INTERNATIONAL JOURNAL OF RESEARCH IN AERONAUTICAL AND MECHANICAL ENGINEERING

Vol.2 Issue.7,

July 2014.

Pgs: 84-93

ISSN (ONLINE): 2321-3051

INTERNATIONAL JOURNAL OF RESEARCH IN AERONAUTICAL AND MECHANICAL ENGINEERING

CFD and Real Time Analysis of a Symmetric Airfoil

Abhay Sharma[1], Manpreet Singh Sarwara[1], Harsimranjeet Singh[1], Lakshya Swarup[1],

Rajeev Kamal Sharma[2]

[1]UG students, School of Mechanical Engineering, Chitkara University, Punjab.

[2]Associate Professor, School of Mechanical Engineering, Chitkara University, Punjab.

Author Correspondence: UG students, School of Mechanical Engineering, Chitkara University, Punjab.

Phone - +91-9478184116., abhay0826@gmail.com

Abstract

A NACA 0015 airfoil has been designed and fabricated as per NACA specification. The performance of the designed airfoil has been evaluated using CFD software and the same has been verified by studying its behaviour in a lab scale wind tunnel. The effect of angle of attack on the lift force, flow velocity profile, pressure profile, flow separation and wake formation has been studied. The airfoil has been tested at 0^o, 5^o 10^o and 15^o angle of attack and it has been found that lift force, velocity and pressure difference between upper and lower surfaces increases with angle of attack. Extent of flow separation and intensity of wake formation also increases with increase in angle of attack.

Keywords: Symmetric airfoil; real time analysis; CFD analysis; angle of attack.

1. Introduction

In the recent past computational fluid dynamics (CFD) has become popular among the researchers, as it fulfils the requirement for faster and accurate methods for predicting the various flow field parameters around the geometries of interest. CFD has found applications in the field of aerospace, automobiles, Marine and industrial processes and components where fluid flow place s a major role in assessing the performance. Flow over the geometries can not only be visualised but certain flow parameters which are difficult, impossible or expensive to measure experimentally can be approximated correctly by using suitable CFD packages.

Study of variation of various hydrodynamic parameters over an airfoil has been an area of research for long time. Before the inception of CFD the variations of hydrodynamic parameters during the flow over an airfoil was estimated by conducting experiments over airfoil in wind tunnels. During the flow over an airfoil, the flow splits and passes over the upper and lower surfaces of the airfoil. The shape of airfoil is such that the flow stretches over the upper surface resulting in decrease of pressure. On the other hand the flow over the lower portion of the airfoil moves with same speed and pressure resulting in an upward force called LIFT. As the flow progresses over the airfoil it experiences the drag force, in the direction of flow. The lift force experience by an airfoil is smaller in comparison to the weight of the airfoil in low Reynolds numbers. With the increase in Reynolds number the lift force also increases, and when lift force is greater than the gravitational force the airfoil starts moving in upward direction. As a Reynolds number of the flow increases the drag force over the airfoil also increases. Karna et. al. (2014) studied the lift and drag forces on NACA 0012 airfoil at zero degree angle of attack using CFD package. The flow was subsonic and it was observed that for such airfoil the lift force was zero at zero degree angle of attack. Variations of static and dynamic pressure over NACA 4412 airfoil has been studied byParbhakar A. (2013) using CFD analysis at different angles of attack. Maximum dynamic pressure was found to occur at upper surface near and around maximum camber and minimum static pressure was found to occur at and around the same location. It was also concluded that with the increase in the angle of attack the stagnation point moves away from leading edge on the lower surface of the airfoil. Many other researchers have tried to model the flow around airfoil with different geometries using various CFD packages [Shankara MHM. et al (2013), Mevadiya M (2013)], but the CFD outcomes has not been validated with real time analysis.

This paper presents a real time and CFD analysis comparison for NACA 0015 airfoil. NACA 0015 has been designed using NACA generator. Real time analysis has been carried out in 600×270mm wind tunnel. CFD analysis has been done using Fluent 13.0. CFD analysis shows good agreement with real time analysis.

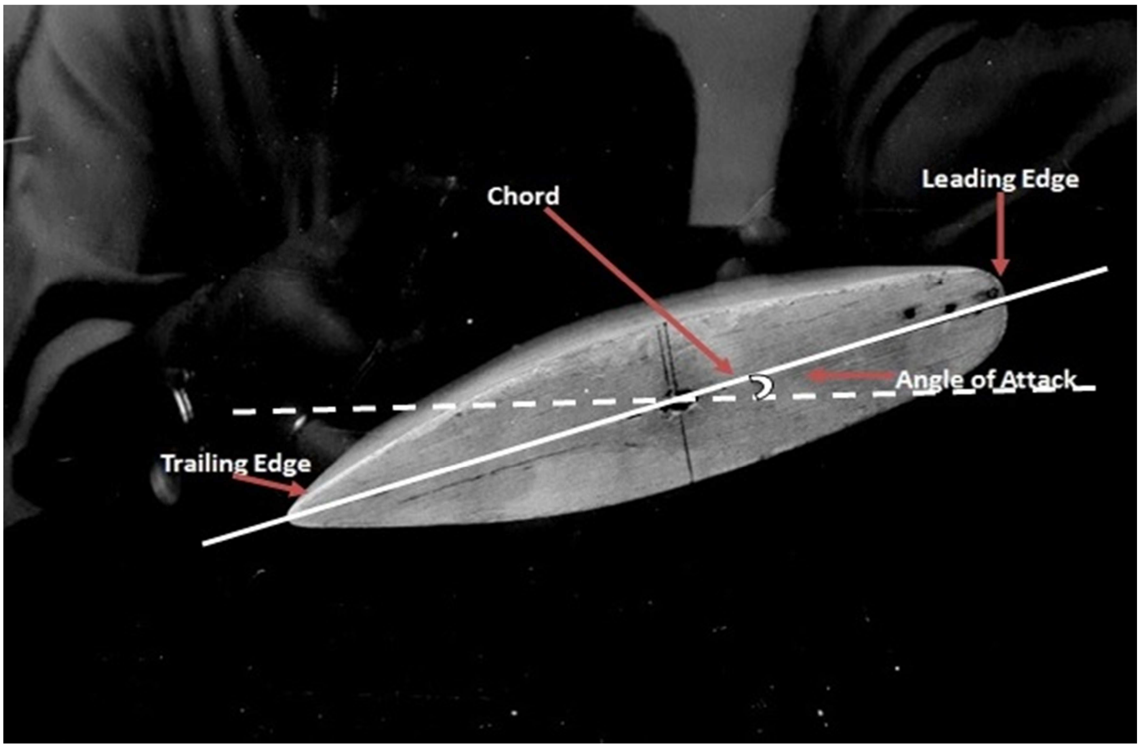


Figure 1 Basic nomenclature of Airfoil.

2. Airfoil Nomenclature

The basic terms of airfoil are leading edge, chord, camber, Angle of attack, trailing edge. In symmetrical airfoil chord and camber lies on each other because of symmetry in shape. Angle of attack is the angle between the incoming air and relative wind and a reference line on the airplane or wing. Figure 1 shows the basic nomenclature of airfoil.

3. Wind tunnel Experimental Setup

Figure 2 shows the wind tunnel (600mm×270mm) set up used for real time flow analysis which has two distinct sections (i) transparent constant cross section for flow visualization and (ii) a divergent outlet. A suction fan has been installed at the end of divergent outlet and a honey comb mesh at the flow entry has been used to decrease the turbulence in the air flow. The real time flow analysis has been carried out at a Reynolds number= 3×10^4. Angle of attack has been varied from 0° to 15° with an increment of 5°. Ambient conditions are pressure equal to 1 atmosphere and temperature equal to ambient temperature. NACA 0015 airfoil profile has been used for real time flow analysis.

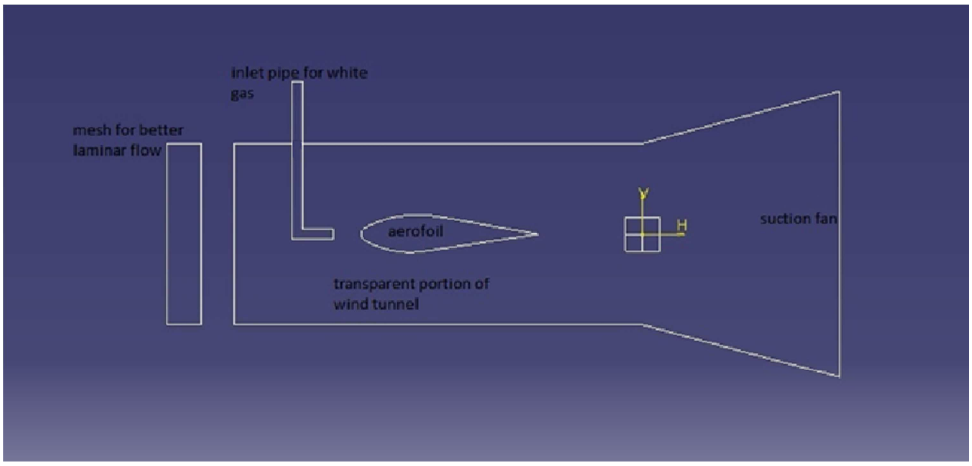


Figure 2 Wind tunnel set up line diagram

4. CFD Analysis

4.1 Modelling of NACA 0015 airfoil.

Figure 3 this shows the NACA 0015 airfoil profile has been generated using NACA Generator [http://airfoiltools.com/airfoil/naca4digit]. CATIA V6 (R2013x) has been used for reproducing 2D model of airfoil as shown in figure 4 below. The dimensions of the airfoil are shown in table 1.

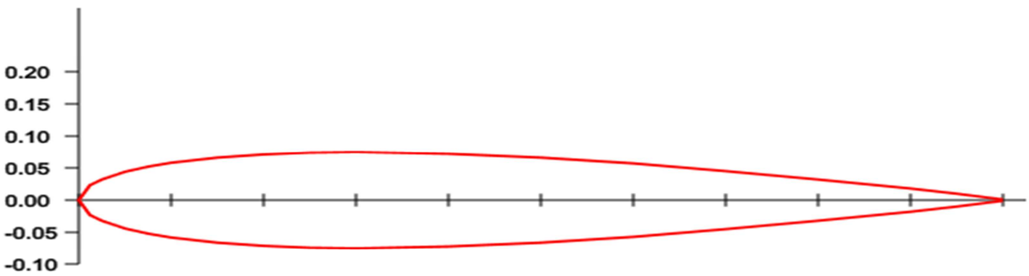


Figure 3 NACA 0015 airfoil

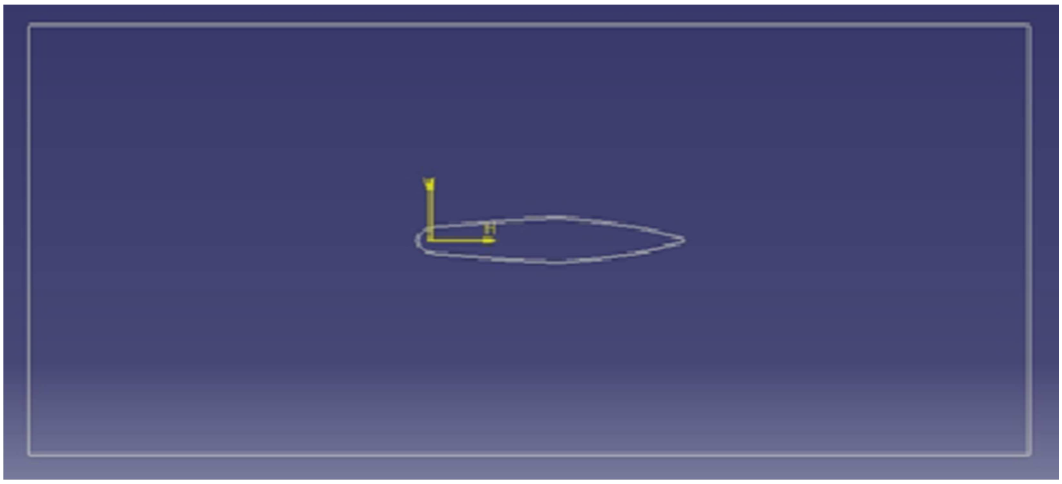


Figure 4 CATIA designed Airfoil

Table 1 Dimensions of NACA 0015 airfoil

S.No.	Parameters	Dimensions
1.	Chord Length	182mm
2.	Span of Airfoil	110mm
3.	Thickness	15%
4.	Leading Edge Radius	4.2%
5.	Lower Flatness	37.4%
6.	Trailing Edge Angle	20.5°
7.	Camber	0%

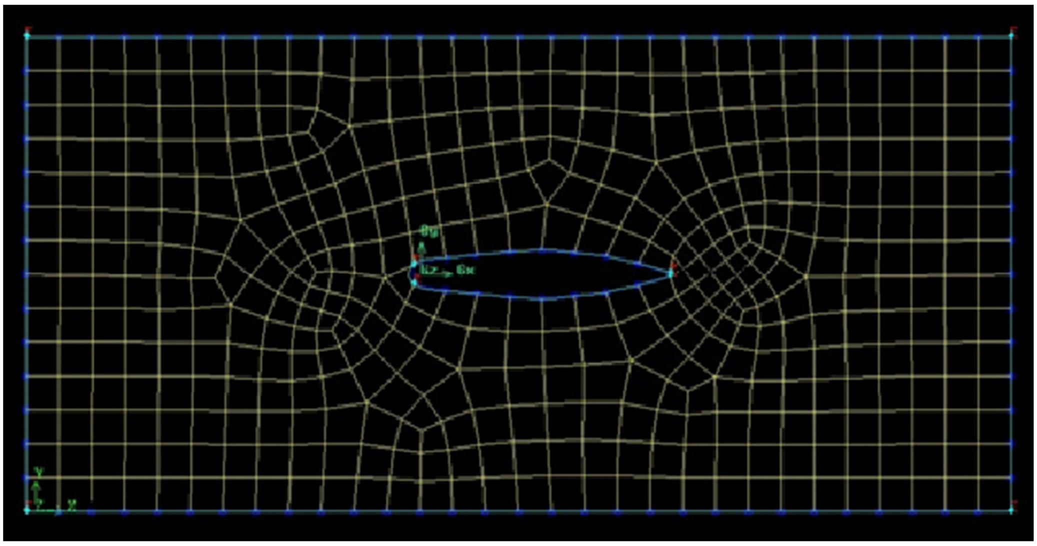


Figure 5 Generated mesh in Gambit

Table 2 Input parameters

S.No.	Parameters	Readings
1.	Pressure(atm)	1atm
2.	Temperature (Kelvin)	300K
3.	Velocity (m/sec)	2m/sec
4.	Density (kg/m^3)	1.2335kg/m^3
5.	Iterations.	1000.
6.	Residual	0.001
7.	Boundary conditions	No slip

4.2 Mesh Generation.

Quadrilateral mesh has been created using Gambit 2.2.30 as shown in figure 5.

4.3 Flow analysis

Fluent 13.0 has been used as a solver for this particular problem. The input parameters have been shown in table 2.

The generated Mesh file with boundary conditions is used in Fluent for further analysis.

5. Results and discussions

5.1 Angle of attack =0°

Figure 6 shows the air flow over NACA 0015 airfoil at AOA = 0 °. The Flow and its visualisation can be improved further by increasing the density of the mesh. Figure 7(a) represents the Velocity Profile and figure 7(b) represents the pressure profile over the NACA 0015 airfoil at AOA = 0^o. It can be observed that the velocity and pressure profiles are symmetric about the camber line when AOA=0^O. Same can be confirmed from the real time flow around the airfoil (figure 6) as the white smoke used for analysis is flowing along the surfaces of the airfoil. No separation of flow has been observed at the trailing edge of the airfoil.

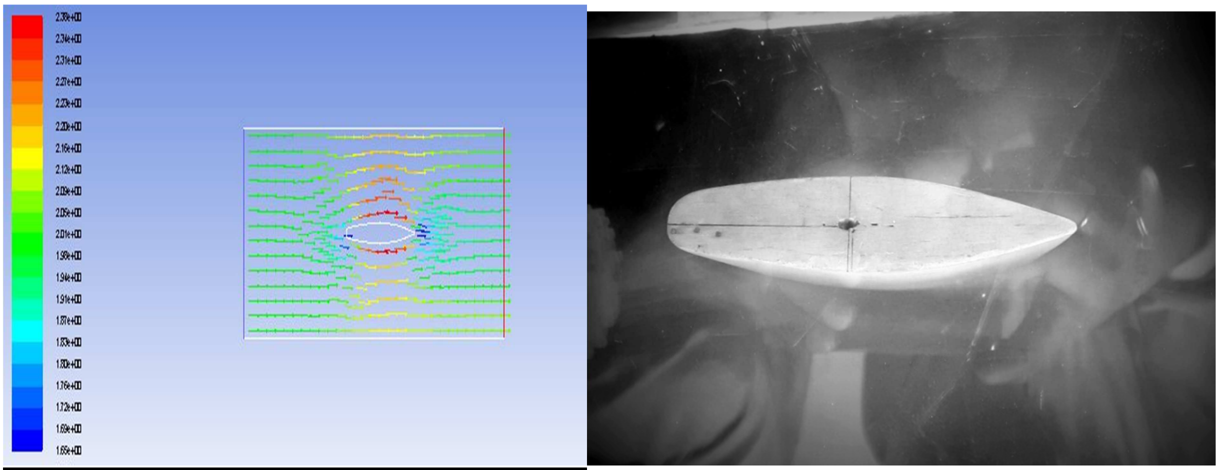


Fig 6 Real time and CFD generated air flow over NACA 0015 airfoil at AOA=0°

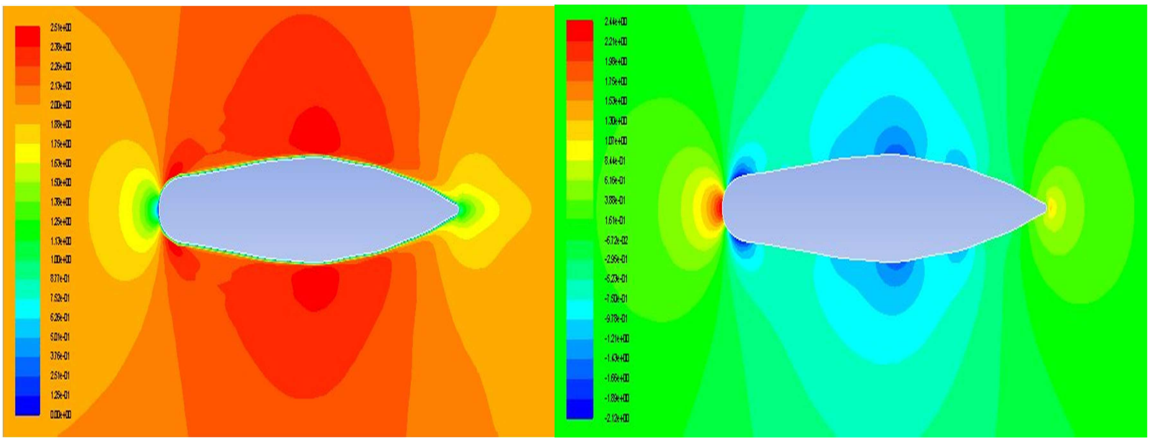


Figure 7(a) Velocity profile over NACA 0015 at AOA=0°   Figure 7(b) Pressure profile over NACA 0015 at AOA=0°

5.2 Angle of attack = 5^o

Figure 8 shows the air flow over NACA 0015 airfoil at AOA = 5°. Figure 9(a) represents the Velocity Profile and figure 9(b) represents the pressure profile over the NACA 0015 airfoil at AOA = 5^o. The velocity at the upper surface of the airfoil is greater than the velocity at the power surface of the airfoil, whereas the pressure at the lower surface of the airfoil is greater than the pressure at the upper surface of the airfoil. Due to this difference in pressure airfoil experiences a lift force in the upper direction. A very wake formation can be observed at the trailing edge of the airfoil.

5.3 Angle of attack =10^o

Figure 10 shows the air flow over NACA 0015 airfoil at AOA = 10°. Figure 11(a) represents the Velocity Profile and figure 11(b) represents the pressure profile over the NACA 0015 airfoil at AOA=10^o. A larger difference in velocity and pressure between upper and lower surfaces of the airfoil can be observed in figure 11(a) and figure 11(b) respectively. The velocity at the trailing edge of the airfoil is minimum and this may be because of the flow separation at the trailing edge which leads to the wake formation at the trailing edge of the airfoil. Same can be observed from the real time diagram in figure 10.

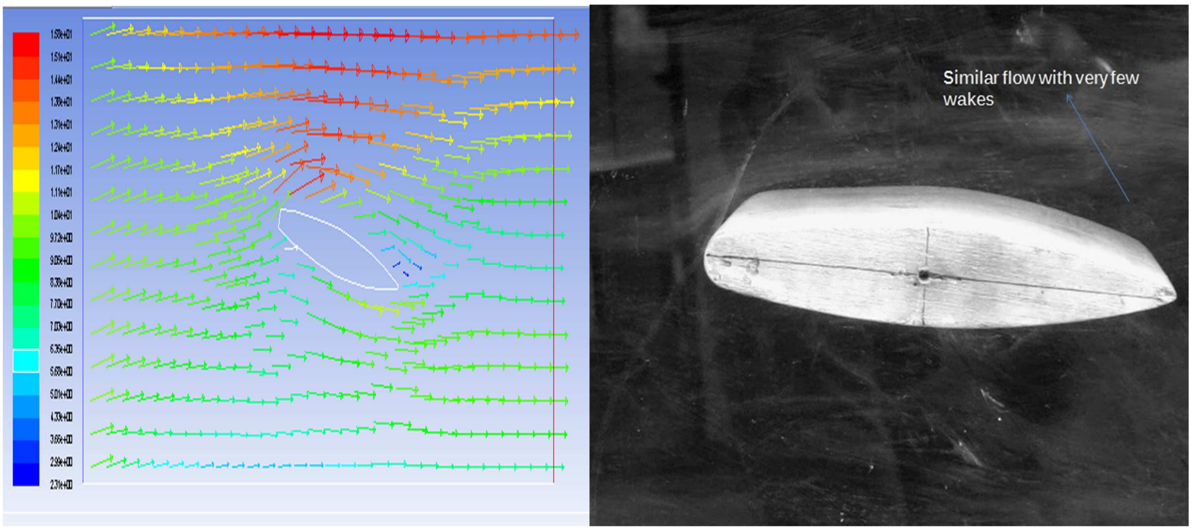
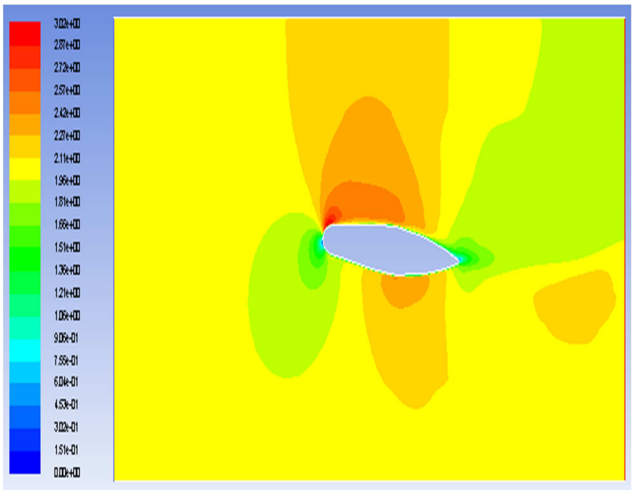


Figure 8 real time and CFD generated air flow over NACA 0015 airfoil at AOA= 5°

Vol.2 Issue.7,
July 2014.
Pgs: 84-93



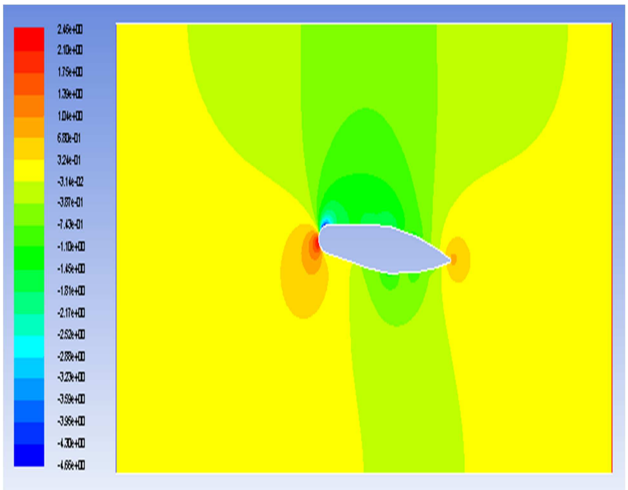


Figure 9(a) Velocity Profile over NACA 0015 at AOA= 5°   Figure 9(b) Pressure Profile over NACA 0015 at AOA= 5°

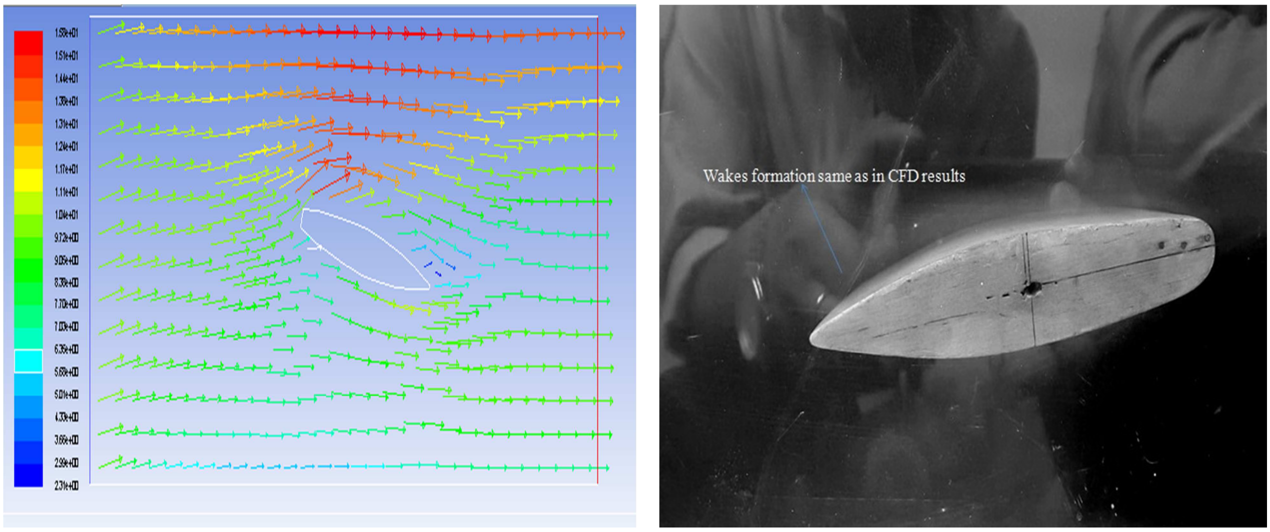


Fig 10 real time and CFD generated air flow over NACA 0015 airfoil at AOA= 10°

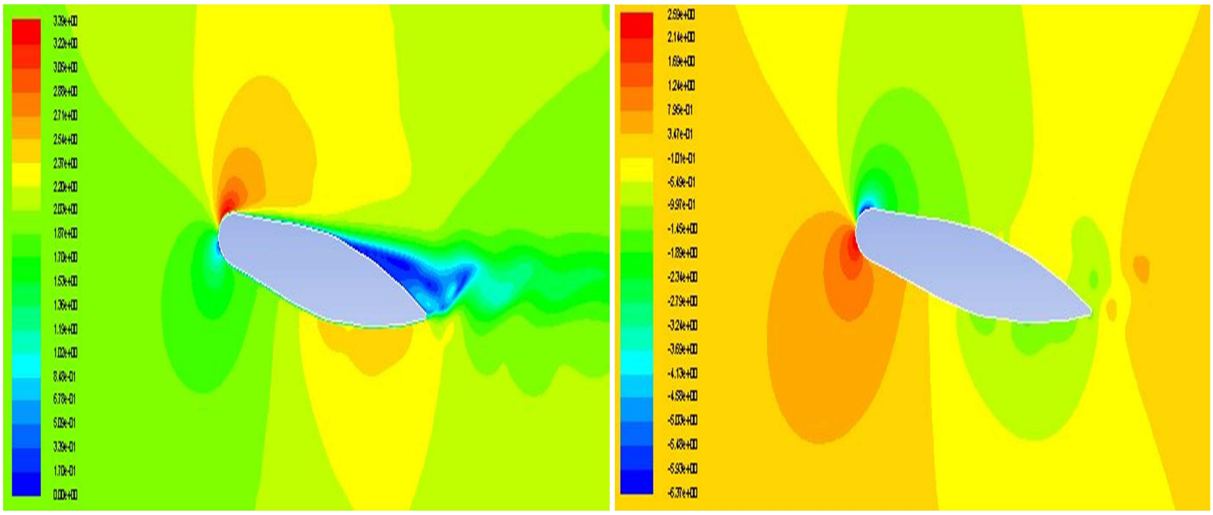


Figure 11(a) Velocity Profile over NACA 0015 at AOA=10° Figure 11(b) Pressure Profile over NACA 0015 at AOA= 10°

5.4 Angle of attack 15°

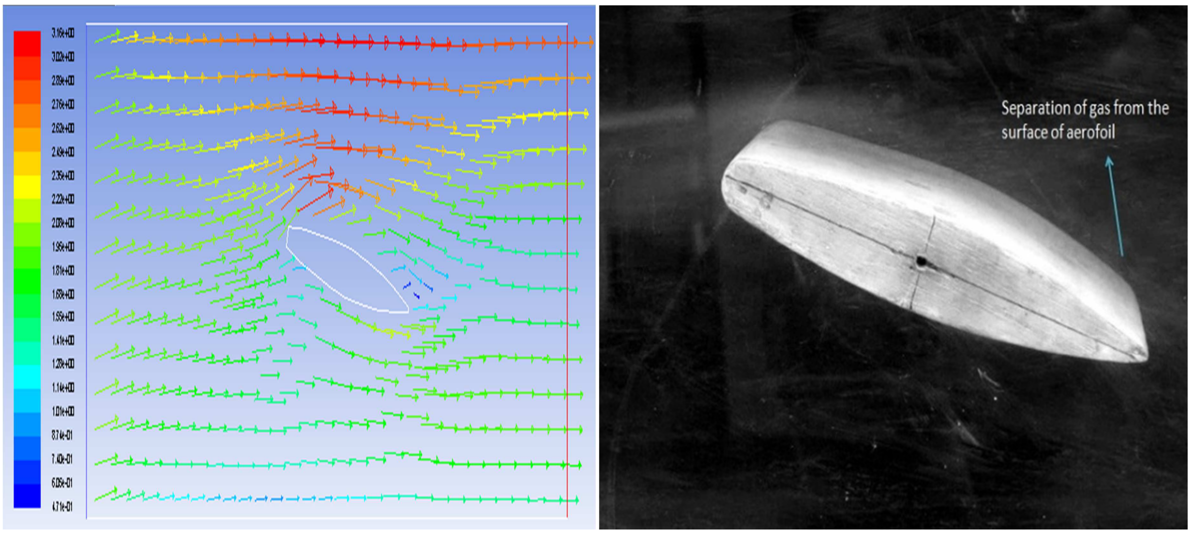
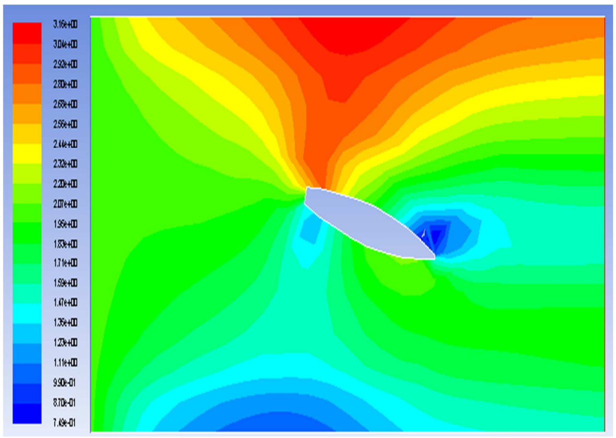


Figure 12 Real time and CFD generated air flow over NACA 0015 airfoil at AOA= 15°

Vol.2 Issue.7,

July 2014.

Pgs: 84-93



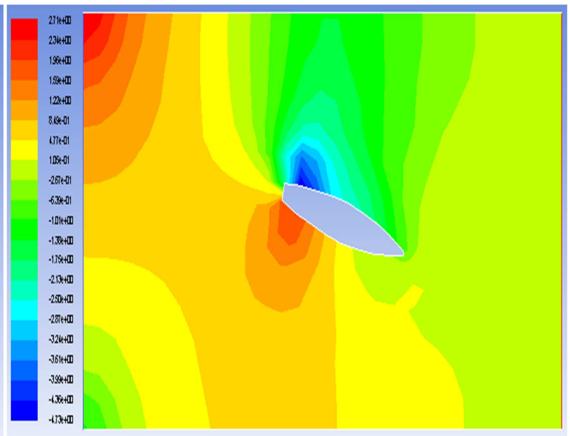


Figure 13(a) Velocity Profile over NACA 0015 at AOA=15°

Figure 11(b) Pressure Profile over NACA 0015 at AOA= 15°

Figure 12 shows the air flow over NACA 0015 airfoil at AOA = 15°. Figure 13(a) represents the Velocity Profile and figure 13(b) represents the pressure profile over the NACA 0015 airfoil at AOA = 15^{o}. It can be observed that the velocity is highest at a point on upper surface of the airfoil close to its leading edge and at the same point the pressure is minimum. This increase the lift further but on the same side the drag force also increases as the frontal area of the airfoil increases. The separation of flow can also be observed at the upper surface of the airfoil in figure 12.

6. Conclusion

The behaviour of the NACA 0015 airfoil at different angles of attack has been studied using CFD and the same has been confirmed by the real time flow analysis in the experimental wind tunnel. Flow has been found to depart(flow separation) from the upper surface with the increase in angle of attack. Due to the flow separation a region of wake formation is observed at the trailing edge and the intensity of wake increases with the increases in angle of attack. The lift force will also increase as the pressure difference between the upper and lower surface increases with increase in angle of attack.

References

http://airfoiltools.com/airfoil/naca4digit

Karna SP, Saumil BP, Utsav BP, Ahuja AP, CFD Analysis of an Airfoil, *International Journal of Engineering Research*, 2014,vol. 3, issue 3, pp 154-58.

Mevadiya M, CFD Analysis of Pressure Coefficient for NACA 4412, *International Journal of Engineering Trends and Technology*, 2013, vol. 4, issue 5, pp 2041-43.

Prabhakar A,CFD analysis of static pressure and dynamic pressure for NACA 4412, *International Journal of Engineering Trends and Technology*,2013, vol 4, issue 8, pp 3258-65.

Shankara MHM, 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*, 2013, vol. 2, issue 7, pp 3137-45.

Author Biography

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.