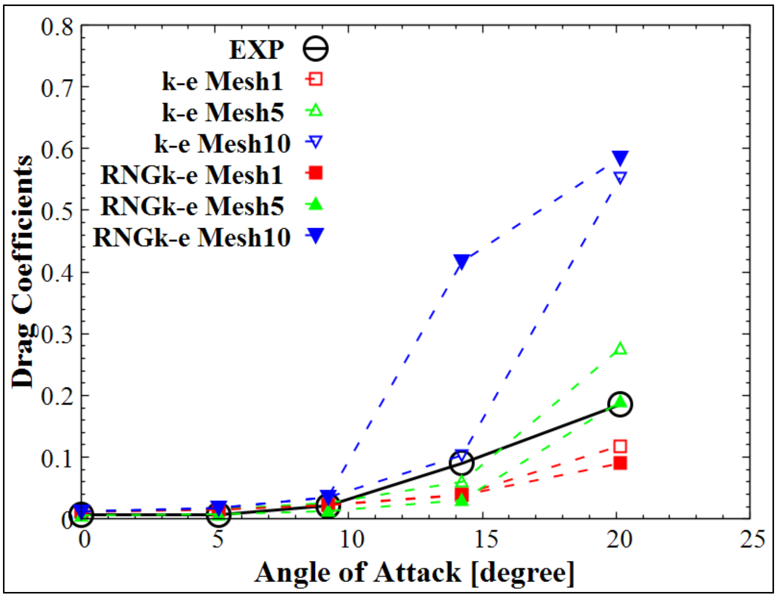


Figure 4. Predicted drag coefficient

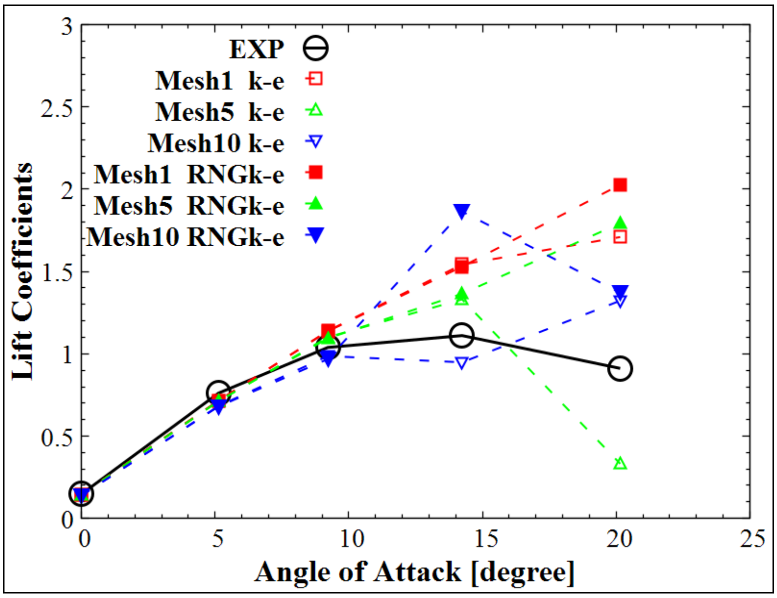


Figure 5. Predicted lift coefficient

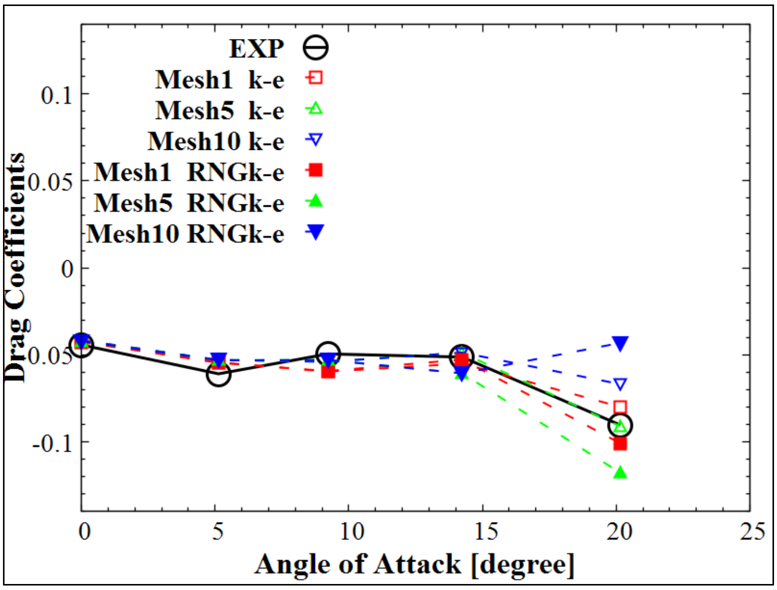


Figure 6. Predicted moment coefficient

3.1 Pressure Coefficient

Figure 7 shows comparisons between numerical and experimental distribution of pressure coefficient of the NREL S809 airfoil at various angles of attack for fine mesh.

We can notice that both turbulence models predict similar results. The graphs show that the pressure coefficient at the upper surface of the airfoil is negative and positive at the bottom surface. They indicate more clearly that the pressure coefficient varies depending on the angle of attack and show a significant difference in pressure coefficient between the leading edge and the trailing edge of the airfoil. We can also see that at 5.13°, the profile indicates a large difference in pressure coefficient between the lower and upper surfaces compared to the other angles of attack.

At angles of attack equal to 14.14° and 20.15°, the two turbulence models k-ε and RNG k-ε failed to predict the pressure coefficient except over the lower surface of the airfoil.

Aerodynamic coefficients

Figures 8, 9 and 10, respectively, show calculated values of drag (C_D), lift (C_L) and moment coefficients (C_m) for fine mesh compared with experimental data. Generally, all the coefficients predicted by the two turbulence models show reasonably good agreement with the experimental results. It seems that the predicted drag and lift coefficients are in quite good agreement with experimental data for the linear part of the curves.

However, for the lift coefficients both models fail to predict stall location. The calculated results compare quite well with experimental data for the angle of attack only up to 9.22 degrees. Figure 8 clearly shows that both turbulence models give the same magnitude of C_L for an angle of attack of up to 14.24°, and shows a separation in the results for AOA > 9.22°, which is found in the calculations obtained by both turbulence models. While for both turbulence models, k-ε and RNG k-ε, smaller values of the C_D are predicted for AOA ≥ 10° (Figure 9), and the divergences with experimental data are more important for AOA greater than this value.

Figure 10 shows the moment coefficient versus angle of attack. The predicted moment coefficient is calculated by assuming that the aerodynamic center is about 25% of chord. The calculated results of the moment coefficient for both k-ε model and RNG k-ε model are in good agreement with experiment data. Again, the comparison looks similar for both turbulence models.

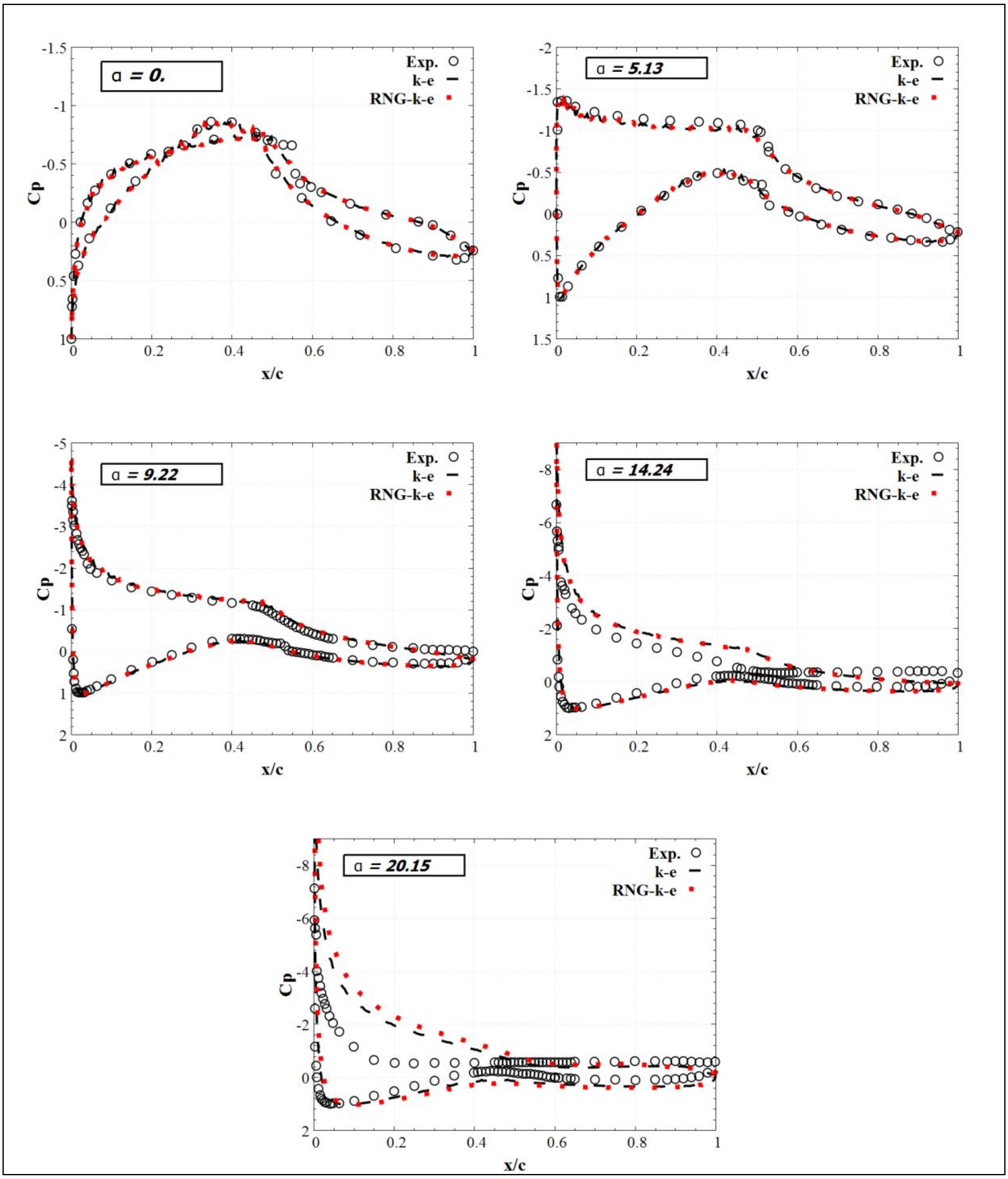


Figure 7. Pressure coefficient at various angles of attack for fine mesh

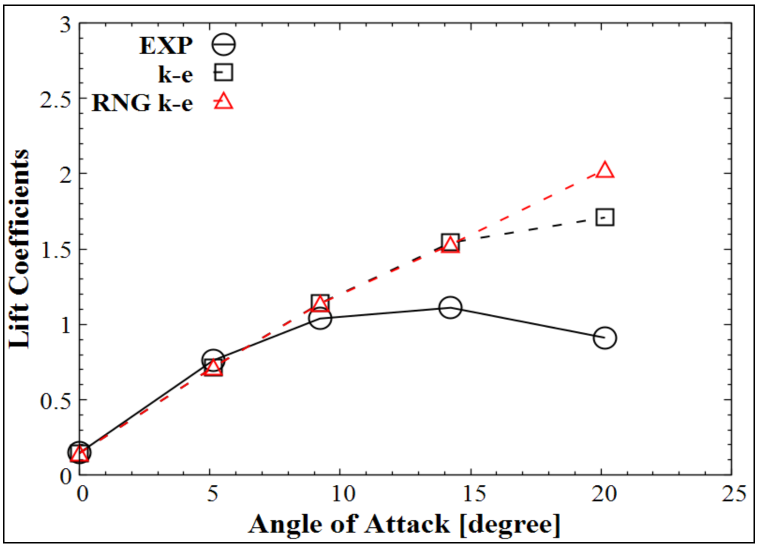


Figure 8. S809 Lift coefficients for fine mesh

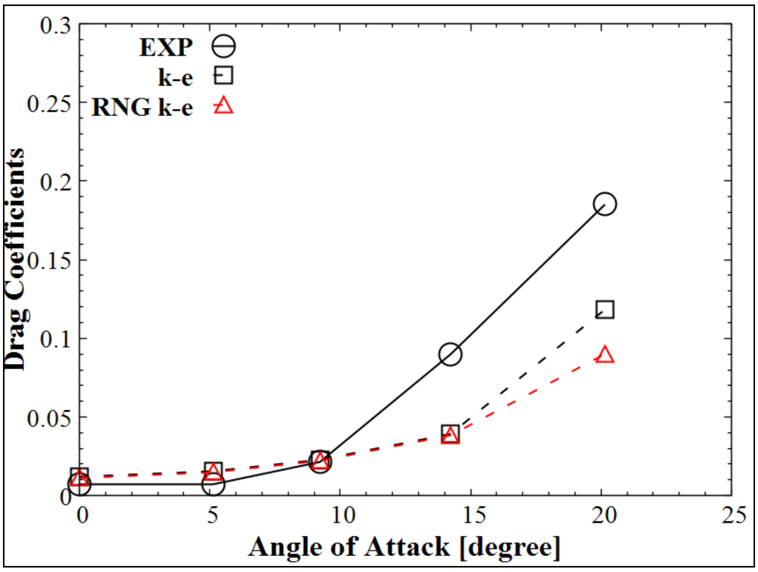


Figure 9. S809 Drag coefficients for fine mesh

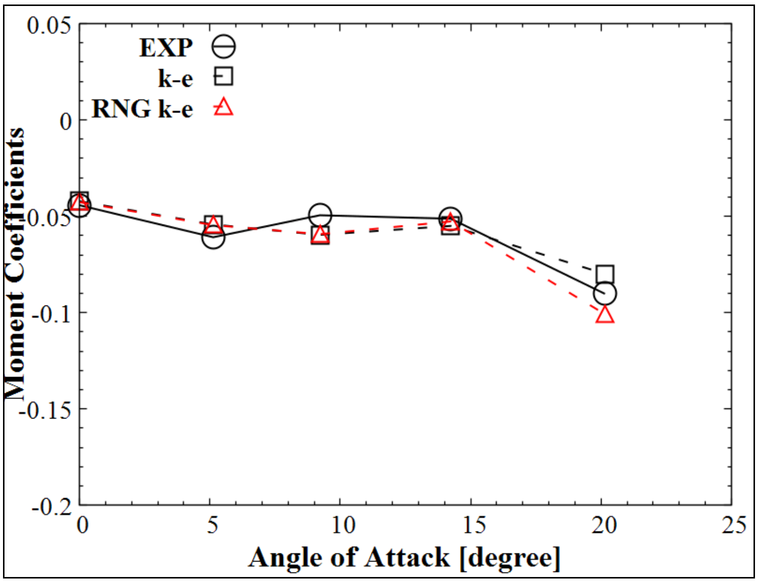
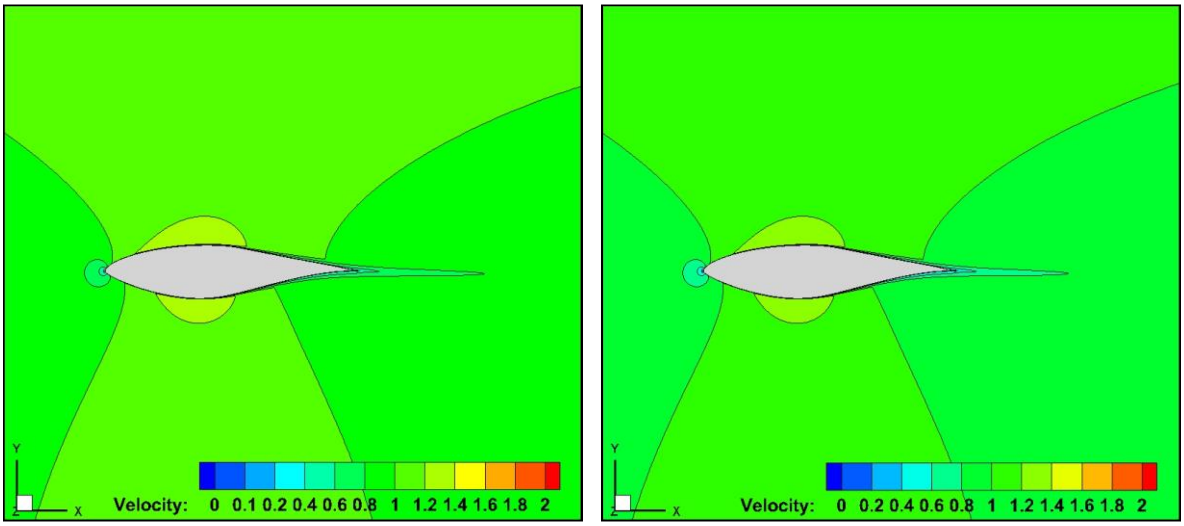


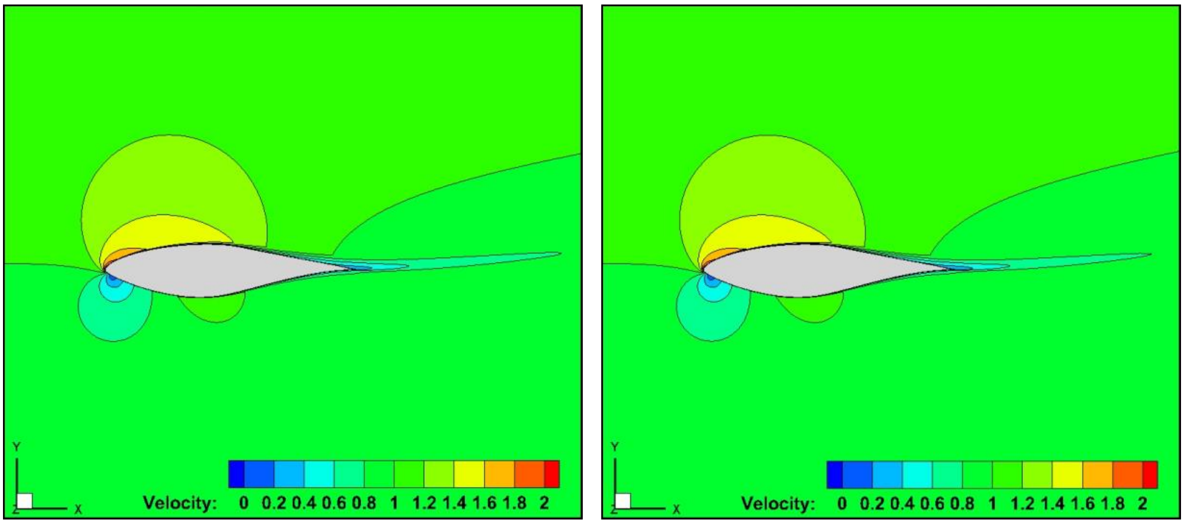
Figure 10. S809 Moment coefficients about 25% chord for fine mesh

Figures 11-12 and 13 show the velocity fields predicted by both turbulence models at 0.0°, 9.22° and 20.15° angle of attack. It should be noted that the k-ε and RNG k-ε models present almost identical velocity contours. Except for AOA = 20.15°, the shape of the wake region formed behind the airfoil present a slight difference. With both models, the turbulent flow remains attached to airfoil surface for AOA = 0° and 9.22°, and is fully separated on the upper surface; separation is predicted at distance superior to 50% from the leading edge. Note that the wake region with zero velocity is important for AOA = 20.15° than AOA = 9.22°, and it is almost nulle for AOA = 0°. In addition, a higher-velocity zone is formed on the extrados close to the leading edge of the airfoil. This zone of increase of speed is more or less important according to angle of attack. The maximum velocity calculated for angle of attack 20.15° is about three times larger than for AOA =0° , and it is about twice times larger than for AOA =9.22°.



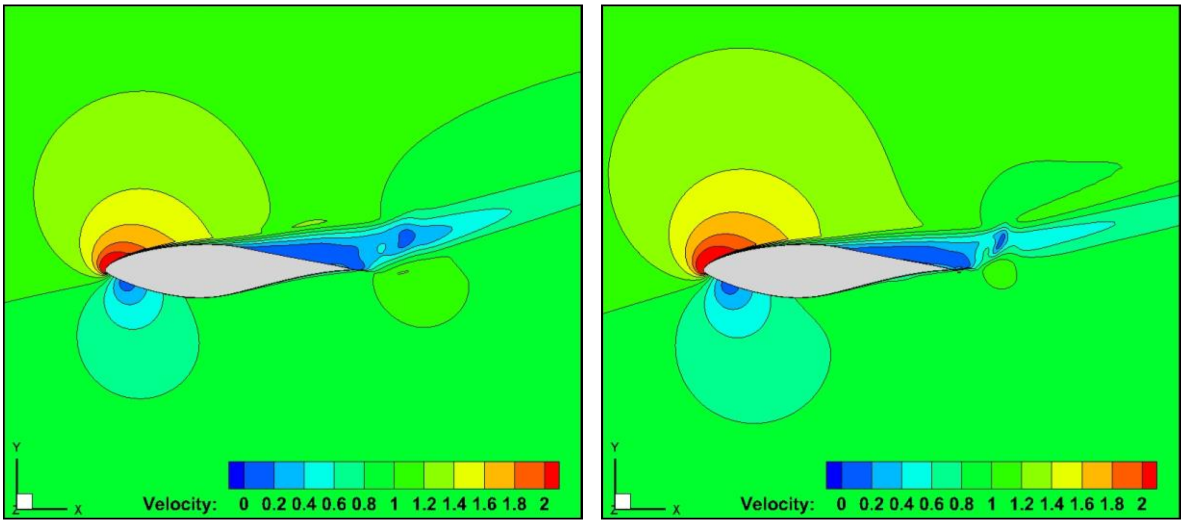
k-ε RNG k-ε

Figure 11. Calculations velocity contours for the angle of attack α = 0°



k-ε RNG k-ε

Figure 5: Calculations velocity contours for the angle of attack α = 9.22°.



k-ε RNG k-ε

Figure 5. Calculations velocity contours for the angle of attack α = 20.15°.

Conclusion

An analysis of two-dimensional incompressible turbulent flow around NREL S809 airfoil was performed using ANSYS CFX CFD code. The analysis was carried out using two turbulence models, k-ε and RNG k-ε, for fixed Reynolds number of 2.0×10^{6}. From the calculations, both k-ε and RNG k-ε models gave near prediction to the experimental values at low angle of attacks, whereas they failed to give good prediction in both pre-stall and post-stall regions. Both models used in this analysis gave more satisfactory predictions of the hydrodynamics characteristics for this airfoil. However, while taking drags and lifts into account, these two models under predict the C_d values and over predict the C_l at higher angles of attack (AOA >= 9.22°), whereas C_d and C_l values of both models lies close to the experimental values at smaller angles of attack (AOA <= 9.22°). The obtained results of k-ε and RNG k-ε models can be improved by increasing the mesh density in the direction of the flow.

Acknowledgements

The present study is supported by LMA, Laboratoire de mécanique appliquée, Université des Sciences et de la technologie - MohamedBoudiaf- Oran, Algérie.

References

ANSYS, Inc. (2017). *ANSYS CFX Reference Guide*. Release 18.2 academic version.

Butterfield, C. P., Scott, G., & Musial, W. (1992). Comparison of wind tunnel airfoil performance data with wind turbine blade data. *Journal of solar energy engineering*, *114*(2), 119-124.

Chang, Y. L., Yang, S. L., & Arici, O. (1996). *Flow field computation of the NREL S809 airfoil using various turbulence models* (No. CONF-960154-). American Society of Mechanical Engineers, New York, NY (United States).

Guerri, O., Bouhadef, K., & Harhad, A. (2006). Turbulent flow simulation of the NREL S809 airfoil. *Wind Engineering*, *30*(4), 287-301.

Hartwanger, D., & Horvat, A. (2008, June). 3D modelling of a wind turbine using CFD. In NAFEMS Conference, United Kingdom.

Launder, B. E. & Spalding, D. B. (1974). The numerical computation of turbulent flows. Computer methods in applied mechanics and engineering, 3(2),269–289.

Wang, H., Zhang, B., Qiu, Q., & Xu, X. (2017). Flow control on the NREL S809 wind turbine airfoil using vortex generators. Energy, 118, 1210-1221.

Wolfe, W. P. and Ochs, S. S., *CFD calculations of S809 aerodynamic characteristics*, AIAA-97-09

Yakhot, V. S. A. S. T. B. C. G., Orszag, S. A., Thangam, S., Gatski, T. B., & Speziale, C. G. (1992). Development of turbulence models for shear flows by a double expansion technique. Physics of Fluids A: Fluid Dynamics, 4(7), 1510-1520.

Author Information

Bouabdellah Abed
Laboratoire de Mécanique Appliquée,
Université des Sciences et de la Technologie-Mohamed Boudiaf- Oran, Algérie.
El Mnaouar, BP 1505, Bir El Djir 31000, Oran - Algérie.
Contact E-mail: bouabdellah.abed@univ-usto.dz

Abdelouahab Benzerdjeb
Laboratoire de Mécanique Appliquée,
Université des Sciences et de la Technologie-Mohamed Boudiaf- Oran, Algérie.
El Mnaouar, BP 1505, Bir El Djir 31000, Oran - Algérie

Habib Achache
University Mohamed Benahamed Oran 2
Boite Postale 1515 El Menouar Oran, Algeria

Abiddine Debab
Departement de Génie Maritime,
Université des Sciences et de la Technologie d'Oran, Algérie.
El Mnaouar, BP 1505, Bir El Djir 31000, Oran - Algérie

Mohamed-Kamel Hamıdou
Laboratoire de Mécanique Appliquée,
Université des Sciences et de la Technologie-Mohamed Boudiaf- Oran, Algérie.
El Mnaouar, BP 1505, Bir El Djir 31000, Oran - Algérie