ISSN : 2249-5762 (Online) | ISSN : 2249-5770 (Print)

CFD and Real Time Analysis of an Unsymmetrical Airfoil

[1]Abhay Sharma, [2]Manpreet Singh Sarwara, [3]Harsimranjeet Singh, [4]Lakshya Swarup, [5]Rajeev Kamal Sharma
[1,2,3,4]UG Students, School of Mechanical Engineering, Chitkara University, Punjab, India
[5]Associate Professor, School of Mechanical Engineering, Chitkara University, Punjab, India

Abstract
An all new NACA4413 airfoil has been generated and fabricated as per prescribed input parameters of NACA airfoil generator. Some of the main behaviours of the airfoil have been examined using CFD software and same has been verified in real life flow in experimental setup of a wind tunnel. With the variation in angle of attack the changes in velocity profile, pressure profile, flow separation, wakes formation has been studied in both ways. In the analysis the angle of attack has been incremented by 5°, with the initialisation from 0° and carried out until flow separates in real time flow and convergent solutions in CFD approach and it has been found that with the increments of angle of attack the pressure increases and velocity decreases at lower surface. The flow separation and wakes formation increases with increase in angle of attack.

Keywords
Real Time Analysis, CFD Analysis, Angle of Attack, NACA 4413, NACA Airfoil Generator

I. Introduction
For the last two decades Computational Fluid Dynamics (CFD) has fascinated researchers, as it is a fast, accurate and reliable method of analysing the variation of hydrodynamic properties of the flow over and within a body. The various industries like automobile industry, marine industries, aeronautics industries, electronic gadgets companies are accepting the usefulness of CFD in assessing the performance of their products under various hydrodynamic fluid flow fields. Some of the parameters which are very hard and expensive to calculate in real time practices, those parameters can be easily approximated by using CFD method.

Many researchers has applied commercially available CFD packages to study either the effect of change of shape of body over the fluid flow or the effect of change of fluid flow parameters over a particular body. During lift and drag coefficient analysis at high Reynolds number of 9 million over a NACA 64618 airfoil, it has been found that in comparison to K-Ɛ model lift and drag coefficients calculated using Mentor's shear stress transport turbulence model are much closer to experimental values of lift and drag coefficients [1]. Study of variation of pressure coefficients at various angles of attack ranging from 0o to 12o over wind turbine blades following NACA 4412 airfoil profile using CFD showed that pressure coefficient increases at the upper surface and decreases at lower surface of the airfoil with increase in angle of attack [2]. While analysing flow under transition (i.e. from laminar to turbulent) over NACA 4412 airfoil, it has been found that in comparison to Spalart allmaras turbulence model, K-ω SST turbulence model with transition capabilities gives close prediction of lift and drag coefficient both in pre stall and post stall region [3]. During the investigation of two dimensional unsteady flow around a vertical axis wind turbine (VAWT) blade following NACA 2415 camber airfoil geometry, it was concluded that camber airfoils have the potential of self start when used in vertical axis wind turbines (VAWT) [4]. However in comparison to the non-self starting airfoils, a reduction in the peak efficiencies of the simulated model has been found. The CFD analysis has been done with a moving mesh technique using the selected airfoil section with fixed pitch three blades based on RANS equations. Increase in lift and drag coefficients has been found to be directly proportional to angle of attack while analyzing the flow over NACA 63018 airfoil at different Reynolds number and at different angles of attack [5]. While comparing CFD analysis and real time analysis with flow over a NACA 0015 airfoil it has been found that lift force, velocity and pressure difference between upper and lower surfaces increases with increase in angle of attack [6]. Extent of flow separation and intensity of wake formation has also been found to increase with increase in angle of attack.

This paper focuses on the real time performance analysis of self designed NACA 4413 unsymmetrical airfoil under two dimensional flow and compares it with computational model results. An all new NACA 4413 airfoil has been generated by using NACA airfoil generator, by changing the various input parameters like maximum camber position and airfoil thickness. Different inputs were checked and finally one of the best in shape to fabricate was being selected. Real time analysis has been done in a 600mm×270mm wind tunnel, whereas computational model of the airfoil has been developed using Fluent 13.0. Model results show good agreement with the real time flow analysis results.

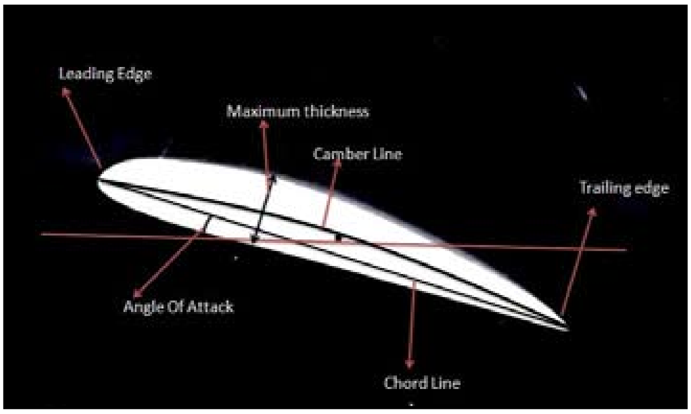


Fig. 1: Basic Nomenclature of NACA 4413 Airfoil

II. Airfoil Nomenclature
Fig. 1 shows the basic nomenclature of an unsymmetrical airfoil. Leading edge, angle of attack, chord line and camber line are the basic terms to describe the geometry of an unsymmetrical airfoil. Chord line is the straight line joining the leading edge with trailing edge. The camber line also connects the leading edge with trailing edge but along with the curvature of the airfoil. Angle of attack is the angle between the fluid flow direction and chord line of the airfoil. With the variation in angle of attack the magnitude of lift and drag forces changes.

III. Experimental Setup
Fig. 2 shows the wind tunnel (600mm×270mm) with two prominent distinct parts (i) transparent cross section for flow visualisation and (ii) a Divergent Outlet has been used for real time flow analysis.

For the flow of air through the wind tunnel a suction fan has been used at the farthest end of the divergent section, and to reduce the turbulence created by fan air a honey comb mesh has been used. All Angle of attack has been changed with the increment of 5° starting with 0° up to 10°. Real time analysis has been carried out at the ambient conditions and Reynolds number of 3×104. NACA 4413 airfoil has been used for real time flow analysis.

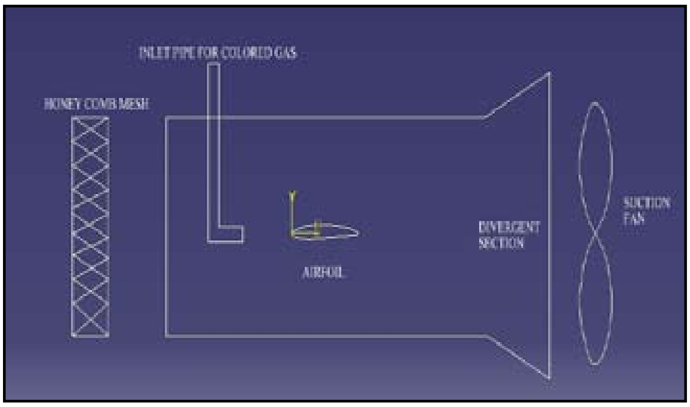


Fig. 2: Wind Tunnel Setup

IV. Airfoil Design

NACA airfoil generator [7] has been used to plot the new NACA 4413 airfoil. Various combinations of parameters like maximum camber position and thickness have been tested to get the new profile. Fig. 3 shows the input parameters to NACA generator and the final NACA 4413 profile generated by the NACA generator.

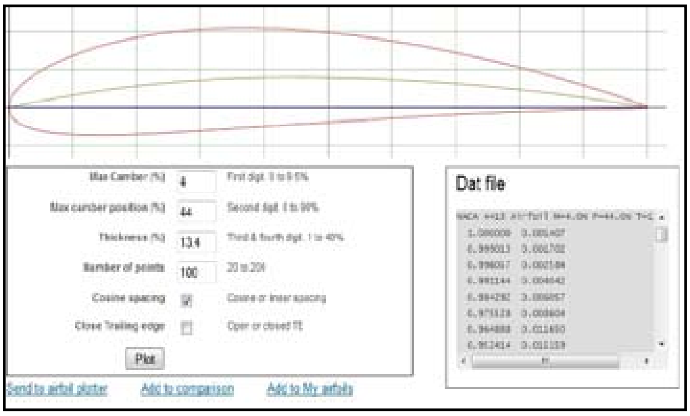


Fig. 3: Input Parameters and Profile Generated by NACA Generator

V. CFD Analysis

A. Modelling of NACA 4413 Airfoil

Fig. 3 shows the NACA 4413 airfoil profile generated using NACA generator. CATIA V6 (R2013x) has been used to reproduce the 2D model of airfoil as shown in Fig. 4. Table 1 shows the dimensions of the airfoil.

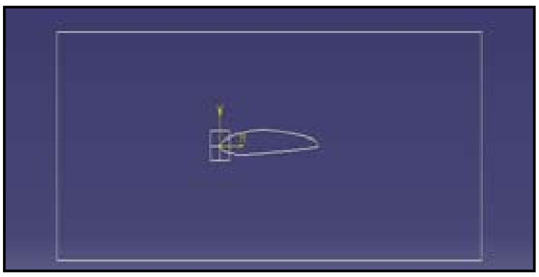


Fig. 4: NACA 4413 2D Model Produced by CATIA V6

B. Mesh Generation

Quadrilateral mesh has been created by using Gambit 2.2.30 and is shown in fig. 5.

C. Flow Analysis

Fluent 13.0 solver has been used of analyse the flow around NACA 4413 airfoil.

Table 1: NACA 4413 Airfoil Dimensions

S.No.	Parameters	Dimensions
1.	Chord Length	182mm
2.	Span of Airfoil	110mm
3.	Thickness	13.4%
4	Maximum Camber	4%
5.	Maximum Camber position	44%

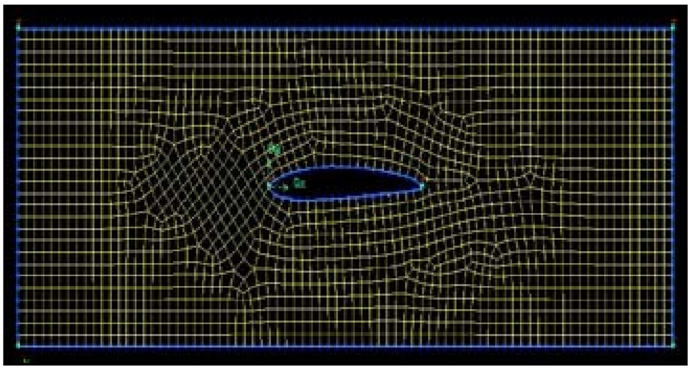


Fig. 5: 2-D Quadrilateral Mesh Around NACA 4413 Airfoil

Input parameters for of Fluent 13.0 solver are shown in Table 2. The mesh file created using Gambit 2.2.30 has been imported to Fluent 13.0 solver for processing.

Table 2: Input Parameters to Fluent 13.0 solver

Flow Parameters	Values
Pressure (atm)	1
Temperature(K)	300
Velocity (m/sec)	2.0
Density (kg/m3)	1.2335
No. of iterations	1000
Residual	0.001
Boundary Conditions	No Slip

VI. Results and Discussions

A. Angle of Attack = 0°

Fig. 6(a) and Fig. 6(b) shows the flow visualisation in CFD and Real time analysis respectively around the NACA 4413 airfoil at angle of attack (AOA) = 0°. The flow visualisation can be further improved by decreasing the mesh size. Fig. 7(a) represents the CFD velocity profile over the airfoil and Fig. 7(b) represents the pressure profile over NACA 4413 airfoil at AOA = 0°. It has been observed that the velocity profile and pressure profile is unsymmetrical over the upper and the lower surfaces. It has also been observed that the flow is unsymmetrical about camber line with higher velocity at upper surface, which results in lower pressure at upper surface in comparison to the lower surface of the airfoil. No flow separation has been observed at AOA = 0°.

ISSN : 2249-5762 (Online) | ISSN : 2249-5770 (Print)

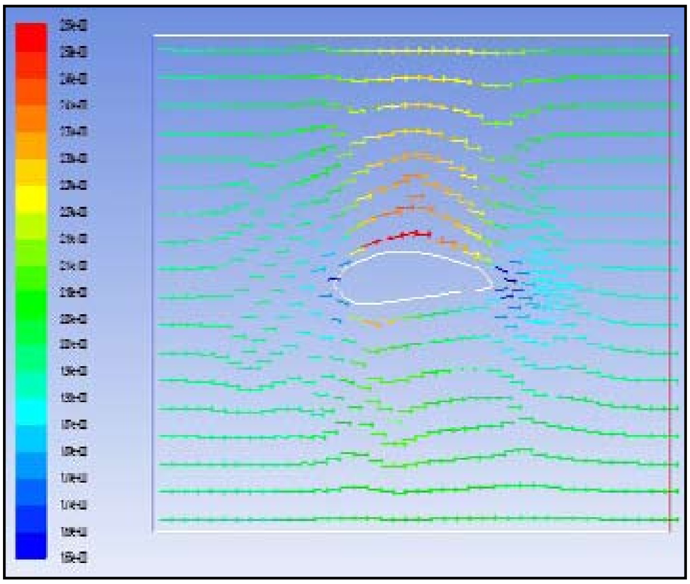


Fig. 6(a): Flow visualization in CFD around NACA 4413 airfoil at AOA = 0°

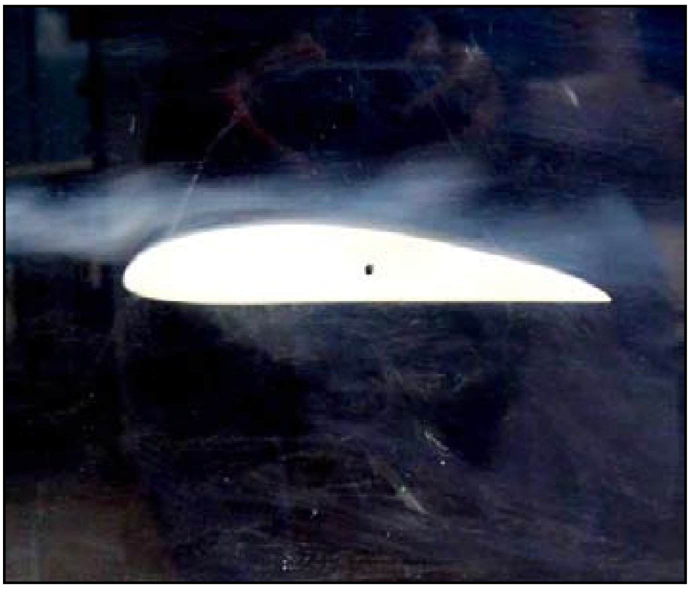


Fig. 6(b): Real Time NACA 4413 airfoil at AOA = 0°

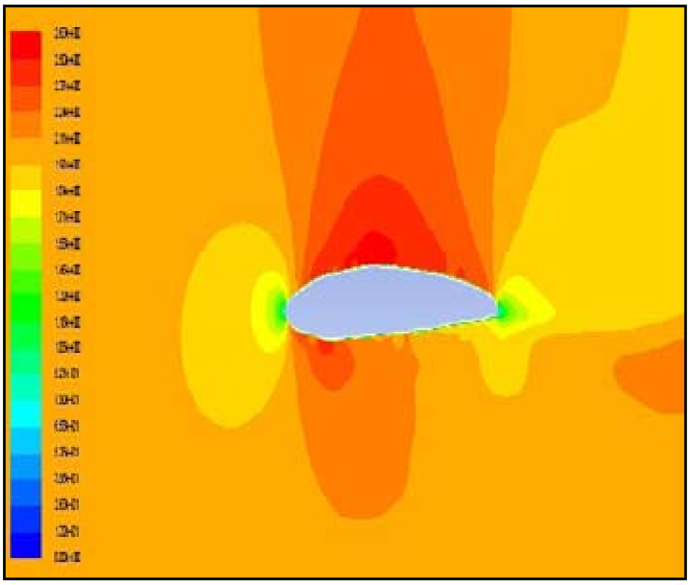


Fig. 7(a): Velocity profile over NACA4413 at AOA=0°

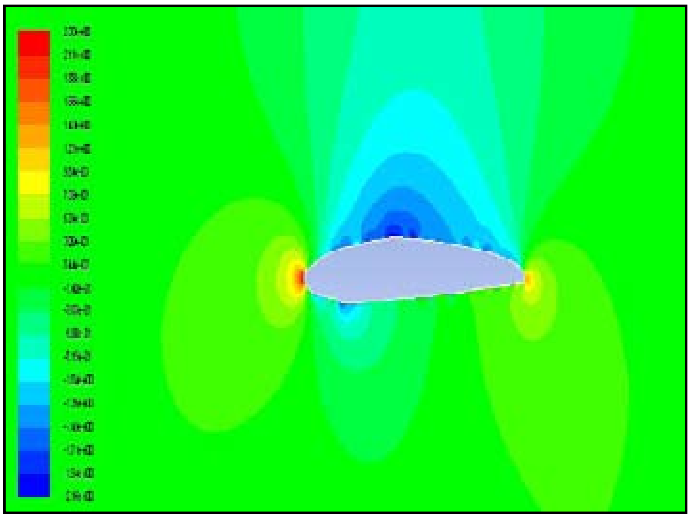


Fig. 7(b): Pressure Profile Over NACA 4413 at AOA=0°

B. Angle of Attack = 5°

Fig. 8(a) and fig. 8(b) shows the software generated and real time flow around NACA 4413 airfoil at AOA = 5° respectively. Software generated velocity variations and pressure variations over NACA 4413 airfoil at AOA=5° are shown in Fig. 9(a) and Fig. 9(b) respectively. Further increase in flow velocity and decrease in pressure on upper surface has been observed at AOA=5o in comparison to AOA = 0o. Minor flow separation at the trailing edge has been observed from the real time flow visualization in Fig. 8(b).

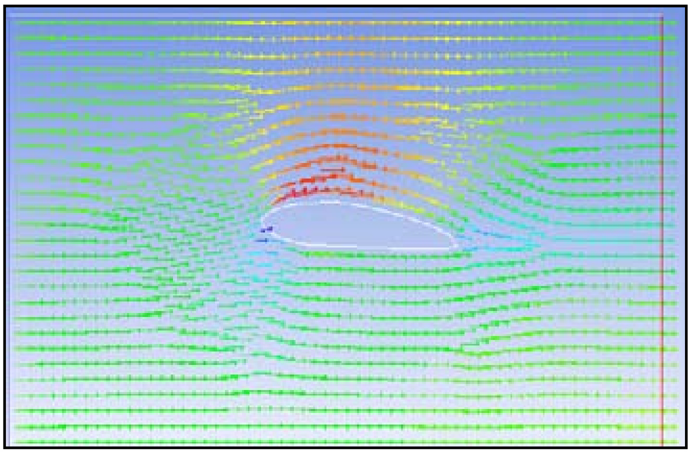


Fig. 8(a): Flow visualization in CFD around NACA 4413 airfoil at AOA = 5°

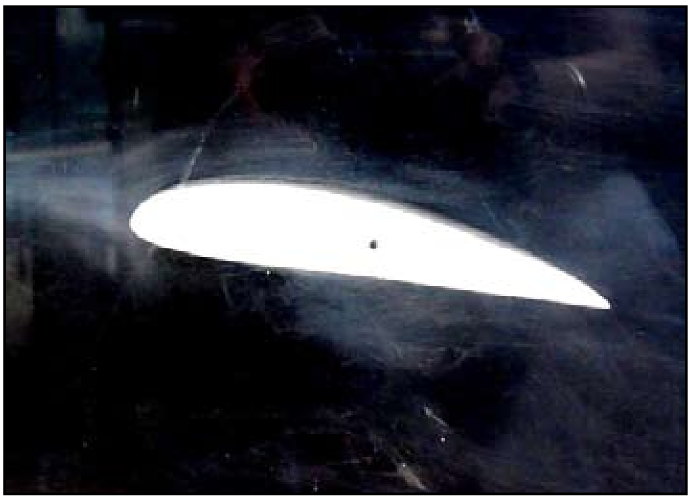


Fig. 8(b): Real Time NACA 4413 airfoil at AOA = 5°

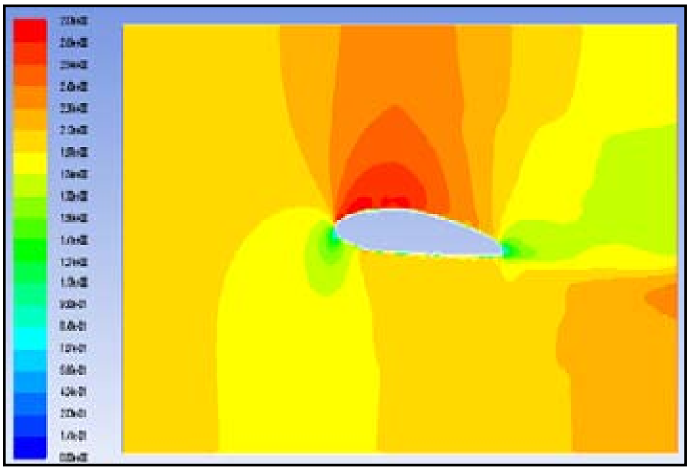


Fig. 9(a): Velocity profile over NACA 4413 at AOA=5°

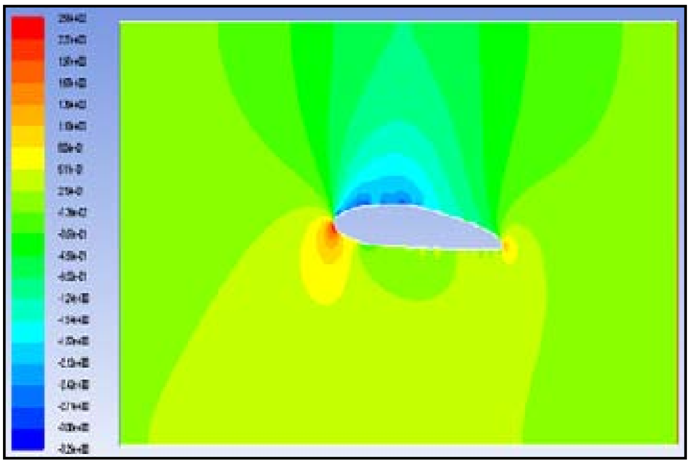


Fig. 9(b): Pressure profile over NACA 4413 at AOA=5°

C. Angle of Attack = 10°

Fig. 10(a) and Fig. 10(b) represents the CFD and real time flow visualisation around the NACA 4413 airfoil at AOA = 10° respectively. Fig. 11(a) represents the velocity profile and Fig. 11(b) represents the pressure profile around NACA 4413 airfoil. The velocity and pressure profile clearly shows the least value of velocity at lower surface of the trailing edge, and similarly the highest value of pressure at the same point. Prominent flow separation and wake formation has been observed over the upper surface of the airfoil.

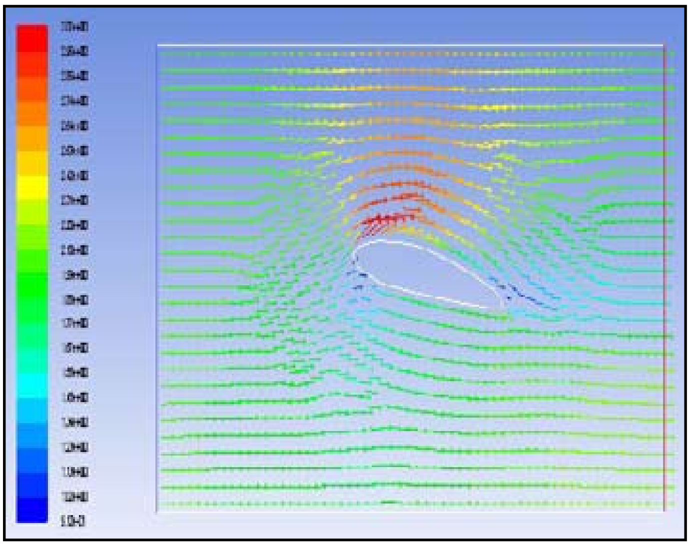


Fig. 10(a): Flow visualization in CFD around NACA 4413 airfoil at AOA = 10°

VII. Conclusion

The behaviour of the flow around an all new designed NACA 4413 airfoil has been analysed at different angles of attack (0°, 5°, 10°) under

Fig. 10(b): Real Time NACA 4413 airfoil at AOA = 10°

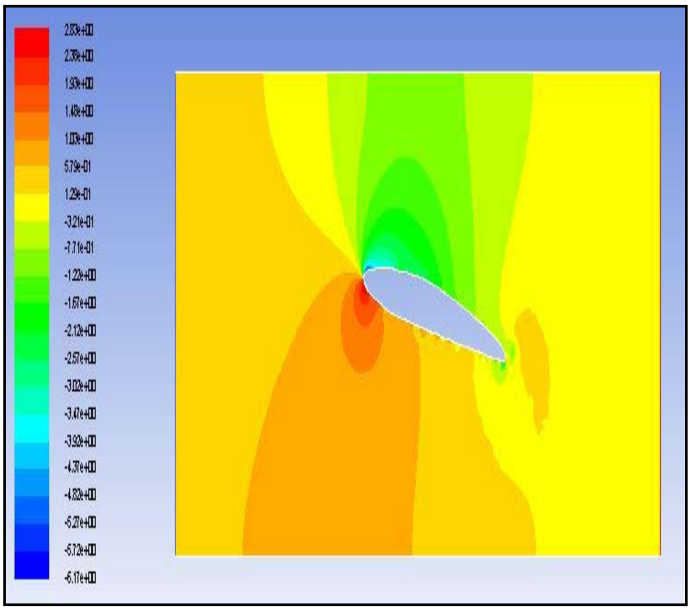


Fig. 11(a): Velocity Profile of NACA 4413 at AOA=10°

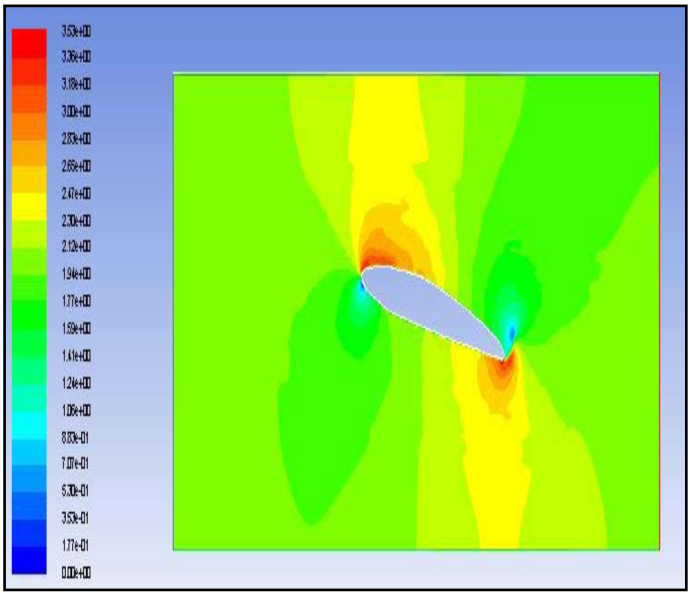


Fig. 11(b): Pressure profile of NACA 4413 at AOA=10°

 ISSN : 2249-5762 (Online) | ISSN : 2249-5770 (Print)

real time situation and by simulating the flow using CFD and Real Time approaches. For CFD analysis Fluent 13.0 solver has been used whereas for real time analysis a lab scale wind tunnel has been used. A good agreement has been found between simulation and real time analysis results. Flow separation has been found to increase at the upper surface with the increase in angle of attack. Wake formation has been found at the trailing edge and its strength has been found to increase with increase in angle of attack. With increase in angle of attack, an increase in flow velocity and decrease in pressure at the upper surface has been observed.

References

[1] Shankara, M.H.M., Madhukeshwara,N., Kumarappa, S.,"Assessment of boundary layer flow modeling approaches in computational fluid dynamics for compressible external aerodynamics using NACA-64618 Airfoil", International Journal of Innovative Research in Science, Engineering and Technology, Vol. 2, No. 7, 2013, pp. 3137-3144.

[2] Kevadiya, M.,"CFD Analysis of Pressure Coefficient for NACA 4412", International Journal of Engineering Trends and Technology,Vol. 4, No. 5, 2013, pp. 2041-2043.

[3] Ravi, H.C., Madhukeshwara, N., Kumarappa, S., "Numerical investigation of flow transition for NACA-4412 airfoil using computational fluid dynamics". International Journal of Innovative Research in Science, Engineering and Technology, Vol. 2, No. 7, 2013, pp. 2778-2785.

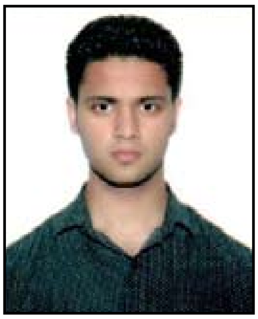
[4] Beri, H., Yao, Y.," Effect of camber airfoil on self starting of vertical axis wind turbine", Journal of Environmental Science and Technology, Vol. 4, No. 3, 2011, pp. 302-312.

[5] Azeez, A., Paul, J.,"CFD analysis of NACA 63-018 airfoil at different Reynolds number", International Journal of Engineering Trends and Technology, Vol. 12, No. 5, 2014, pp: 258-264.

[6] Sharma, A., Sarwara, M., Singh, H., Swarup, L., Sharma, R.K.,"CFD and Real Time Analysis of a Symmetric Airfoil" International Journal Of Research In Aeronautical And Mechanical Engineering, Vol. 2, No. 7, 2014, pp: 84-93.

[7] [Online] Available: http://airfoiltools.com/airfoil/naca4digit.

Abhay Sharma: Abhay Sharma is an enrolled student in School of Mechanical Engineering at Chitkara University, Rajpura, Punjab. His areas of interest are Computational Fluid Dynamics, Designing, Problem Solving Techniques, and Heat Transfer.



Manpreet Singh Sarwara: Manpreet Singh Sarwara is an enrolled student in School of Mechanical Engineering at Chitkara University, Rajpura, Punjab. His areas of interest are Computational Fluid Dynamics, Problem solving Techniques, Thermodynamics, and Fluid Mechanics.

Harsimranjeet Singh Dhaliwal: Harsimranjeet Singh Dhaliwal is an enrolled student in School of Mechanical Engineering at Chitkara University, Rajpura, Punjab. His areas of interest are Computational Fluid Dynamics, Designing, Mathematics, and Thermodynamics.

Lakshya Swarup: Lakshya Swarup is an enrolled student in School of Mechanical Engineering at Chitkara University, Rajpura, Punjab. His areas of interest are Mathematics, Computational Fluid Dynamics, and Thermal Engineering.

Rajeev Kamal Sharma: Rajeev Kamal Sharma is working as Associate Professor in the School of Mechanical Engineering at Chitkara University, Rajpura, Punjab. His areas of interest are fluid mechanics, heat transfer and computational fluid mechanics. He has published research articles in many journals, international conferences and national conferences.