iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 8 **Issue:** VIII **Month of publication:** August 2020

DOI: https://doi.org/10.22214/ijraset.2020.31159

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 8 Issue VIII Aug 2020- Available at www.ijraset.com

CFD Analysis on NACA 63-215 Airfoil using ANSYS Fluent for Wind Turbine

Surya Vamshi Penumarthi
Department of Mechanical Engineering, Vasavi College of Engineering, Hyderabad, India.

***Abstract:** Analysis of an airfoil (NACA 63-215) is done with the help of Ansys Fluent 18.1 (Two-dimensional). The main agenda of this work is to carry out analysis with a suitable viscous model and compare the results of the NACA 63-215. Details of the generation of geometry and mesh are given in section II. A total of 19581 nodes are present in this model. The main viscous model used for this work is k-ω SST (section III). The computation model is solved for different attack angles varying from -16^o to +16^o with an increment of 2^o (section IV). In results (section V), the Coefficient of lift and drag are plotted with respect to attack angles. In section VI, the Coefficient of lift and drag of NACA 63-215 are compared with practical values and the conclusion is made.*

***Keywords:** CFD, NACA 63-215, airfoil, incompressible, flow, Spalart-Allmaras, K-ω SST*

NOMENCLATURE

C	Chord length (m)
α	The angle of attack (deg)
D	Drag Force (N)
L	Lift Force (N)
V_∞	Free Stream Velocity (m^2/s)
ρ	Air density (kg/m^3)
μ	Dynamic viscosity (Ns/m^2)
Re	Reynolds number
C_L	Coefficient of lift
C_D	Coefficient of drag
G_ν	Production of turbulent viscosity
Y_ν	Destruction of turbulent viscosity that occurs in the near-wall region due to wall blocking and viscous damping.
$\sigma_{\tilde{\nu}}$	Constant
C_{b2}	Constant

Kinematic viscosity ν

$S_{\tilde{\nu}}$ User-defined source term

$\tilde{G}_k$ Generation of turbulence kinetic energy due to mean velocity gradients

G_ω Generation of ω

Γ_k Effective diffusivity of k Effective diffusivity of Dissipation of k due to turbulence

Dissipation of due to turbulence ω

D_ω Cross-diffusion term Γ_ω ω

S_k User-defined source term S_ω User-defined source term Y_k

Y_ω

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 8 Issue VIII Aug 2020- Available at www.ijraset.com

I. INTRODUCTION

An increase in the demand for power supply must be met by generating more power. By taking pollution into account, we should move towards renewable energy. One of these energy sources is wind. Nevertheless, setting up a windmill is not sufficient, we need to make sure the efficiency in capturing the wind energy is high. The shape of the wind turbine blade plays a critical role where a high ratio of C_L (coefficient of lift) to C_d (coefficient of drag) is desirable. Thus, the selection of airfoil for the wind turbine blade places a vital role. In this project, the analysis of NACA 63-215 is performed.

Computational Fluid Dynamics (CFD) is a simulation tool which is used for solving complex fluid flow problems. This is a fast and reliable method to analyse airfoils. The CFD contains three main elements which are Pre-processor, Solver and Post-Processor. The solution of these simulations depends upon the viscous model selected. Different models have their respective governing equations which are used in the solver. Some of these models are Spalart-Allmaras, K-ε, k-ω SST, Transient SST, Reynolds Stress, etc. It is the job of the engineers to select the right viscous model for a given problem.

When fluid is flowing over an airfoil, two forces are produced (Lift force and Drag force). Drag force is the component of resultant force parallel to chord length whereas lift force is the component of resultant force in the direction perpendicular to chord length. For an airfoil high coefficient of lift with a low coefficient of drag is desirable.

II. GEOMETRY AND MESH SETUP

The geometry of the fluid domain is set up by DesignModeler. The Cartesian coordinates of NACA 65-215 aerofoil have been imported from the airfoiltools.com data file. The airfoil has a chord length of 1m. The C-type fluid domain of 15C and 25C has been constructed in this model. To create a mesh for an accurate solution, the domain has been divided into four surfaces by drawing a horizontal line through the airfoil and vertical line at the trailing edge of the airfoil.

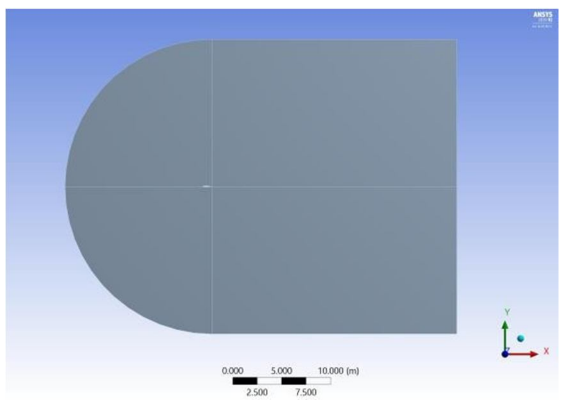


Fig1: Fluid Domain

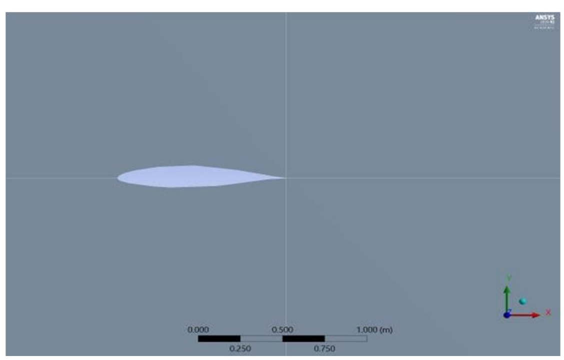


Fig2: NACA 65-215 Airfoil

Face mesh is carried out on the fluid domain. To get accurate results and take up fewer computation resources, fine mesh is required around the airfoil and coarse mesh at the other boundaries. So, sizing is done on the edges. The bias factor (1.15) has been used to provide high mesh density around the airfoil for greater accuracy and better flow visualization. The fluid domain has a 19581 number of nodes.

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 8 Issue VIII Aug 2020- Available at www.ijraset.com

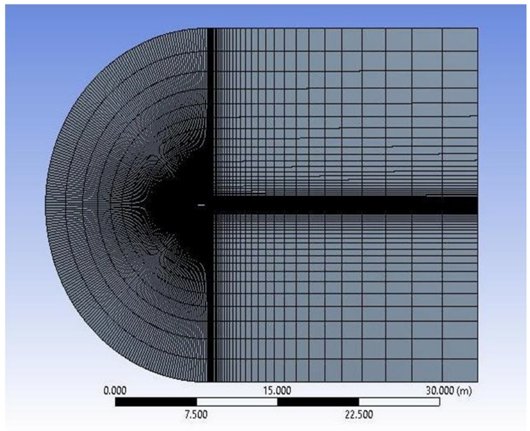


Fig3: Mesh generation of the fluid domain

III. FLOW SPECIFICATIONS

The pressure-based solver has been used as air is considered as incompressible for the following conditions. Here the air was considered to have a constant velocity ($V\infty$) of 7.3m/s (Inlet condition) and the density of the air was taken 1.225 kg/m^3. The temperature of the air is taken as 288.16 K (Dynamic viscosity of 1.7894e-05 kg/m-s). From airfoiltools.com, Reynold's number for this model is 500,000. The turbulence intensity has been set as 5%.

Boundary Condition at airfoil is set as a stationary wall with a no-slip condition. Whereas at the outlet, gauge pressure is given value of 0 pascals. The coupled scheme is used with the momentum of second-order upwind. The limit for the Convergence is 10^{-4}.

IV. SELECTION OF TURBULENT MODEL

As there is no universal turbulent model for all CFD problems, we have to select a suitable model according to the problem requirement and convergence. Spalart-Allmaras (SA) is a one-equation turbulence model that has been developed specifically for aerodynamic flows such as transonic flow over airfoils.

$$\frac{\partial}{\partial t}(\rho\tilde{\nu})+\frac{\partial}{\partial x_i}(\rho\tilde{\nu}u_i) = G_\nu+\frac{1}{\sigma_{\tilde{\nu}}}\left[\frac{\partial}{\partial x_j}\left\{(\mu+\rho\tilde{\nu})\frac{\partial\tilde{\nu}}{\partial x_j}\right\}+C_{b2}\rho\left(\frac{\partial\tilde{\nu}}{\partial x_j}\right)^2\right]-Y_\nu+S_{\tilde{\nu}}$$

This is the equation of the Spalart-Allmaras turbulent model. Spalart-Allmaras is not memory-intensive and has good convergence. However, for NACA 65-215, the solution was not converged as shown in Fig4. This is because residuals values had increased with the number of iterations as shown in fig5. In this case, the angle of attack is 0^o.

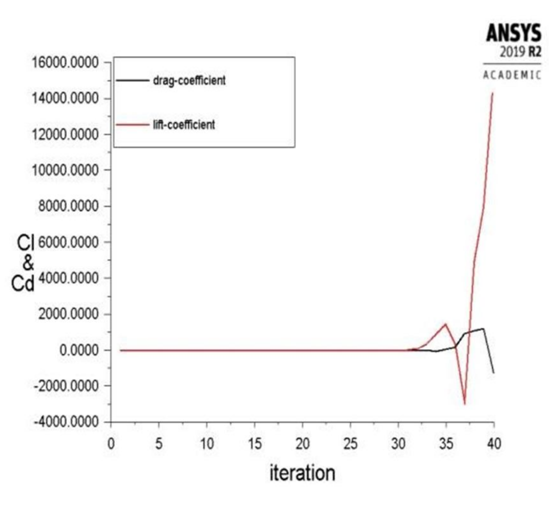


Fig4: C_L & Cd plot for SA model at 0^o AOA

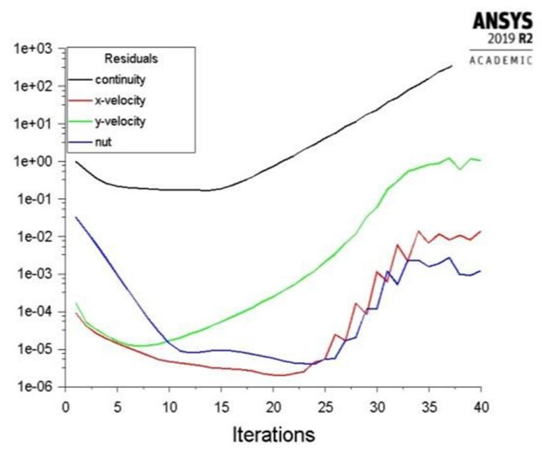


Fig5: Scaled Residuals plot for SA model at 0^o

K-ω is also a well-used turbulent model. K-ω SST has become a popular choice in aerospace applications where the flow is deemed too complex for Spalart-Allmaras. This model has 2 equations.

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left(\Gamma_k\frac{\partial k}{\partial x_j}\right)+\tilde{G}_k-Y_k+S_k$$

And

$$\frac{\partial}{\partial t}(\rho\omega)+\frac{\partial}{\partial x_i}(\rho\omega u_i)=\frac{\partial}{\partial x_j}\left(\Gamma_\omega\frac{\partial\omega}{\partial x_j}\right)+G_\omega-Y_\omega+D_\omega+S_\omega$$

Although this model is considered as difficult to converge and requires high computational power compared to the Spalart-Allmaras model. For this study, K-ω SST is used as it converges better (Fig6).

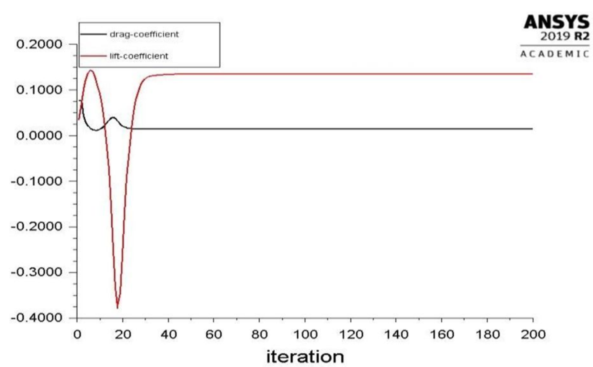


Fig6: C_L & C_d plot for K-ω SST model at 0^o

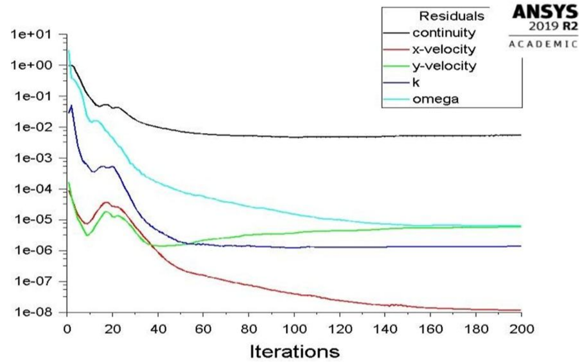


Fig7: Residuals plot for K-ω SST model at 0^o

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 8 Issue VIII Aug 2020- Available at www.ijraset.com

V. SIMULATION OUTCOMES

Simulation is carried out at various angles of attack from -16^o to 16^o with an increment of 2^o. Velocity contours at -10^o, 0^o and 10^o are shown in the figures below. Blueish regions indicate the low velocity of air and Reddish regions indicate the high velocity of air.

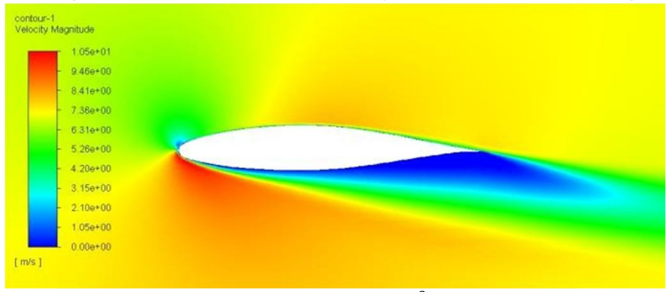


Fig8: Velocity contour at -10^o angle of attack

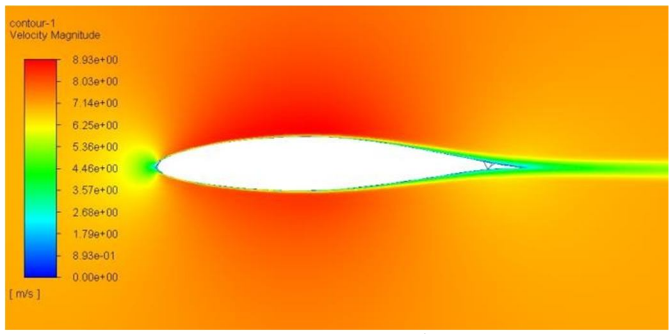


Fig9: Velocity contour at 0^o angle of attack

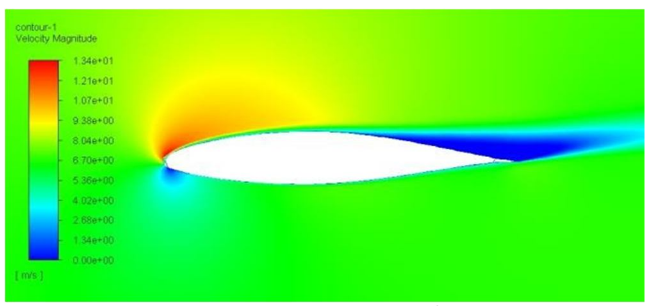


Fig10: Velocity contour at 10^o angle of attack

Similarly, Pressure contours are shown in Figures 11 to 13. Analogous to velocity contours, in pressure contours, Blueish regions indicate low pressure and Reddish regions indicate high pressure. These figures can be validated by Bernoulli's Theorem i.e., Regions with high pressure (Red) has low velocity (Blue) and vice versa.

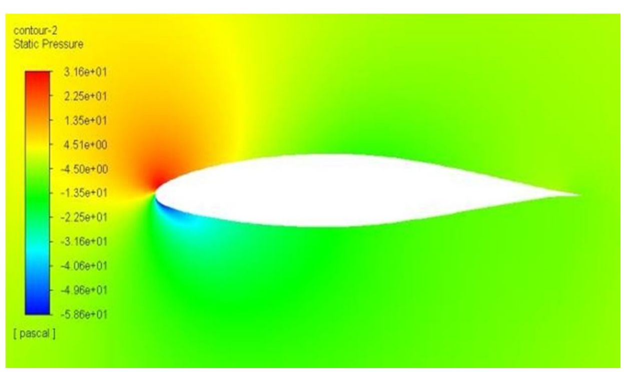


Fig11: Pressure contour at -10^o angle of attack

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 8 Issue VIII Aug 2020- Available at www.ijraset.com

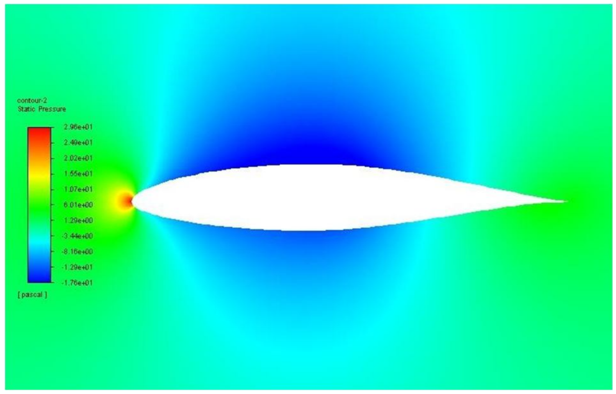


Fig12: Pressure contour at 0^o angle of attack

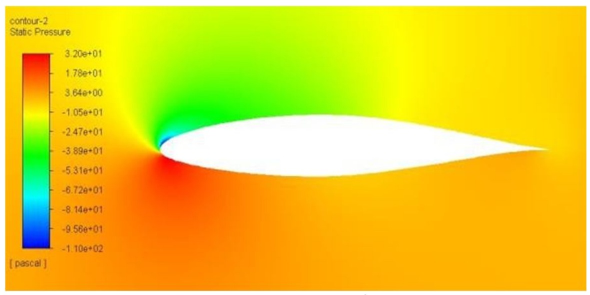


Fig13: Pressure contour at 10^o angle of attack

VI. RESULTS

As NACA 65-215 is a cambered airfoil, The Coefficient of lift at 0^o is not equal to zero. Also, the values of the Coefficient of lift and drag are not symmetrical in the positive and negative angle of attacks. The Coefficient of lift and drag increases with the angle of attack as shown in figures 14 and 15. Stall condition comes near 11^o angle of attack.

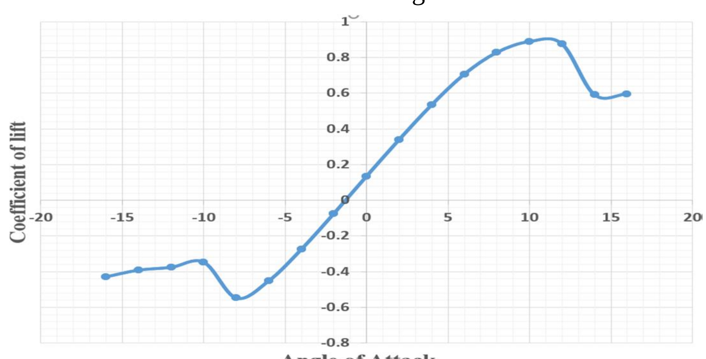


Fig14: Coefficient of lift for the various angle of attack

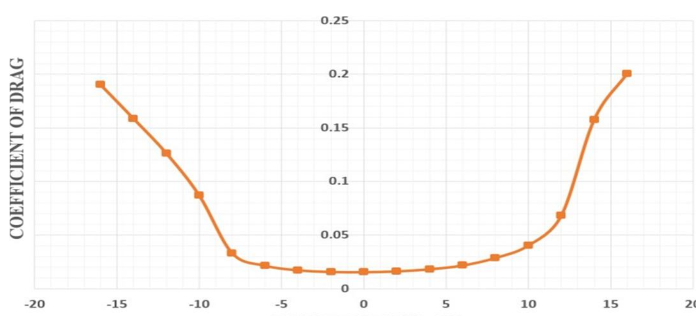


Fig15: Coefficient of drag for the various angle of attack

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 8 Issue VIII Aug 2020- Available at www.ijraset.com

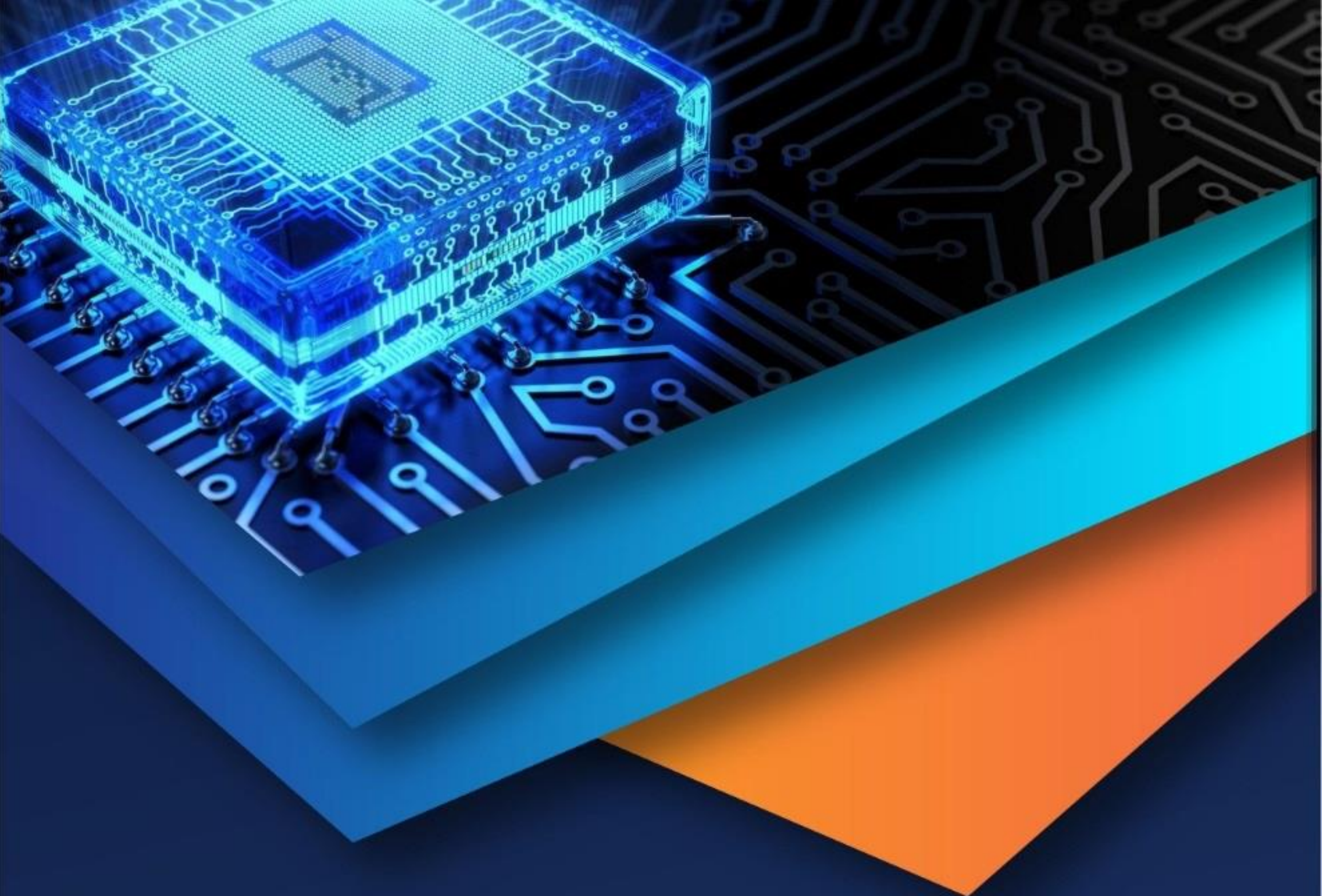
VII. CONCLUSION

Although the Spalart-Allmaras turbulent model is best suitable for aerodynamic flows over an airfoil. In some cases, it fails to converge the solution. K-ω SST turbulent model is an alternative option. Comparing the values obtained with data from airfoiltools.com, K-ω SST turbulent model is a viable option. Keeping in mind that it requires more computational resources.

NACA 65-215 has a C_L about 0.9 near 11^o angle of attack. So, this airfoil shape can be used in blades for Horizontal axis wind turbine (HAWT).

REFERENCES

[1] N. Manikandan, B. Stalin, Design of Naca63215 Airfoil for a Wind Turbine, Volume 10, Issue 2 (Nov. - Dec. 2013), PP 18-26
[2] Liyana Kharulaman, Abdul Aabid, Fharukh Ahmed Ghasi Mehaboobali, Sher Afghan Khan, "Research on flow for NACA 2412 Airfoil using Computational Fluid Dynamics Method," Volume-9 Issue-1, October 2019
[3] Amit Kumar Saraf, Mahendra Pratap Singh, Tej Singh Chouhan, "Aerodynamic analysis of NACA0012 airfoil using CFD," Volume- 5, Issue-12, Dec.-2017, PP 21-25.
[4] Peter J. Schub and Richard J. Crossley, " Wind Turbine Blade Design," 6 September 2012, PP 3425-3449.
[5] Bertagnolio, F.; Sørensen, Niels N.; Johansen, Jeppe; Fuglsang, P, "Wind turbine airfoil catalogue," 2001 PP 9-12
[6] Pradip Niyogi, Sunil Kumar Chakrabartty, Manas Kumar Laha, "Introduction to Computational Fluid Dynamics," 1st Edition 2009.
[7] K. S. Patel, S. B. Patel, U. B. Patel, and P.
A. P. Ahuja, "CFD Analysis of an Aerofoil,", 2015, Volume- 3, Issue 3, PP. 154–158
[8] S. Merryisha and P. Rajendran, "CFD Validation of NACA 2412 Airfoil," 2019
[9] D. C. Eleni, T. I. Athanasios, and M. P. Dionissios, "Evaluation of the Turbulence Models for the Simulation of The Flow over a National Advisory Committee for Aeronautics (NACA) 0012 Airfoil," 2012, Volume- 4, Issue 3, PP. 100-111.
[10] A. K. Saraf, M. P. Singh, and T. E. J. S. Chouhan, "Aerodynamics Analysis of NACA 0012 Airfoil using CFD," 2017, Volume- 5, Issue 12, PP. 21–25.



10.22214/IJRASET

INDEX COPERNICUS 45.98

IMPACT FACTOR: 7.129

TOGETHER WE REACH THE GOAL

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY