POLİTEKNİK DERGİSİ

JOURNAL of POLYTECHNIC

ISSN: 1302-0900 (PRINT), ISSN: 2147-9429 (ONLINE)
URL: http://dergipark.org.tr/politeknik

Numerical analysis of NACA 6409 and Eppler 423 airfoils

NACA 6409 ve Eppler 423 kanat profillerinin sayısal analizi

Authors: *Seyhun DURMUŞ[1], Aytekin ULUTAŞ[2]*

ORCID[1]: 0000-0002-1409-7355

ORCID[2]: 0000-0002-5230-7122

To cite to this article: Durmus, S., Ulutas, A., "Numerical analysis of NACA 6409 and Eppler 423 airfoils", ***Journal of Polytechnic***, 26(1): 39-47, (2023).

Bu makaleye şu şekilde atıfta bulunabilirsiniz: Durmus, S., Ulutas, A., "Numerical analysis of NACA 6409 and Eppler 423 airfoils", ***Politeknik Dergisi***, 26(1): 39-47, (2023).

Erişim linki (To link to this article): http://dergipark.org.tr/politeknik/archive

DOI: **10.2339/politeknik.959517**

Numerical Analysis of NACA 6409 and Eppler 423 Airfoils

Highlights

- *Current paper focuses on RANS based turbulence analysis of NACA 6409 and Eppler 423.*
- *θ-Reθ SST was the RANS model that gave the most consistent results with the experimental data.*
- *CFD Simulation results of pressure and velocity fields at different angles of attack are clearly presented.*
- *CFD results with experimental data and XFoil are evaluated in terms of best glide ratio-$(Cl/Cd)_{max}$ and minimum sink-$(Cl^{1.5}/Cd)_{max}$ criteria.*

Graphical Abstract

In this study, by using ANSYS Fluent 19.0, which is a commercial computational fluid dynamics software, a numerical analysis and comparison of the airfoils were carried out by using the lift, drag coefficients as well as pressure and velocity fields by utilizing different RANS based turbulence models of Eppler 423 and NACA 6409 airfoils.

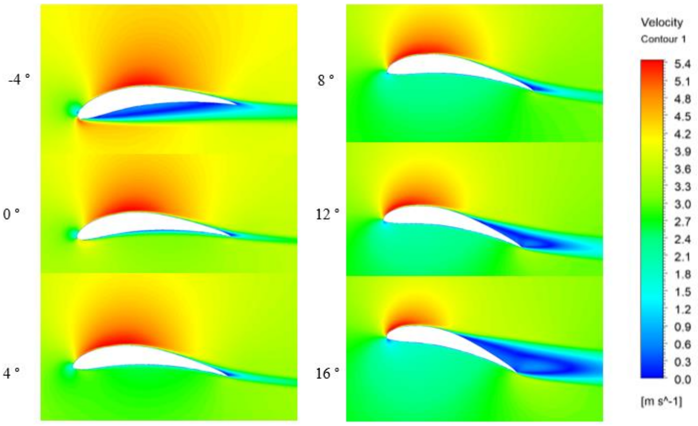


Figure. Representation of velocity contours at different angles of attack around Eppler 423

Aim

The aim of this study is to numerically analyze these airfoils using various turbulence models with ANSYS Fluent 19 and compare the simulation results with experimental data and Xfoil

Design & Methodology

In literature review, it has been found that k-ω SST, k-kl-ω and γ-Reθ transition models have a good prediction in Cl-Cd estimation of drag bucket region, transition flow regime, and laminar separation point

Originality

Testing of various RANS-based turbulence models, demonstrating pressure and velocity fields at different angles of attack, comparing the airfoils in terms of glide rate and minimum sink rate, both experimentally and with XFoil are the paper's key originality.

Findings

θ-Re_θ SST model is the best compared to the other models that predict the lift coefficients and drag coefficients. After the 8-degree angle of attack, the start of flow separation is observed in both airfoils, while the separation trend in NACA6409 is more severe than in Eppler243 at 16-degree angle of attack.

Conclusion

Thin airfoils are generally utilized to minimize drag in cross-country flights, but Eppler 423 will be highly successful in maximizing lift at low speeds. However, maximum glide ratio could be better by usage of NACA 6409. For this reason, NACA 6409 would be much successful for gliding performance while Eppler 423 can ensure better performance in the thermal thanks to better behavior at slow speed.

Declaration of Ethical Standards

The author(s) of this article declare that the materials and methods used in this study do not require ethical committee permission and/or legal-special permission.

Numerical Analysis of NACA 6409 and Eppler 423 Airfoils

Research Article

Seyhun DURMUŞ[1], Aytekin ULUTAŞ[2*]
Edremit School of Civil Aviation, Balikesir University, Balikesir, Turkey.
(Received : 29.06.2021 ; Accepted : 23.08.2021 ; Early View : 01.09.2021)

ABSTRACT

The present study aims to numerically examine two commonly used airfoils that provide high lift at low Reynolds numbers, using various turbulence models with the commercial code ANSYS Fluent 19. Eppler 423 airfoil is widely used in wind turbine blades, wings of gliders and R/C airplanes designed for sport flying and competitions thanks to high lift capability. NACA 6409 airfoil is seen in free-flight model planes. In literature review, it has been found that k-ω SST, k-kl-ω and $\gamma\text{-}Re\theta$ transition models have a good prediction in Cl-Cd estimation of drag bucket region, transition flow regime, and laminar separation point. In addition to RANS models, laminar model was also investigated. In order to ensure the accuracy and precision of numerical investigations; the grid sensitivity study and near wall model were used. The method of decomposition of flow direction vector procedure has been used to provide the advantage of simulating all angle of attacks using only a single mesh. The lift coefficient curves and polar diagrams were formed from the angle of attack of -4 degree up to the stall. Experimental data, XFoil analysis data and numerical analysis data were evaluated in terms of best glide ratio-(Cl/Cd)max and minimum sink-($Cl^{1.5}$/Cd)max criteria. As expected, at high angle of attack, the transition ramp on the upper surface moves forward and the pressure gradient became more adverse. The results showed that fine mesh gives better results than other mesh sizes and transitional γ-Reθ SST model was successful in modelling the transition event than other turbulence models.

Keywords: γ-Reθ SST Turbulence Model, NACA 6409, Epler 423, RANS, CFD.

NACA 6409 ve Eppler 423 Kanat Profillerinin Sayısal Analizi

ÖZ

Bu çalışma, düşük Reynolds sayılarında yüksek taşıma sağlayan yaygın olarak kullanılan iki kanat profilini, ticari kod ANSYS Fluent 19 ile çeşitli türbülans modelleri kullanarak sayısal olarak incelemeyi amaçlamaktadır. Eppler 423 kanat profilini sayısal olarak analiz etmektir. Eppler 423 kanat profili, yüksek taşıma kabiliyeti nedeniyle rüzgâr türbini kanatlarında, çapraz akış fanlarında, planörlerin ana kanatlarında, sportif uçuşlar ve yarışmalar için tasarlanmış radyo kontrollü uçaklarda yaygın kullanılan bir profildir. NACA 6409 kanat profili, serbest uçuş model uçaklarda kullanılır. Literatür taramasında k-ω SST, k-kl-ω ve $\gamma\text{-}Re\theta$ geçiş modellerinin sürükleme poler bölgesinin Cl-Cd tahminlerinde, geçiş bölgesi akış rejiminde ve laminer ayrılma noktasında iyi bir tahmin yeteneğine sahip olduğu görülmüştür. Çalışmada RANS modellere ek olarak laminer model de incelenmiştir. Sayısal çözümlerin doğruluğunu ve hassasiyetini sağlamak için kafes duyarlılık çalışması ve yakın duvar modeli kullanılmıştır. Akış yönü vektörü ayrışması tekniği, sadece tek bir ağ kullanarak tüm hücum açılarını simüle etme avantajını sağlamak için kullanılmıştır. Taşıma katsayısı eğrileri ve kutup şemaları, -4 derece hücum açısından başlanarak stol hücum açısına kadar oluşturulmuştur. Deneysel veriler, XFoil analiz verileri ve nümerik analiz verileri en iyi süzülme oranı- (Cl/Cd)maks ve minimum çöküş-($Cl^{1.5}$/Cd)maks kriterleri açısından değerlendirilmiştir. Beklendiği gibi, yüksek hücum açısında, üst yüzeydeki geçiş rampası ileri doğru hareket eder ve üst yüzeydeki basınç gradyanı negatif hale gelmiştir. Sonuçlar, ince ağın diğer ağ boyutlarına göre daha iyi sonuçlar verdiği; γ-Reθ SST modelinin diğer türbülans modellerine göre geçiş olayını modellemede daha başarılı olduğunu göstermiştir.

Anahtar Kelimeler: γ-Reθ SST Türbülans Modeli, NACA 6409, Epler 423, RANS, HAD.

1. INTRODUCTION

There are a wide range of applications for airfoil producing high lift at low Reynolds numbers. These airfoils are used in wind turbine blades, airfoil selection of cross flow fans, airfoil selection of sailplanes, radio control aircraft designed for fun flights and competitions. Selig and Guglielmo [1] proposed a design philosophy about high-lift airfoils at low Reynolds numbers and suggested that aft loading and concave pressure recovery in airfoils led to an increase in the lift coefficients. Ma and Liu [2] suggested that the thicknesses ratio affect the aerodynamic parameters of high lift producing blades with low Reynolds number, and that the S1223 airfoil with a relative thickness of 5% can be used for the wing tip of wind turbines while S1223 with a relative thickness of 12-13% can be applied to wing root. Rahimi et al. [3] simulated the FX 79-W-15A and NACA 63-430 airfoils using the k-ω SST model and the k-kl-transition models using OpenFOAM software, and claimed that both models were successful in predicting the flow in the

**Corresponding Author*
e-mail :aytekinulutas@hotmail.com

transition zone. Winslow et al. [4] utilized TURNS2D (a RANS solvent configured with a laminar-turbulent transition model) software to recognize low Reynolds-number aerodynamics for micro UAVs using NACA 0009 and NACA 0012 airfoils. It is claimed that over 100,000 Reynolds numbers, for most airfoils, lift and drag characteristics are not constant with the Reynolds number, but under 100,000 Reynolds numbers, the cambered airfoils show better lift and drag characteristics than conventional thick airfoils. Morgado et al. [5] conducted a numerical study on the Selig 1223 and Eppler 387 airfoils with the help of the revised version of the k-kl-ω transition model and the low Reynolds corrected SST k − ω turbulence model; and suggested that the XFoil code yielded better results than the CFD software. It is argued that the ability to detect boundary layer in turbulence models does not always give good results in estimating airfoil data. Coder and Maughmer [6] utilized the XFOIL, PROFIL07 and RANS OVERFLOW software for the analysis of E387, S805, PSU94-097, HTR1555, S903 airfoils, and suggested that although each code showed inconsistency in predicting the lift coefficients, it has been seen that the drag predictive coefficients in the drag bucket matched perfectly with the Experimental results. Collision et al. [7] conducted a numerical study of Eppler E387 and concluded that the k-ω SST and γ-$Re\theta$ transition models predicted the laminar separation and reconnection points with excellent precision. Abobaker et al. [8] proposed a computer-based method based on viscous flow interaction to determine the aerodynamic properties of an airfoil and applied this method to E387 and S8036 airfoils by solving conformal mapping and integral boundary layer equations together. Chen and Bernal [9] measured velocity profiles of the SD 7080 airfoils by two-dimensional Particle Imaging Velocimetry (PIV) at 60,000 and 250,000 Reynolds numbers, and claimed that XFoil made consistent estimates of the formation of a laminar separation bubble at multiple attack angles. Sorenson [10] used the θ-Reθ SST model for various airfoils and claimed that it was successful in predicting Cl-Cd. Aftab et al. [11] investigated various turbulence models on the NACA4415 wing in the Reynolds Number of 120,000 and claimed that Spallart-Allmaras was a successful model in numerical analysis only until the stall developed, although the γ-Re SST model was successful in the entire attack angle. Sarlak et al. [12], in S826 airfoils, at 40,000, 100,000 and 200,000 Reynolds numbers applied the SST k-ω model and found that the SST k-model can predict 3D flow structures. Ahmed et al. [13] conducted an experimental and numerical study on SG6043 airfoils at Reynolds numbers ranging from 38,000 to 200,000, with turbulence levels ranging from 1%, 5% and 10%. It has been reported that increased lift coefficients in parallel with the increase in turbulence levels. Burdet et al. [14] used PROFIL (Eppler Airfoil Design and Analysis Code) and XFOIL software to examine the effect of Reynolds number on the E387 wing profile. It is argued that the design angle of attack depends on the Reynolds number and that the PROFIL data lead to a difference of more than 3 degrees compared to the experimental data for wind turbine blade design. Bai et al. [15] developed an algorithm to solve the fluid structure interaction problem and found that the k– ω SST model was successful in 2D numerical simulations with the block-iterative coupling of the airfoil cross section. Murayama et al. [16] studied the flow around the three-component wings, high-lift device, and flaps. It's argued that Spalart-Allmaras model and Menter's SST model produce similar aerodynamic forces with low attack angles, but the SST model gives better results in the high angle of attack. Dong et al. [17] used the transitional SST model to numerically analyze the FX-63-137 airfoil at 200,000 and 300,000 Reynolds numbers to investigate the laminar separation bubble. In the literature, there are studies on symmetrical profiles, which are widely used in turbine blades and airfoils of UAVs. The aerodynamic properties NACA 0018 were investigated numerically and experimentally with two different aspect ratios [18]. The aerodynamic properties of the humpback whale were numerically examined in the modified NACA 0015 and it was determined that the modified M1 airfoil provided an improvement of 3.81% [19]. When the effect of rib structure on NACA 2412 was examined, it was reported that the stall angle was delayed thanks to the rib structure [20]. In current study, numerical analyses were carried out to estimate the lift and drag coefficients of Eppler 423 and NACA6409 airfoils, which give high lift to low Reynolds numbers, using ANSYS Fluent 19.0, a commercial computational fluid dynamics software. In addition, the display of pressure and velocity contours at different angles of attack is given using the Langtry-Menter 4-equation transitive θ-Reθ SST model, which gives the most successful results among the RANS models.

2. MATERIAL and METHOD

It is possible to measure the aerodynamic performance of airfoils experimentally by using wind tunnel tests or numerically by using computational fluid dynamics software and panel method-based programs such as XFoil or JavaFoil. The experimental data applied in present study were obtained from the wind tunnel test data of the University of Illinois Applied Aerodynamics Group (UIUC) in Urbana-Champaign. There are many models of turbulence that predict boundary layer development and transition regime. In literature review; it is found that the k-kl-ω, low Reynolds addition SST k-w and γ-$Re\theta$ models provided successful simulations for airfoil analysis in the flow regime in the transition region. In current study, all of the RANS-based turbulence models were applied to Eppler 423and NACA 6409 airfoils and the analysis showed that the θ-Reθ SST model is the best compared to the other models that predict the lift coefficients and drag coefficients. The input limit condition of the Θ-Reθ SST model is given below [21,22].

Intermittency:

$$\gamma=1 \tag{1}$$

Transition momentum thickness Reynolds number:

$$f(x) = \begin{cases} 1173.51 - 589.428\text{Tu} + \dfrac{0.2196}{\text{Tu}^2} & \text{if Tu} < 1.3 \\ \dfrac{331.5}{\text{Tu} - 0.5658^{0.671}} & \text{if Tu} > 1.3 \end{cases} \tag{2}$$

Where

$$\text{Tu}=100\frac{\sqrt{2/3k}}{|u\infty|} \tag{3}$$

Fig. 1 shows a graphical sectional view of Eppler 423 and NACA 6409. In ANSYS Fluent simulation, density-based solver which considers compressibility effect was applied instead of pressure-based solver. The Reynolds number is fixed at 200,000. The flow rate of the air is defined by Mach number input. Air characteristics are based on ISA conditions: T = 288.16 K, ρ = 1.225 kg / m3, μ = 1.794 x 10-5.

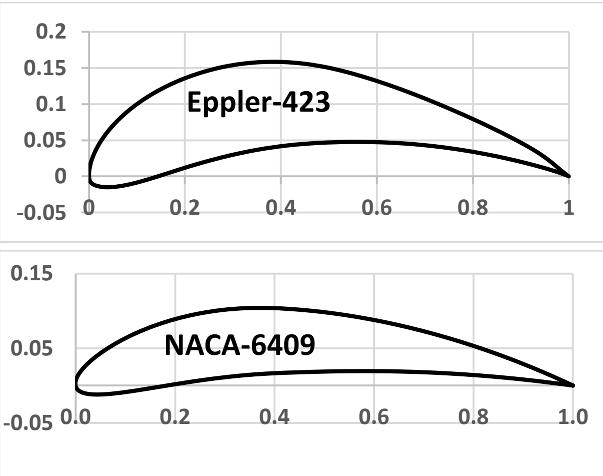


Figure 1. Graphical sectional view of NACA-6409 and Eppler 423

The near wall model is used to reduce the negative effects of the wall on the flow. Dimensionless wall distance y +, developed by Schlichting and Gersten [23], is one of the most significant parameters when evaluating the applicability of wall functions. In Eq.4, y is the absolute distance from the wall; uT is the relative friction velocity and ν is the kinematic viscosity. The first point of the mesh layer is kept at dimensionless wall distance y + <1 to ensure that the boundary layer is correctly captured.

$$y^+ = \frac{yu_T}{\upsilon} \tag{4}$$

In addition, mesh size (number of nodes and number of grid cells) has a large impact on numerical results. Therefore, a mesh sensitivity study was conducted to support the accuracy of the numerical research. Fig. 2 shows the lift coefficient curve for the Eppler 423 at the angle of attack 6 according to the various mesh dimensions, i.e. the number of nodes, Thanks to the mesh sensitivity study, fine mesh was used, which gave parallel results with the experimental data.

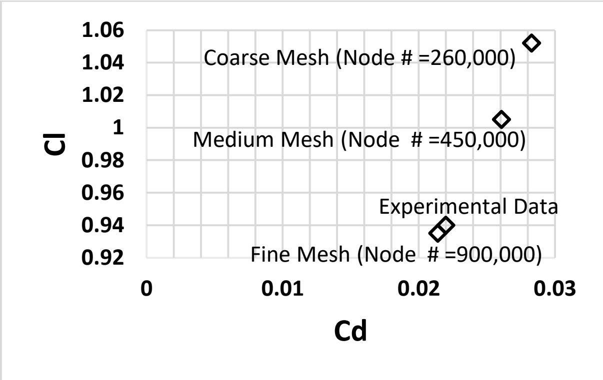
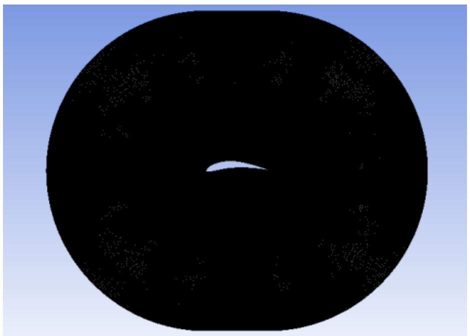


Figure 2. Lift and drag coefficients based on mesh size (node numbers) using the θ-Reθ SST model at 6° angle of attack for Eppler 423

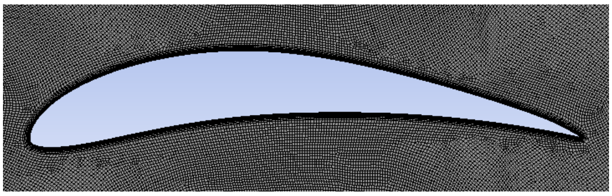
The method of decomposition of the flow direction vector given in Equation (5) and Equation (6) has the advantage of simulating all attack angles using only one mesh. L and D symbolize lift and drag and L 'and D' symbolize aerodynamic force components based on the mesh coordinate system.

$$L = L'\cos(a) - D'\sin(a) \tag{5}$$

$$D = L'\sin(a) + D'\cos(a) \tag{6}$$



(a)



(b)

Figure 3. Mesh (a) Oval shaped control surface (b) Mesh near profile wall

Fig. 3 shows the oval-shaped control surface and the tight mesh on the proximal wall. For the flow area around the profile, the oval computational domain and the pressure farfield are applied as the control boundary condition.

3. RESULTS

In terms of performance, the NACA 6409 is an excellent low speed, floater airfoil, but the large camber severely limits the high-speed performance. Since all analyses are performed at 200,000 Reynolds numbers, an important criterion for the success of the methods is the accurate estimate of the values in the transition region. In addition, an accurate estimation of the drag coefficient in the region where the drag pole forms a laminar bucket is another important criterion.

Fig. 4 shows the estimation of the Cl and Cd coefficients of the laminar model, and various RANS-based turbulence models. The Langtry-Menter 4-equation transitive θ-Reθ SST model, which gives the best results at different angles of attack, was utilized as a turbulence model for Eppler 423 and NACA 6409 analysis at different angles of attack.

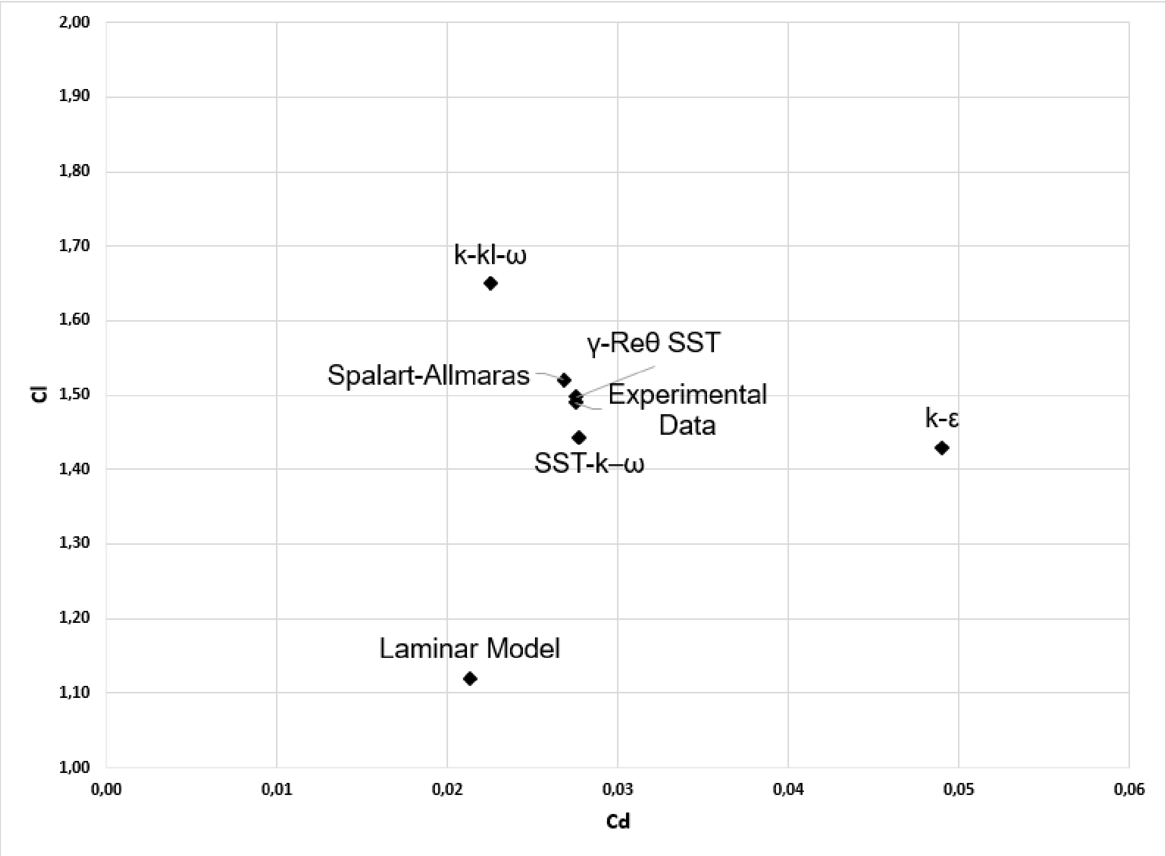


Figure 4. Cl and Cd estimates of the laminar model and various RANS-based turbulence models at 6º attack angle of Eppler 423 airfoil

Drag polar diagrams and lift coefficient curves are shown in Fig. 5 (a, b) and Fig. 6 (a, b) for Eppler 423 and NACA 6409 relatively, including experimental data, Xfoil and Fluent software estimates. Lift coefficient curves and drag polar diagrams were generated from -4 degrees of attack angle to 10 degrees of attack angle when pre-stall occurred. The movement in transition zone created a transition ramp on the upper surface in the drag bucket area. As the angle of attack increases, the transition zone moves forward and the upper surface pressure gradient becomes more adverse. ANSYS Fluent, a computational fluid dynamics code, has produced more compatible results with experimental data than XFoil data.

At negative attack angles, air flow in the lower region of the airfoil was virtually absent, whereas at high attack angles (angle of attack> 10 °), adverse pressure gradients at the trailing edge caused flow separation and loss of lift. The notable differences can be seen at 16 degrees, that is the flow separation reached almost to the leading edge in NACA 6409.

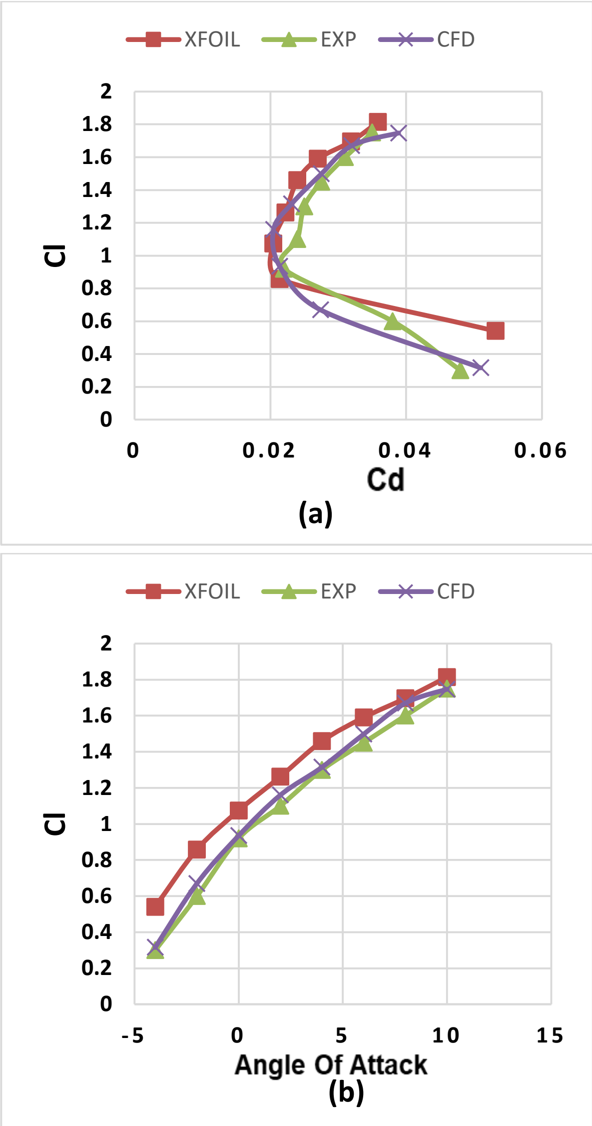


Figure 5. a) drag polar curve and b) lift curve for Eppler 423 airfoil including experimental data and, Xfoil and Fluent software estimates

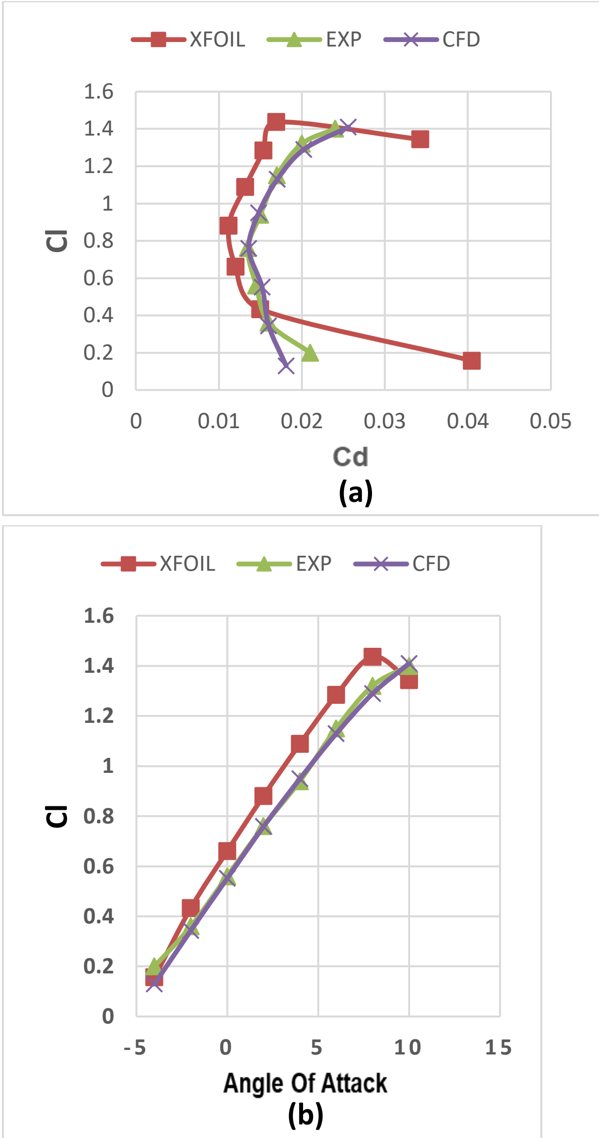


Figure 6. a) drag polar curve and b) lift curve for Eppler 423 airfoil including experimental data and, Xfoil and Fluent software estimates

Fig. 7 and Fig. 8 shows the pressure contours around Eppler 423 and NACA 6409 at different angles of attack from -4 degrees to 16 degrees. It is possible to say that the low-pressure area in the upper region of the airfoil (dark blue region), that is, the center of pressure shifts towards the leading edge, and at high attack angles it causes losses in lift relatively high pressures from the trailing edge at the top of the airfoil. According to figures, low pressure area is more stable in Eppler 423 than NACA 6409 at high angles of attack while low pressure area is larger and stronger in NACA 6409 at low angles of attack.

Fig. 9 and Fig. 10 shows the velocity contours around Eppler 423 and NACA 6409 at different angles of attack from -4 degrees to 16 degrees. The best glide ratio or lift/drag ratio occurs at high (Cl/Cd)max minimum friction conditions, while minimum sink occurs under (Cl1.5/Cd)max conditions. The minimum drag point is found by drawing a straight line from the origin to the point with the maximum slope on the Cl-Cd curve in the drag bucket region. High endurance factor or minimum sink rate factor occurs at the $(Cl^{1.5}/Cd)_{max}$ point.

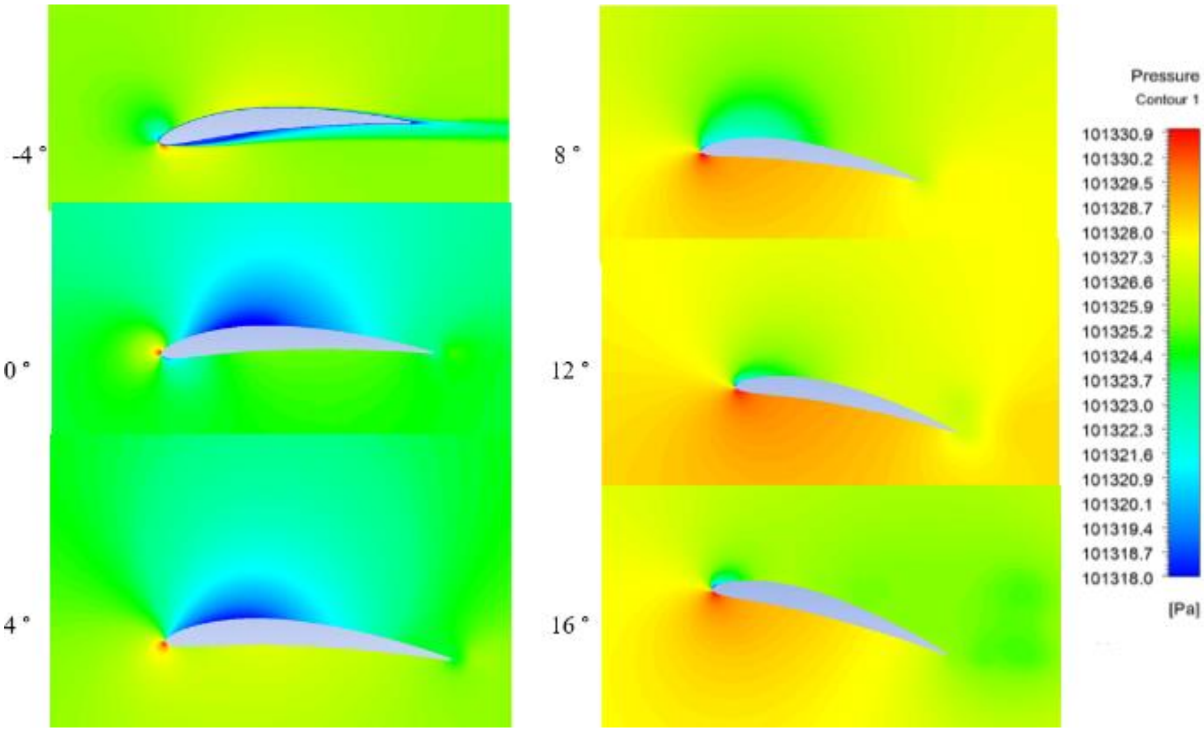


Figure 7. Display of pressure contours around the Eppler 423 airfoil at -4, 0.4, 8, 12 and 16 angles of attack

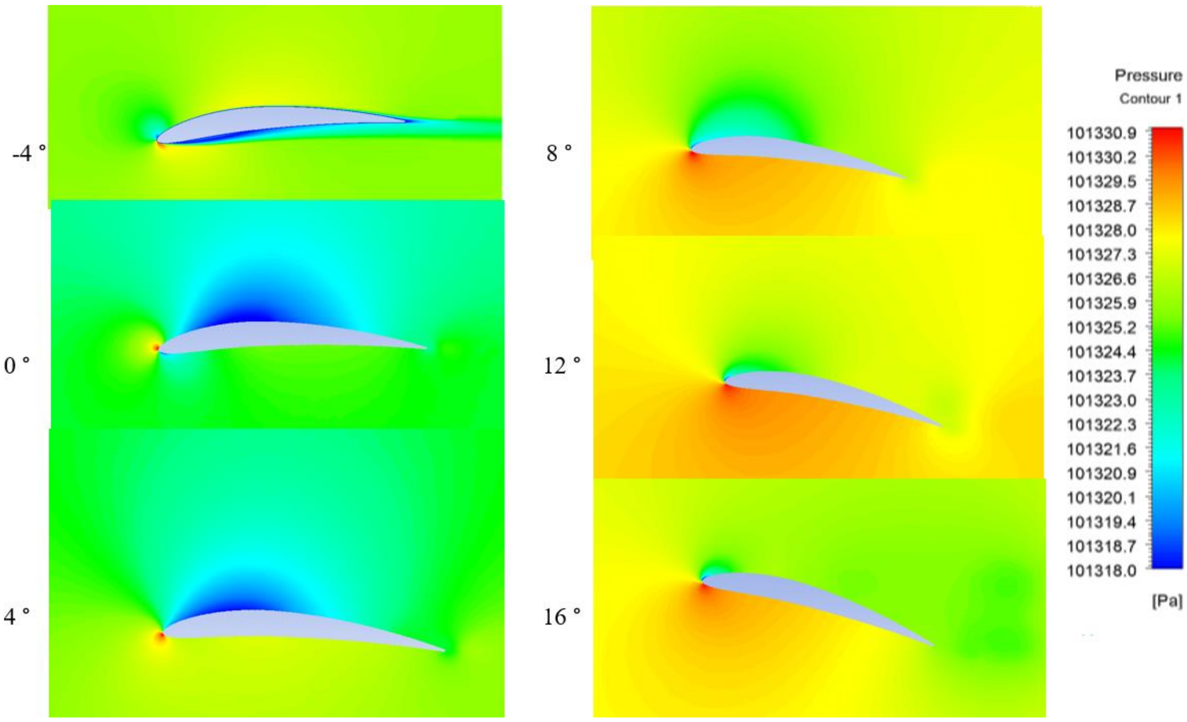


Figure 8. Display of pressure contours around the NACA6409 airfoil at -4, 0.4, 8, 12 and 16 angles of attack

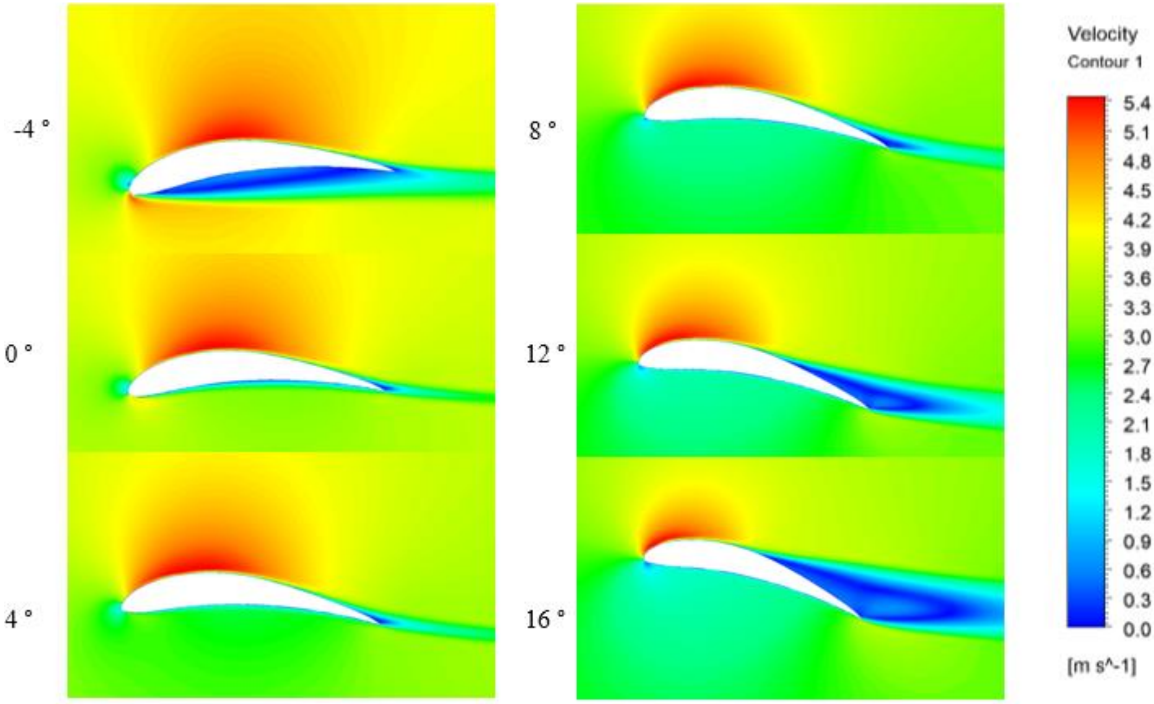


Figure 9. Representation of velocity contours at different angles of attack around Eppler 423

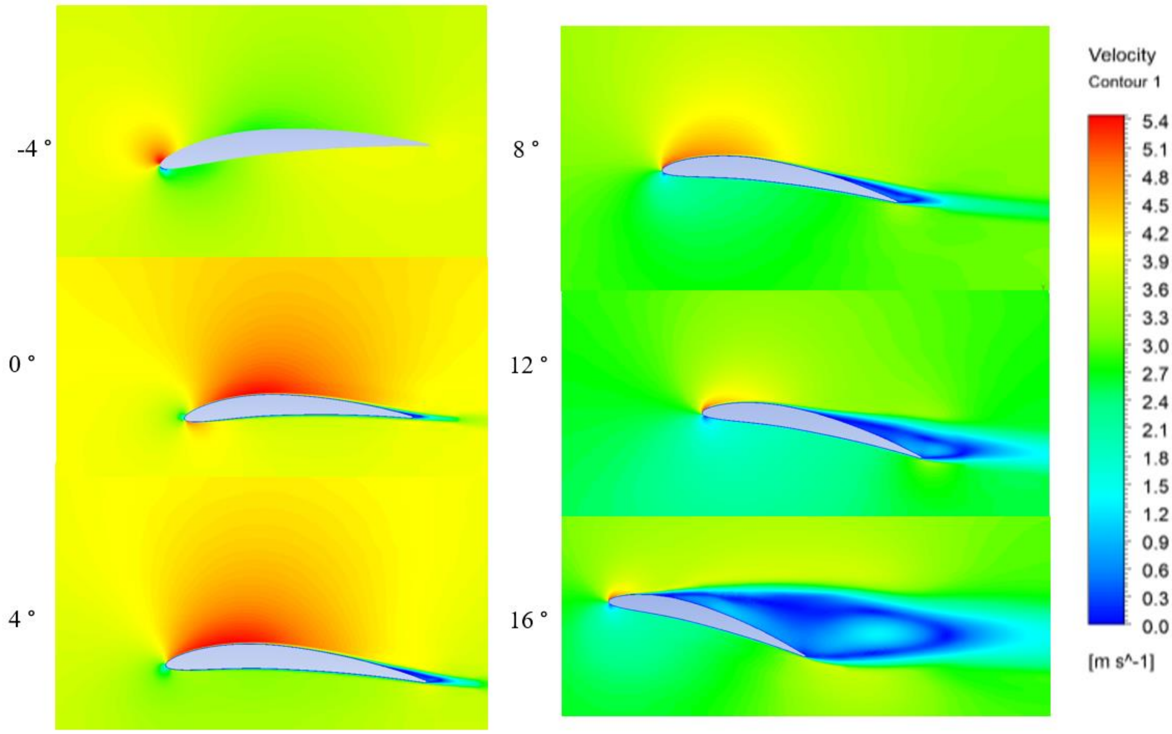


Figure 10. Representation of velocity contours at different angles of attack around NACA6409

Fig. 11 shows the comparison of Xfoil and Fluent solutions of Eppler 423 and NACA 6409 in terms of best glide ratio $(Cl/Cd)_{max}$ and minimum sink rate $(Cl^{1.5}/Cd)_{max}$ performances. As it can be understood from analyses results, Ansys Fluent gave more consistent results than the Xfoil code.

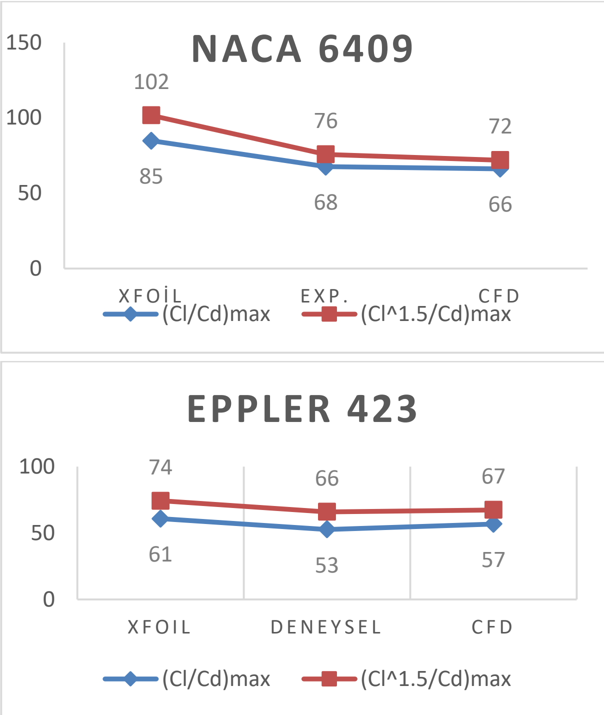


Figure 11. Comparison of Xfoil and Fluent analyses of NACA 6409 and Eppler 423 in terms of best glide ratio $(Cl/Cd)_{max}$ performance and minimum sink rate $(Cl^{1.5}/Cd)$max performance

4. CONCLUSION

Many airfoils designed for low Reynolds numbers are far from full manufacturability and are very difficult to implement in real life due to excessive thinness on the trailing edge, excessive camber etc. In addition, most experimental studies are affected by test conditions and experimental mechanism. Therefore, determining the airfoil for the design with Ansys Fluent provides realistic results while accelerating and facilitating the design. Although the airfoil is difficult to full manufacture, such as the S1223, this airfoil can be used very comfortably in an unmanned glider, especially designed to endurance flights. Thin airfoils are generally utilized to minimize drag in cross-country flights, but Eppler 423 is highly successful in maximizing lift at low speeds. However, maximum glide ratio could be better by usage of NACA 6409. For this reason, NACA 6409 would be much successful for gliding performance while Eppler 423 can ensure better performance in the thermal thanks to better behavior at low speeds.

DECLARATION OF ETHICAL STANDARDS

The authors of this article declare that the materials and methods used in this study do not require ethical committee permission and/or legal-special permission

AUTHORS' CONTRIBUTIONS

Seyhun DURMUŞ: Performed the simulations, made evaluations, wrote whole manuscript
Aytekin ULUTAŞ: Supported to correct formal errors

CONFLICTS OF INTEREST

No conflict of interest was declared by the authors.

REFERENCES

[1] Selig, M.S., Guglielmo, J.J.,"High-lift low Reynolds number airfoil design", ***Journal of Aircraft***, 34(1):72-79, (1997).

[2] Ma, R., Liu, P.,"Numerical simulation of low-Reynolds-number and high-lift airfoil S1223". In Proceedings of the ***World Congress on Engineering***, London, U.K., 2:1-3, (2009).

[3] Rahimi, H., Medjroubi, W., Stoevesandt, B., Peinke, J.,"2d numerical investigation of the laminar and turbulent flow over different airfoils using OpenFOAM". Journal of Physics: Conference Series, ***IOP Publishing***, Oldenburg, Germany, 555(1): 012070, (2014).

[4] Winslow, J., Otsuka, H., Govindarajan, B., Chopra, I.,"Basic Understanding of Airfoil Characteristics at Low Reynolds Numbers (104–105)", ***Journal of Aircraft,*** 55(3):1050-1061, (2018).

[5] Morgado, J., Vizinho, R., Silvestre, M.A.R., Páscoa, J.C.,"XFOIL vs CFD performance predictions for high lift low Reynolds number airfoils", ***Aerospace Science and Technology***, 52:207-214, (2016).

[6] Coder, J.G., Maughmer, M.D.,"Comparisons of theoretical methods for predicting airfoil aerodynamic characteristics", ***Journal of Aircraft***, 51(1):183-191, (2014).

[7] Collison, M.J., Harley, P.X., di Cugno, D.,"Experimental and Numerical Investigation of Transition Effects on a Low Reynolds Number Airfoil", In ***ASME Turbo Expo 2017:*** Turbomachinery Technical Conference and Exposition: V02BT41A008-V02BT41A008, American Society of Mechanical Engineers Digital Collections, U.S., 1-11, (2017).

[8] Abobaker, M., Petrovic, Z., Fotev, V., Toumi, N., Ivanovic, I.,"Aerodynamic characteristics of low Reynolds number airfoils", ***Technical Gazette***, 24(1):111-118, (2017).

[9] Chen, W., Bernal, L.,"Design and performance of low Reynolds number airfoils for solar-powered flight", In 46th AIAA ***Aerospace Sciences Meeting And Exhibit, Reno***, Nevada, U.S.,316, (2008).

[10] Sørensen, N.N., "Airfoil computations using the γ-Reθ model", Danmarks Tekniske Universitet, Risø Nationallaboratoriet for Bæredygtig ***Energi***, Roskilde, Denmark, 7-18 (2009).

[11] Aftab, S.M.A., Rafie, A.M., Razak, N.A., Ahmad, K.A.,"Turbulence model selection for low reynolds number flows", ***PloS One***, 11(4): e0153755, (2016).

[12] Sarlak, H., Nishino, T., Sørensen, J.N.,"URANS simulations of separated flow with stall cells over an NREL S826 airfoil", In ***AIP Conference Proceedings,***1738(1): 030039, AIP Publishing, Rhodes, Greece, (2016).

[13] Ahmed, M.R., Narayan, S., Zullah, M.A., Lee, Y.H.,"Experimental and numerical studies on a low Reynolds number airfoil for wind turbine blades", ***Journal of Fluid Science and Technology***, 6(3):357-371, (2011).

[14] Burdett, T., Gregg, J., Van Treuren, K.,“An examination of the effect of Reynolds number on airfoil performance”, In ASME 2011 ***5th International Conference on Energy Sustainability***, Washington, DC, 2203-2213, (2011).

[15] Bai, Y., Sun, D., Lin, J., Kennedy, D., Williams, F.,“Numerical aerodynamic simulations of a NACA airfoil using CFD with block-iterative coupling and turbulence modelling”, ***International Journal of Computational Fluid Dynamics***, 26(2):119-132, (2012).

[16] Murayama, M., Lei, Z., Mukai, J., Yamamoto, K.,“CFD validation for high-lift devices: Three-element airfoil”, ***Transactions of the Japan Society for Aeronautical and Space Sciences***, 49(163):40-48, (2006). .

[17] Dong, H., Xia, T., Chen, L., Liu, S., Cui, Y. D., Khoo, B. C., & Zhao, A. (2019). Study on flow separation and transition of the airfoil in low Reynolds number. ***Physics of Fluids***, 31(10), 103601

[18] Tanürün,, H. E., İsmail, A. T. A., CANLI, M. E., & Adem, A. C. I. R. (2020). Farklı açıklık oranlarındaki NACA-0018 rüzgâr türbini kanat modeli performansının sayısal ve deneysel incelenmesi. ***Politeknik Dergisi,*** 23(2), 371-381.

[19] Tanürün, H. E., & Adem, A. C. I. R. (2019). Modifiye edilmiş NACA-0015 kanat yapısında tüberkül etkisinin sayısal analizi. ***Politeknik Dergisi,*** 22(1), 185-195.

[20] Tanürün,, H. E., AKIN, A. G., & Adem, A. C. I. R. Rüzgâr Türbinlerinde Kiriş Yapısının Performansa Etkisinin Sayısal Olarak İncelenmesi. ***Politeknik Dergisi,*** 1-1.

[21] Langtry, R.B., Menter, F.R.,“Correlation-based transition modeling for unstructured parallelized computational fluid dynamics codes”, ***AIAA Journal***, 47(12):2894-2906, (2009).

[22] Menter, F.R., Langtry, R., Völker, S.,“Transition modelling for general purpose CFD codes”, ***Flow, Turbulence and Combustion,*** 77(1-4):277-303, (2006).

[23] Schlichting, H., Gersten, K., **Boundary-layer theory 9th ed.** , Springer–Verlag, Heidelberg,304-319 (2016).