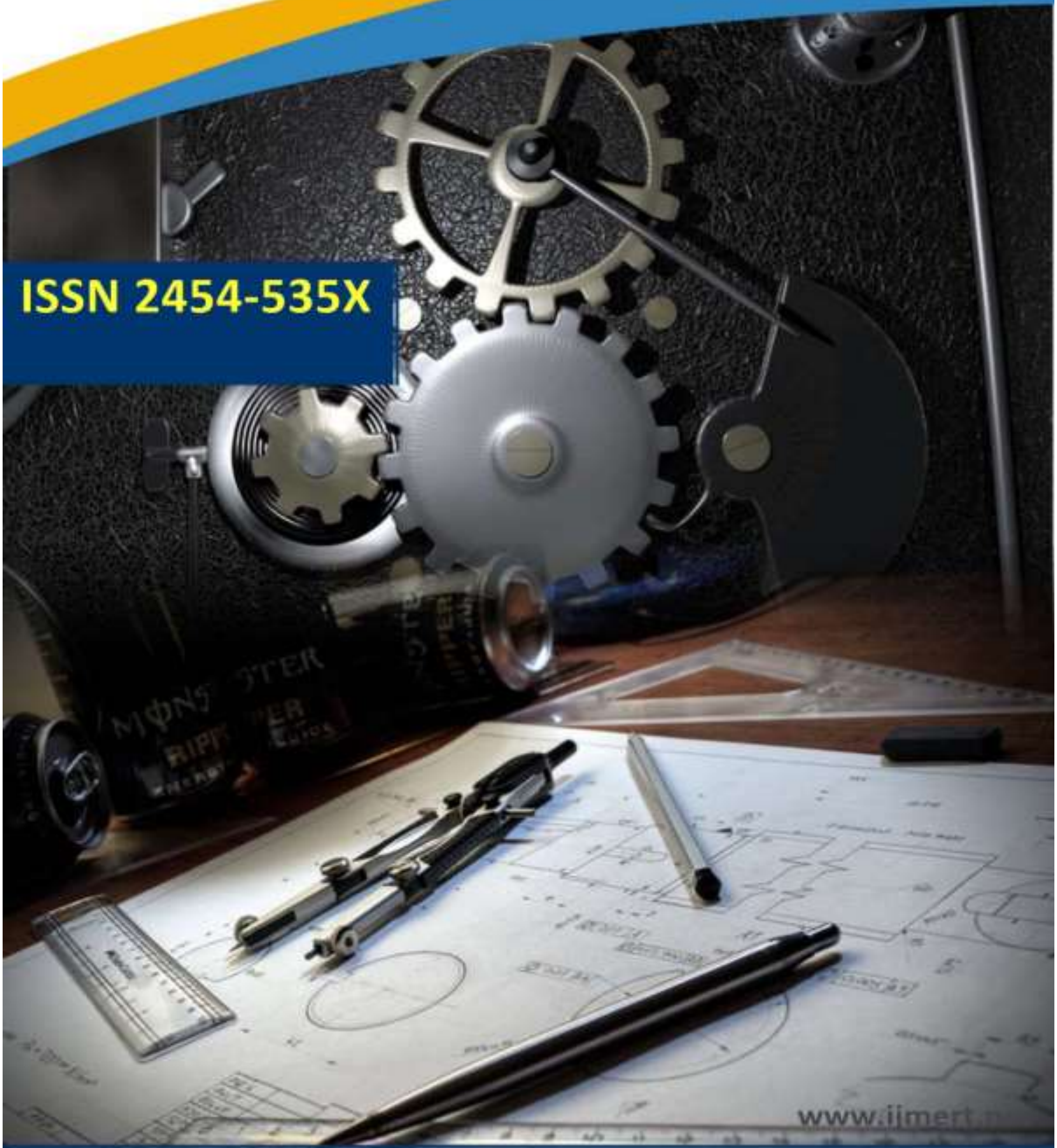
International Journal of
Mechanical Engineering Research and Technology



ISSN 2454-535X

Email ID: info.ijmert@gmail.com or editor@ijmert.net

Int. J. Mech. Eng. Res. & Tech 2019

International Journal of Mechanical Engineering Research and Technology

ISSN 2454 – 535X www.ijmert.com
Vol. 11, Issue. 1, Jan 2019

COMPUTATIONAL FLUID ANALYSIS OFDIFFERENT AIRFOIL SHAPES

Mr. M. Vijaya Kumar, Mr.G Balakrishna Reddy, Mr.K. Satya Vara Prasad, Ms. D.Mahalakshmi

Abstract
This project aims to investigate the design of air foils and determine the flow parameters, as well as the lift and drag forces acting on various air foils such as NACA-0012, 495 and 540, under different viscosity conditions. Air foils are a critical component of aircraft, including passenger planes, jet planes, and helicopters. They play a significant role in determining whether the lift force is sufficient to support the weight of the aircraft and the amount of drag force applied to the aircraft. This project employs three different profile shapes of air foil, including symmetric and asymmetrical designs, to analyse their lift and drag coefficients and their impact on the airfoil's performance.

Keyword: Computational fluid dynamics (CFD), airfoil, aerodynamic coefficients, lift, drag, turbulence models, transition point.

Introduction

The aerodynamic performance of an airfoil is a key factor in the design of many aerospace systems. Airfoils are used in a wide range of applications, from aircraft wings to wind turbines. The NACA 0012 airfoil has been widely used in many applications due to its simple design and predictable performance characteristics. However, as new technologies and design methods have been developed, modified airfoil shapes have emerged that may offer improved performance. In this project, we will compare the aerodynamic performance of the NACA 0012 airfoil with modified versions of the NACA 4-digit series airfoils, specifically the ISRO 405 and ISRO 540 airfoils. These airfoils were developed by the Indian Space Research Organization (ISRO) and have been used in various aerospace applications. The main objective of this project is to determine which airfoil shape provides the best overall performance in terms of lift, drag, and stall characteristics. Computational Fluid Dynamics (CFD) will be used to simulate the flow over the airfoils and analyze their performance. Specifically, pressure and velocity analysis will be conducted to determine the lift and drag characteristics of each airfoil shape. The results of this study will provide insight into the relative performance of different airfoil shapes and may inform the design of future aerospace systems. This project also aims to contribute to the existing body of knowledge on airfoil performance and help engineers make informed decisions when selecting airfoils for specific applications.

Department of Mechanical Engineering, Pragati Engineering College (Autonomous), Surampalem, India

Statement of the Problem

The main problem that this project aims to address is the need for improved aerodynamic performance in aerospace systems that rely on airfoils. While the NACA 0012 airfoil has been widely used and is well understood, there is a possibility that modified airfoil shapes, such as the ISRO 405 and ISRO 540, could offer improved performance characteristics. However, the relative performance of these airfoils has not been thoroughly investigated, and a clear understanding of their strengths and weaknesses is needed. To address this problem, this project will conduct a comprehensive analysis of the aerodynamic performance of the NACA 0012 airfoil and the modified ISRO 405 and ISRO 540 airfoils. By analysing pressure and velocity data using CFD simulations, we will compare the lift, drag,

and stall characteristics of each airfoil shape to determine which airfoil performs best overall. This project aims to provide insight into the relative performance of different airfoil shapes and inform the selection of airfoils for specific aerospace applications. By addressing the problem of aerodynamic performance in airfoil design, this project has the potential to contribute to the development of more efficient and effective aerospace systems.

Objectives of the study

- To compare the aerodynamic performance of the NACA 0012 airfoil with modified versions of the NACA 4-digit series airfoils, specifically the ISRO 405 and ISRO 540 air foils.
- To analyze the pressure and velocity data obtained from CFD simulations to determine the lift, drag, and stall characteristics of each airfoil shape.
- To contribute to the existing body of knowledge on airfoil performance and inform the design of more efficient and effective aerospace systems.

Review of Literature

Abbott IH and Von Doenhoff AE [1] The prime reason for the appearance in the Dover series of this eleven-year-old book lies in the compilation of data it contains on the geometry and subsonic aerodynamic characteristics of NACA wing sections. Bacha WA and Ghaly WS [2] When simulating the flow over airfoils at low Reynolds numbers, transition from laminar to turbulent flow plays an important role in determining the flow features and in qualifying the airfoil performance such as lift and drag. Badran O [3] (2008).A large difference in the pressure coefficient is observed between the top and bottom surface in the case of lower Reynolds number and thus it indicates that at low Reynolds number high lift is generated than at high Reynolds number. L/D study also reveals that with increasing Reynolds number the NACA0012 aerofoil losses its lifting aerodynamics property.The results discussed in the present study indicate that proper transition point prediction is crucial, especially when considering the drag characteristics. Johansen J [4] The computations of the NACA0012 airfoil 20 Ris0-R-987(EN) at high Reynolds number show a minor effect on the lift prediction while the drag characteristics are more influenced. Launder BE and Spalding DB [5] The experimental results of the water surface profile gave a high agreement with the results of the numerical models. The maximum value 28.78 of E% was obtained in single step broad crested weir in the experimental result and 27.35 in numerical result at S = 0.004. Finally, the range of the relative error of the energy dissipation between experimental and numerical results was achieved and the maximum was 6.76 in all runs. Ma L, et al. [6] When the attack angle changed from -8° to 13°, steady numerical methods could be applied to predict the aerodynamic performance of airfoil, the lift and drag coefficient curve of four turbulence models had consistent movements and shapes with the experimental curve. The lift coefficient curves of four turbulence models were much closer with the experimental data, while drag coefficient curves differed largely with the experimental data. McCroskey WJ [7] Aerodynamic results a seldom duplicated in different facilities to the level of accuracy that is required either for risk-fne engineering development or for the true verification of theoretical and numerical methods. Menter FR [8] (1994)The SST two equation turbulence model was introduced in 1994 by F.R.

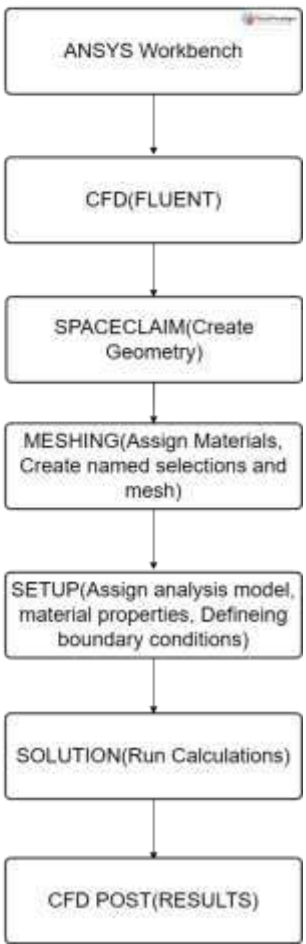
Menter to deal with the strong freestream sensitivity of the k-omega turbulence model and improve the predictions of adverse pressure gradients

. The formulation of the SST model is based on physical experiments and attempts to predict solutions to typical engineering problems. Silisteanu PD and Botez RM [9] there is a current need for a simple and effective way for determining the transition onset and transition extent on a solid surface in a general CFD solver, in order to include the transition effects in the aerodynamic coefficients' calculation. Spalart PR and Allmaras SR [10] (1992) Menter also reported somewhat disappointing results over a backward-facing step, traced to an excessively-rapid build-up of the shear stress (personal communication).Wilcox DC [11] The ZLES method and laminar simulation most accurately match experimental lateral-average adiabatic effectiveness along the streamwise direction from the trailing edge of the hole to 35-hole diameters downstream of the hole (X/D = 0 to X/D = 35), with RMS deviations of 5.1% and 4.2%, and maximum deviations of 8% and 11%, respectively.

Research Methodology

The research methodology for this project involves subjecting 2D models of the NACA 0012 airfoil, ISRO 405 and ISRO 540 modified versions of the NACA 4-digit series airfoils, to the viscous model and k-epsilon turbulence model.

To compare the performance of each airfoil shape, lift and drag forces, pressure and velocity distributions, and turbulent kinetic energy (TKE) distributions will be analyzed using CFD simulations. These simulations will be performed using commercial software such as ANSYS Fluent, which is commonly used in the aerospace industry for aerodynamic analysis. The procedure outline is illustrated in graph 1.



Graph 1: Process Flow

The lift and drag forces will be calculated using the simulation results, and the stall characteristics of each airfoil will be analyzed. Furthermore, the pressure and velocity distributions around the airfoil will be analyzed to gain insights into the flow behaviour and the separation point. The TKE distribution analysis will provide an indication of the level of turbulence near the airfoil and its impact on performance.

Overall, this research methodology aims to provide a comprehensive analysis of the aerodynamic performance of each airfoil shape by comparing their lift and drag forces, pressure and velocity distributions, and turbulent kinetic energy distributions. By applying these techniques, this project aims to provide valuable insights into the relative strengths and weaknesses of each airfoil shape, which can be used to improve the design of aerospace systems.

Results &Discussion:

The ANSYS Tool CFD-Post results are shown in Figures [1-9]. These figures depict the Pressure Distribution, Velocity distribution, Turbulence Kinetic Energy Distribution, Lift and Drag Forces. The analysis was performed subjecting these airfoils to viscous model and the above-mentioned parameters are produced in CFD-Post.

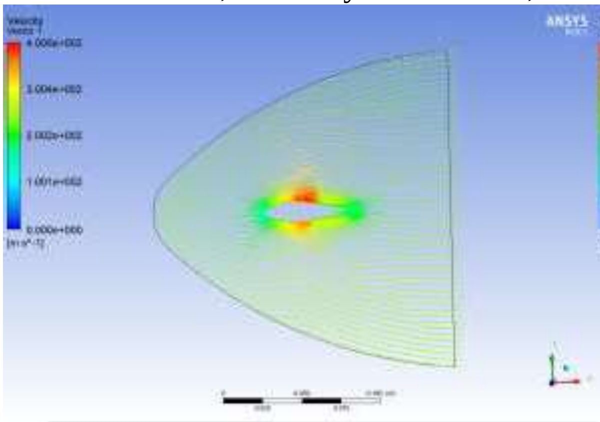


Fig 1- Velocity Distribution of NACA 0012

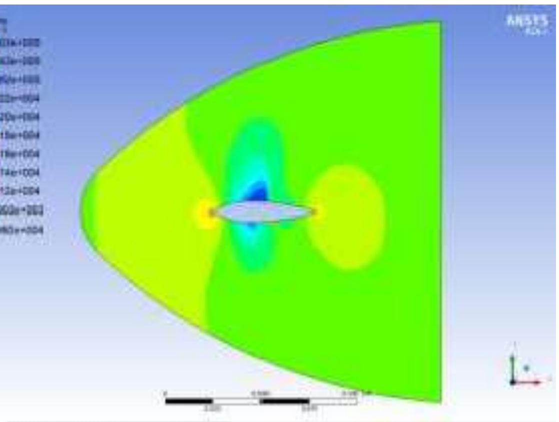


Fig 2- Pressure Distribution of NACA 0012

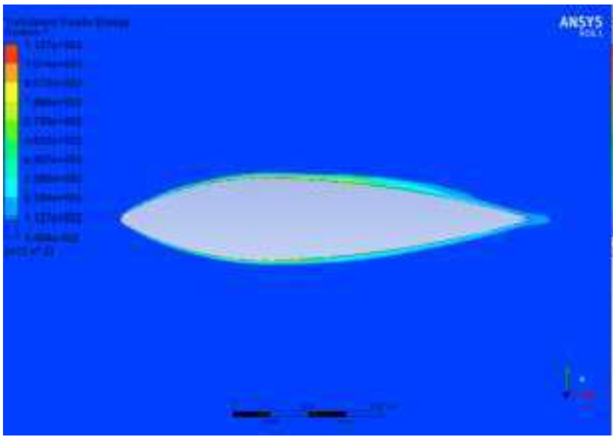


Fig 3 TKE Distribution of NACA 0012

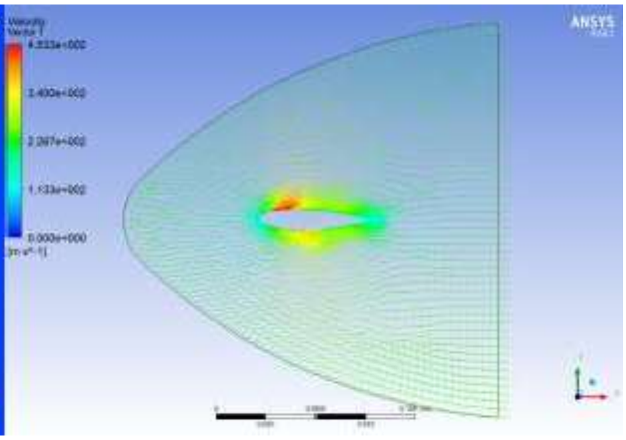


Fig 4 Velocity Distribution of ISRO 400

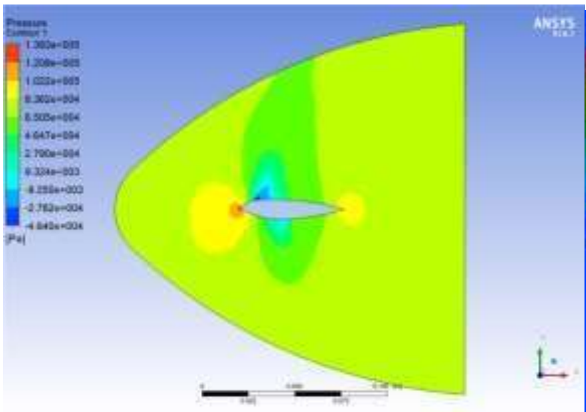


Fig 5 Pressure Distribution of ISRO 400

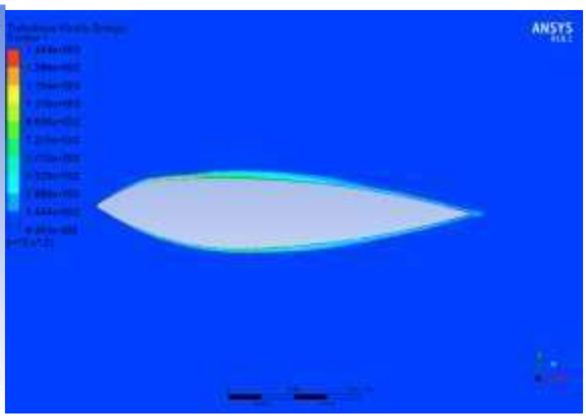
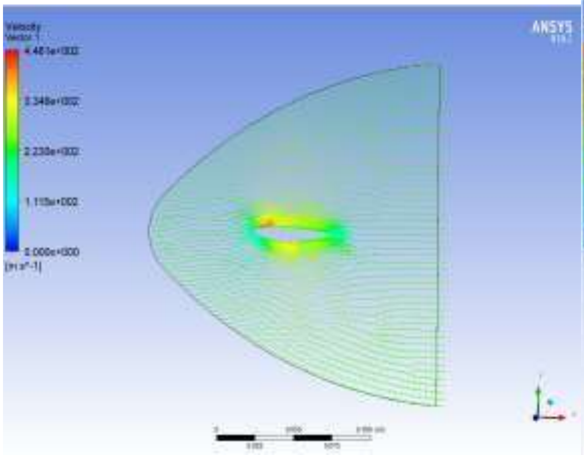


Fig 6 TKE Distribution Of ISRO 400



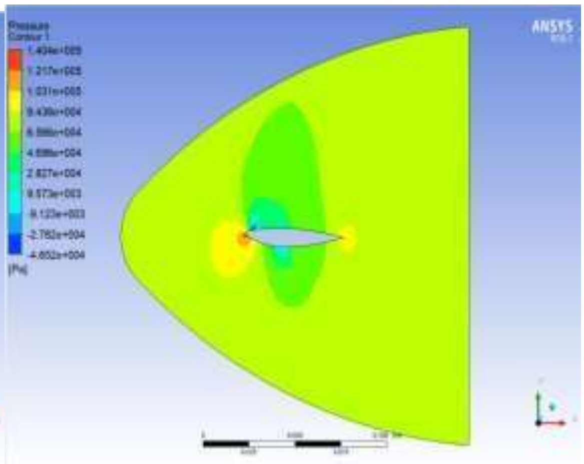


Fig 7 Velocity Distribution of ISRO 500 Fig 8 Pressure Distribution of ISRO 500

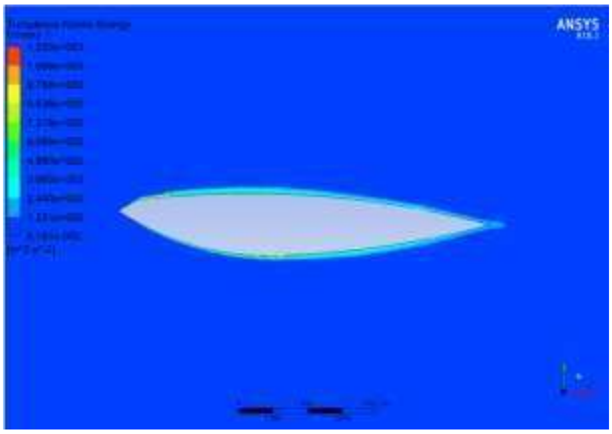
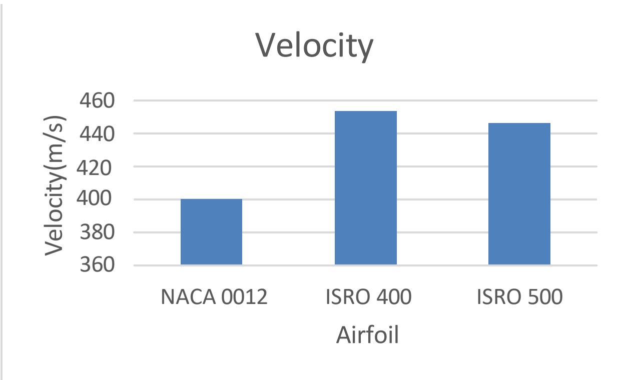
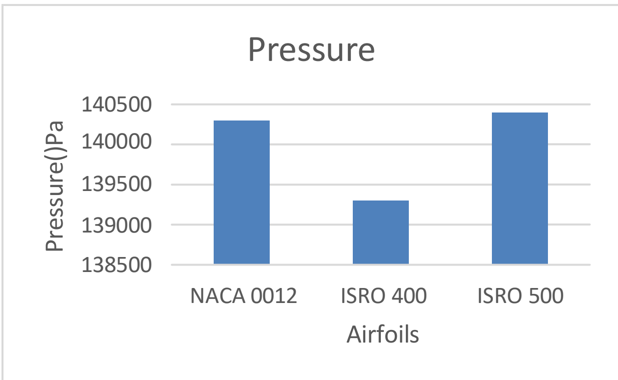


Fig 9 TKE Distribution of ISRO 500

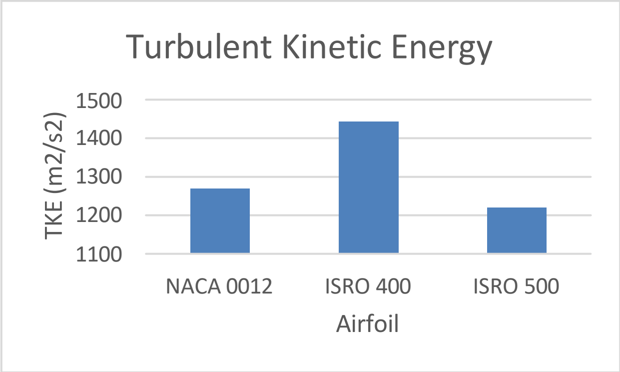
RESULTS:

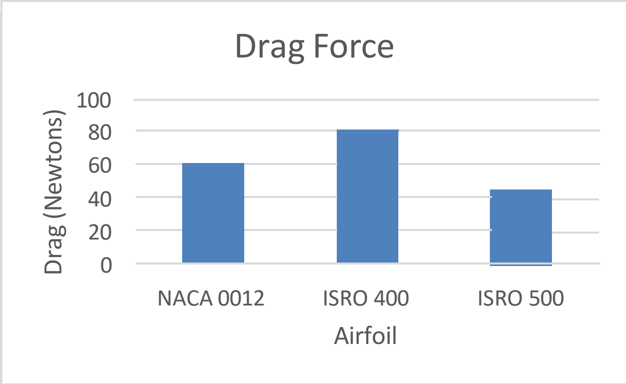
The ANSYS Fluent analysis provides data on velocity, pressure, turbulent kinetic energy, lift and drag forces for each airfoil shape. To illustrate the performance of each airfoil, graphs are generated that show the distribution of these parameters across the airfoil surface. These graphs(1-5) provide valuable insights into the relative strengths and weaknesses of each airfoil shape..





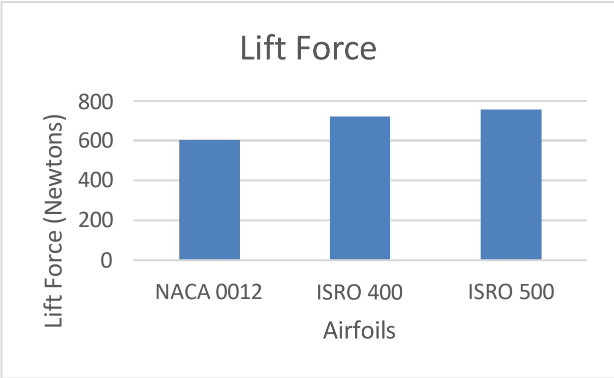
Graph :1 Velocity Comparision Graph :2 PressureComparision





Graph :3TKEComparision

Graph :4Drag Force Comparision



Graph :5 Lift Force Comparision

CONCLUSION:

In conclusion, the ANSYS Fluent analysis revealed that the 500 series airfoil outperformed both the NACA0012 and 400 series airfoils in terms of lift, turbulent kinetic energy, drag and velocity. The 400 series airfoil performed better than NACA0012 in terms of lift, but not as well as the 500 series airfoil. However, the 400 series airfoil had worse turbulent kinetic energy than both the NACA0012 and 500series airfoils.

Further analysis is required to determine the performance of these airfoils at different angles of attack, as well as to investigate their stall properties. Nonetheless, the current study provides valuable insights into the relative performance of these airfoil shapes, which can be used to inform the design of aerospace systems.

REFERENCE:

1. Abbott IH, Von Doenhoff AE (1959). Theory of Wing Sections. Dover Publishing, New York.
2. Bacha WA, Ghaly WS (2006). Drag Prediction in Transitional Flow over Two-Dimensional Airfoils, Proceedings of the 44th AIAA Aerospace Sciences Meeting and Exhibit, Reno, NV.
3. Badran O (2008). Formulation of Two-Equation Turbulence Models for Turbulent Flow over a NACA 4412 Airfoil at Angle of Attack 15 Degree, 6th International Colloquium on Bluff Bodies Aerodynamics and Applications, Milano, 20-24 July.
4. Johansen J (1997). Prediction of Laminar/Turbulent Transition in Airfoil Flows. Risø National Laboratory, Roskilde, Denmark.
5. Launder BE, Spalding DB (1974). The numerical computation of turbulent flows. Comp. Meth. Appl. Mechanics Engr., 3: 269-89.
6. Ma L, Chen J, Du G, Cao R (2010). Numerical simulation of aerodynamic performance for wind turbine airfoils. Taiyangneng Xuebao/Acta Energiae Solaris Sinica, 31: 203-209.

7. McCroskey WJ (1987). A Critical Assessment of Wind Tunnel Results for the NACA 0012 Airfoil. U.S. Army Aviation Research and Technology Activity, Nasa Technical Memorandum, 42: 285-330.
8. Menter FR (1994). Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications. AIAA J., 32: 1598-1605. Shih TH, Liou WW, Shabbir A, Zhu J (1995). A new k-ε eddy – viscosity model for high Reynolds number turbulent flows – model development and validation. Computers Fluids, 24: 227-238.
9. Silisteanu PD, Botez RM (2010). Transition flow occurrence estimation new method. 48th AIAA Aerospace Science Meeting. Orlando, Floride, Etats-Unis, Janvier, pp. 7-10
10. Spalart, PR, Allmaras SR (1992). A One-Equation Turbulence Model for Aerodynamic Flows. AIAA Paper, pp. 92-439. Sutherland W (1893). The viscosity of gases and molecular force. Philosophical Magazine, S., 5, 36: 507-531.
11. Wilcox DC (1988). Re-assessment of the scale-determining equation for advanced turbulence models. AIAA J., 26: 1414-1421.