Effects of Leading-edge Tubercles and Dimples on a Cambered Airfoil and its Performance

J.S.S Manideep[1], Levin Curtis Nixon[2], Koganti Naga Sai Ramu[3], Ranjitha Eswaran[4]

School of Aeronautical Sciences, Hindustan Institute of Technology and Science, Tamil Nadu, India.

Abstract - A computational fluid dynamics (CFD) analysis is used in this research study to explore the combined impacts of tubercles and dimples on the Wortmann FX 60-126 airfoil. The primary goal of adding Tubercles and Dimples to the wing is to create vortices by causing turbulence(s) and so delaying boundary layer separation, resulting in a decrease in pressure drag and a rise in stall angle. The flow analysis was performed after the basic wing was modified with four different variables of wavelengths and amplitudes for tubercles. The precise point of boundary layer separation on the tubercle added wing was determined, and semi-sphere inward dimples were introduced at those locations. The airfoil was then tested with various angle of attacks (0,5,10,15 and 20) with an inlet velocity of 80m/s and the required results were achieved. This analysis supports the tubercle and dimple effect by increasing the L/D ratio and so achieving optimum aerodynamic efficiency, and thus improving the aircraft performance. The wing with the wavelength of 50mm and amplitude of 10mm had produced the better results comparatively to the other wings.

Keywords: Bio-mimicry, Flow separation, CFD, Dimples, Tubercles, Angle of Attack, Wavelength, Amplitude

1. Introduction

Does addition of Tubercles or Dimples improve aircraft performance? For a long time, engineers have been fascinated by bio-mimicry, particularly in the field of aeronautics. A plethora of research has been conducted on how the addition of components such as tubercles on humpback whales and the semi sphere inward dimples of a golf ball can be beneficial for improving the performanceof the aircraft and the results suggests that the tubercle/dimple effect operates by diverting flow over the airfoil into narrower streams by increasing velocities and so delaying boundary layer separation and the bright side of this finding states that this additional wing component reduces pressure drag by creating wingtip vortices and considerably increases lift force. Tubercles [3] can stabilize flow by generating vortices, the drag increases slightly in less challenging flow conditions.

Our research is based upon the combined effects of tubercles and dimples on a cambered airfoil and how this addition will impact the performance of the aircraft. A lot of research papers on tubercle or dimple addition on symmetrical airfoils are available but our focus is towards studying the impact of these additional wing components on a cambered airfoil. Following a Grid Independence study and a validation case derived from existing research papers on symmetrical airfoil with tubercles and dimples, the viscous Computational Fluid Dynamics (CFD) approach was applied for the investigations.

1.1 Flow Separation

Flow separation is what needs to be avoided on any surface which is subjected to airflow. Flow separations happen due to change in pressure gradient. The pressure gradient increases as it moves along the surface, there comes a point where the pressure gradient increases drastically. This certain point is the location where the flow starts to separate. The reason why it should be avoided is Pressure drag. When the flow separates from the surface, it starts to form low pressure zone. This causes drag, hence called pressure drag. Flow separation limits the aircraft from flying at higher angle of attacks. This causes stalling of the wing. Wing modifications can be done to delay flow separation. Tubercles on the leading edge/ trailing edge, Vortex generators and upward dimples/downward dimples delay the flow separation from the surface. A lot of research has been going on these modifications. In this particular study, Tubercles on the leading edge and dimples on the surface have been added for better performance of the wing. This way, the wing will be able to produce desired lift coefficient values at higher angle of attacks.

1.2 Boundary Layer

The viscosity of the fluid features a complex effect on aerodynamic forces. Because the fluid passes past the thing, the molecules near the surface adhere there to the molecules immediately above the surface are stalled by collisions with the molecules on the surface. These molecules, in turn, impede the flow directly above them. The more aloof from the surface one moves, the less collisions are laid low with the item surface. This leads to the formation of a skinny layer of fluid near the surface, within which the speed changes from zero at the surface to the free stream value distant from the surface. This layer is thought because the physical phenomenon because it exists on the fluid's boundary. The intricacies of the flow within the physical phenomenon are critical for several aerodynamic difficulties, like wing stall, skin friction drag on an object, and warmth transfer in high-speed flight. Looking on the worth of the Reynolds number, boundary layers will be laminar (layered) or turbulent (disordered). The physical phenomenon is laminar at lower Reynolds numbers, and also the streamwise velocity changes evenly. The physical phenomenon becomes turbulent at increasing Reynolds numbers, and therefore the streamwise velocity is defined by unstable (changing with time) swirling flows inside the physical phenomenon. The external flow reacts to the boundary layer's draw close the identical way that it might to an object's physical surface. As a result, the physical phenomenon provides the article an "effective" shape that differs somewhat from its physical shape. The physical phenomenon may lift or split from the body, leading to a good shape that differs greatly from the particular shape. this happens because the flow near the boundary has extremely little energy (in comparison to the free stream) and is more easily driven by pressure changes. The explanation for wing stall at high angles of attack is flow separation.

1.3 Effect of Tubercles on Boundary Layer:

The leading-edge shape of the humpback whale flipper inspired the concept of using leading edge protrusions or tubercles for passive flow control. Miklosovic et [1] colleagues discovered that in comparison to identical model with a smooth leading edge, an idealised scale model whale flipper with tubercles achieves a better maximum lift coefficient and bigger stall angle with low drag penalty. Experimental analysis of finite wing models by L. Howle and colleagues[2] have demonstrated that the presence of tubercles produces a delay within the angle of attack until stall, thereby increasing maximum lift and decreasing drag. Leading-edge bumps on flippers or wings are compared to vortex generators, which are small objects placed on a wing that introduce momentum into the physical phenomenon (i.e., make it turbulent) so as to postpone flow separation. However, because the wavelength and amplitude of the tubercles are significantly larger than the physical phenomenon thickness, it's unlikely that they operate as vortex generators. We propose a completely unique mechanism during which the tubercles modify the pressure distribution on the wing, causing the separation of the physical phenomenon to be delayed behind the tubercles. This eventually ends up in a decreased pressure drag and a bigger stall angle. Another study [4] discovered that the presence of tubercles improves the airfoil performance by delaying or perhaps preventing stall within the investigated range of operating conditions (α and Re). The drag performance of airfoils with and without tubercles is remarkably similar at low angles of attack. When approaching the stall angle, the airfoil with tubercles sees a greater rise in drag than the unmodified airfoil. However, after stalling, the redesigned airfoil has less drag. The tubercles may offer more obvious benefits for airfoils with sweep and/or taper where there's a way larger amount of span-wise flow. the actual fact about tubercles is that it doesn't achieve significant performance enhancement at low Reynolds numbers, except post stall.

1.4 Effect of Dimples on Boundary Layer:

To know the effect the dimples on the wing, we need to know the effect of dimples on a golf ball. If you observe,the golf ball has dimples on it. These dimples, contribute in decreasing the pressure drag of the golf ball and helps the ball travel more distance. The mechanism is very much similar to the tubercles. When the dimples are introduced to the wing, as the flow passes over them, vortices are generated. These vortices having more energy, traps the laminar flow inside of them and does not allow the flow to separate from the surface of the wing preventing from pressure drag.

Airfoil performance prediction is important in advanced rotor design and performance development techniques. There are three ways to analyze the texture of a wing. Numerically, analytically and experimentally. In particular, numerical analysis based on computational fluid dynamics (CFD) software reduces cleanliness, time consumption, installation costs and more. Numerical simulation analysis of the aerodynamic performance of wind turbine airfoil focuses primarily on the effects of mesh density and turbulence models. , Leading edge roughness, profile camber and Reynold’s number.

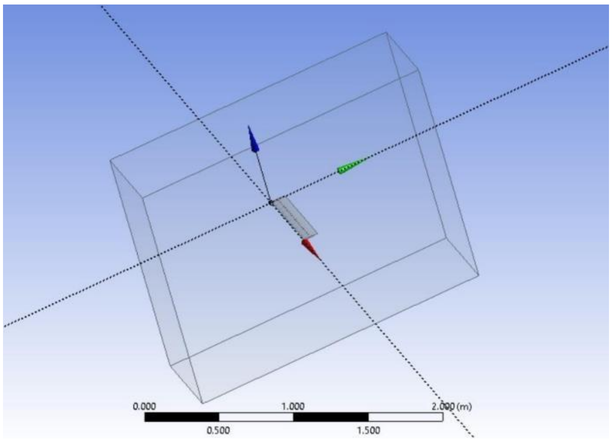
1.5 Airfoil Selected:

The airfoil chosen for the study is Wortmann FX-60-126. After the comparative studies has been done among the airfoils, this came out to be the better performing of all. FX series has been taken because very few studies have been done on this airfoil. The first two terms denotes the name of the Aerodynamicist. FX stands for the Aerodynamicist that designed this wing. The second two terms denotes the year of the design. 60 is the year of the design which is 1960. The last 3 digits of the airfoil denotes the location of the maximum thickness with respect to chord i.e 10^{th} of the chord. For this airfoil, the maximum thickness is located at 12.6% of the chord. The coordinates of this airfoil are taken from the website “airfoil tools” and has been converted to 3D using CATIA V5.

2. Methodology:

2.1 Numerical Analysis

There are three numerical methods for discretizing the mass, momentum, and energy of the corresponding differential equation. These are finite different method (FDM), finite element method (FEM) and finite volume method (FVM). Wind turbine flow, wing and blade simulation is misunderstood with CFD software flow or CFX leaching. Use FVM in fluent or CFX solver numerical analysis. In this study, the fluent solver is used by ANSYS software to solve the performance of the airfoil.



2.2 CFD Modelling and Meshing Geometry:

Airfoil geometry is created or imported as the first step in a numerical simulation. Wing coordinates are obtained froman existing wing database. Airfoil geometry sketches are imported by other modeling software CATIA V5. Geometry can also be created with the Ansys software Design Modeler.

The Wortmann FX 60-126 airfoil is created at different angle of attack with the Ansys software. The different angle of attack are 0, 5, 10, 15, 20. The chord length for the wing is 100mm and the span of the wing is 500m

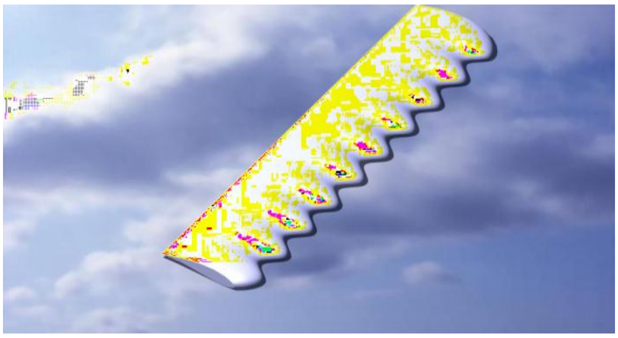


Fig-1 (a): W50A10

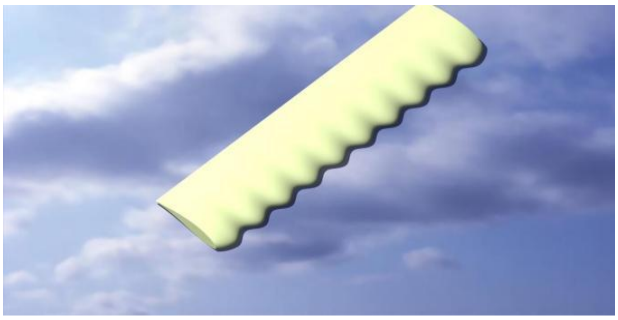


Fig-1 (b): W50A20

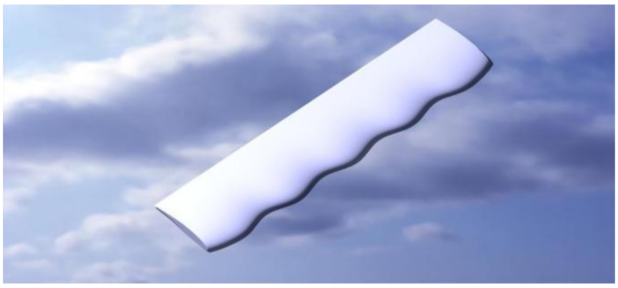


Fig-1 (c): W100A10

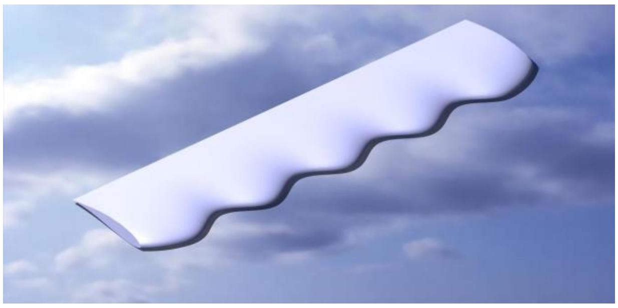


Fig-1 (d): W100A20

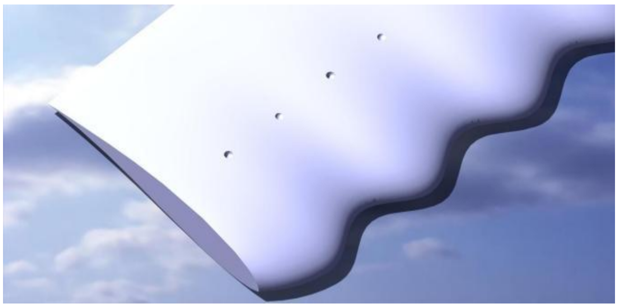


Fig-1 (e): Wing with Tubercles and Dimples

Wavelength	mm	Amplitude	mm	
10 % of Span	50	10 % of Chord	10	W50A10
		20 % of Chord	20	W50A20
20 % of Span	100	10 % of Chord	10	W100A10
		20 % of Chord	20	W100A20

2.3 Meshing:-

The mesh is run to create individual boundary conditions on the fluid domain interface. The size function is accessibility, and the center of curvature and association is good. Fine mesh gives good results, but it takes too long. After the mesh is complete, the Fluent solver predicts wing performance.

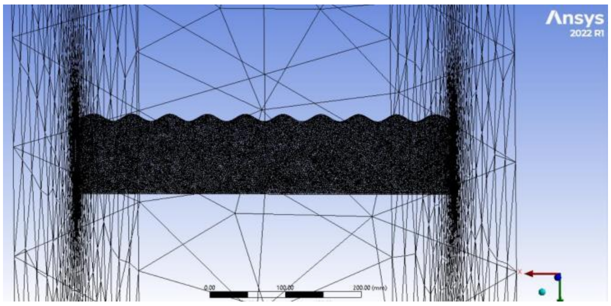


Fig-2(a): Fluid Domain Meshed Model

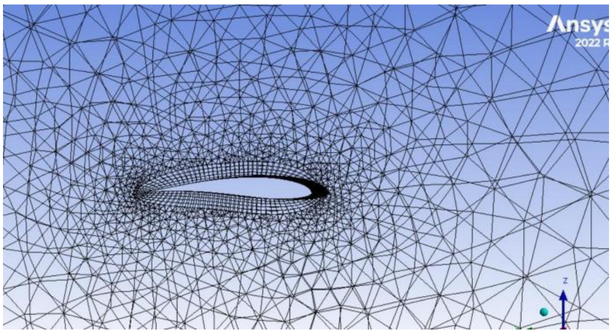


Fig-2(b): Elevation view mesh refinement in near wallregion

2.4 CFD Governing Equation

Present study is conducted at chord based Reynolds Number Re=120000, the corresponding upstream velocity at the chord length of 100mm is 80m/s at sea level conditions. As at this value of free stream velocity. Therefore the continuity equation and momentum equation for the incompressible flow are given as follow.

$$\frac{\partial}{\partial t}(\rho u_i)+\frac{\partial}{\partial x_j}(\rho u_i u_j)=-\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}-\frac{2}{3}\delta_{ij}\frac{\partial u_i}{\partial x_i}\right)\right]+\frac{\partial}{\partial x_j}\left(-\rho u_i' u_j'\right)$$

Solver	Pressure-based
Simulation type	Steady
Fluid material	Air
Temperature	288 K
Kinematic Viscosity	1.76e-5m m^2/s
Interpolating scheme	Pressure (standard) Density (Second order Upwind) Momentum (Second order upwind) Turbulent viscosity(Second order viscosity)
Turbulence model	k-⍰, SST
Inlet velocity	80m/s
Reference area	0.575, 0.10866

3. Result & Discussions

This model calculation has to be carried out for Lift and Coefficient of Lift as well as for Drag and co efficient of drag.

Available Data:

Angle of Incident (α) = 10 degree

Lift (L) = 251.07 N

Drag (D) = 36.698 N

Calculation for C_L and C_D

Velocity V = 80 m/s

Density ρ = 1.225 kg/m^3

Surface Area (s) = 0.052945 m^2

Lift (L) = 251.07 N

Drag (D) = 36.698 N

Co efficient of Lift (C_L) = 2L / ρv^{2}s = 1.22

Co efficient of Lift (C_D) = 2D / ρv^{2}s = 0.17832

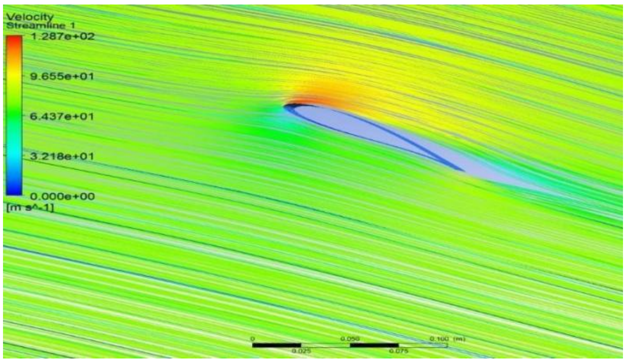


Fig-3(a): Streamline Flow Over Wing (Plane Wing)

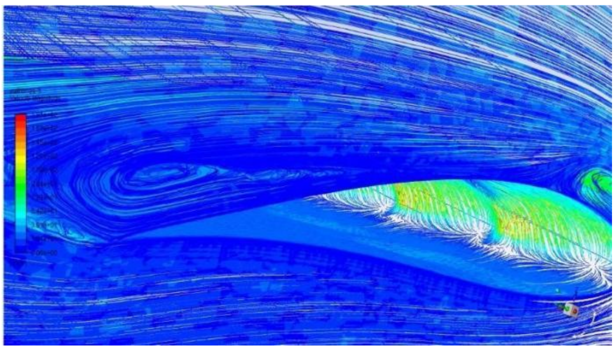


Fig-3(b): Streamline Flow Over Wing (W50A10)

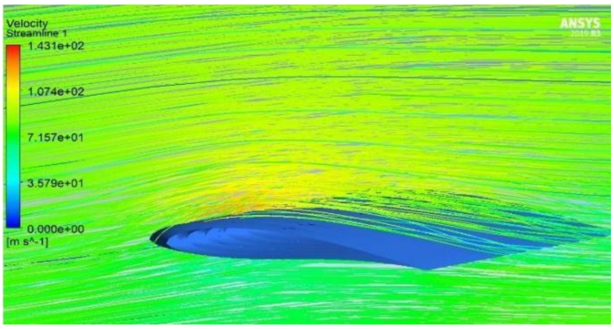


Fig-3(c): Streamline Flow Over Wing (Dimple Wing)

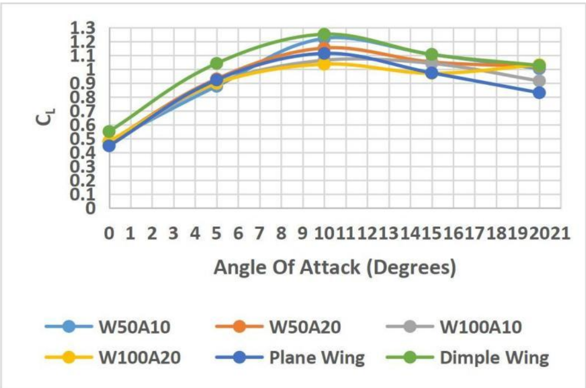


Fig-3(d): C_L vs Angle of Attack

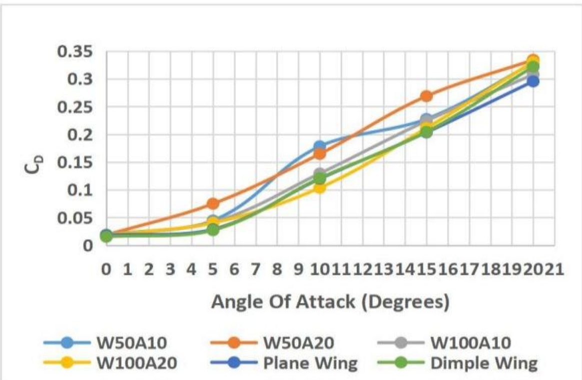


Fig-3(e): C_D vs Angle of Attack

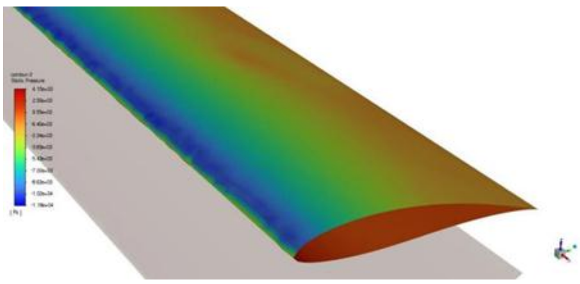


Fig-3(f): Pressure Contour of Plain Wing

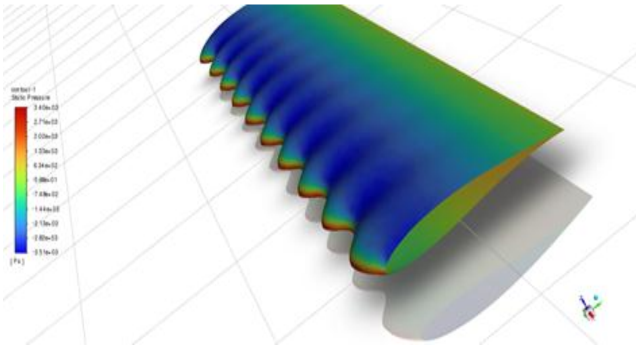


Fig-3(g): Pressure Contour of Tubercle Wing

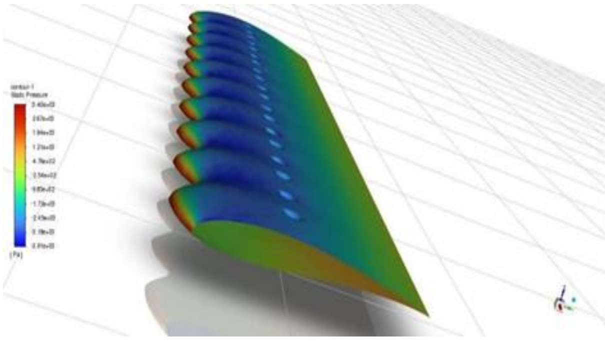


Fig-3(h): Pressure Contour of Tubercle+Dimple Wing

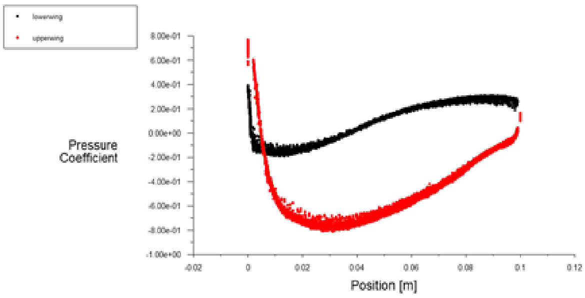


Fig-3(i): Pressure Coefficient Graph of Plane Wing at 0 AOA

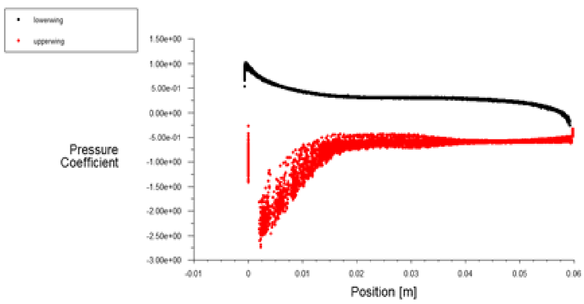


Fig-3(j): Pressure Coefficient Graph of Plane Wing at 20 AOA

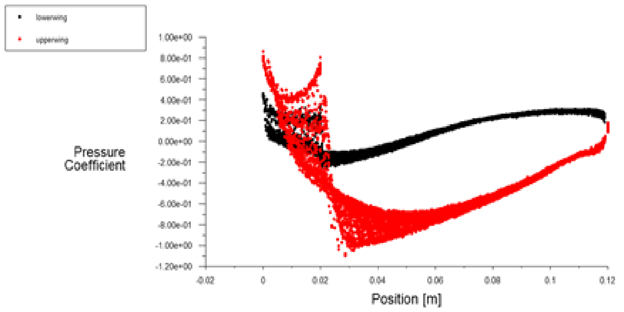


Fig-3(k): Pressure Coefficient Graph of W50A10 wing at 0 AOA

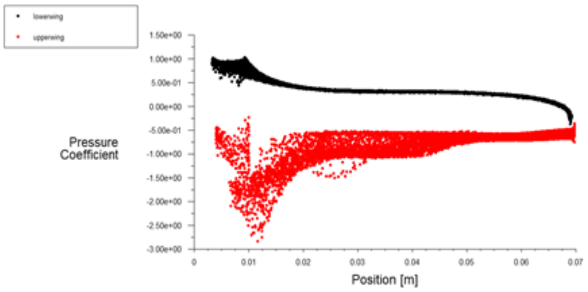


Fig-3(l): Pressure Coefficient Graph of W50A10 wing at 20 AOA

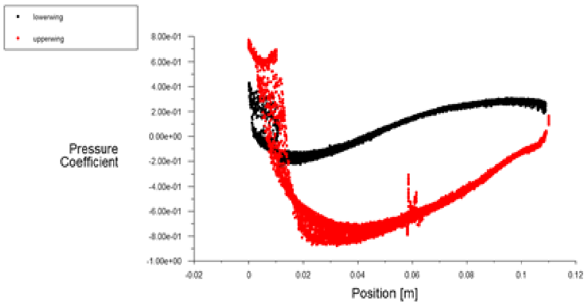


Fig-3(m): Pressure Coefficient Graph of W50A10 wing with Tubercles and Dimples(together) at 0 AOA

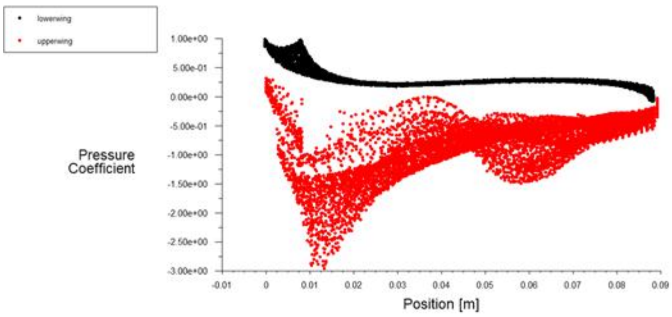


Fig-3(n): Pressure Coefficient Graph of W50A10 wing with Tubercles and Dimples(together) at 20 AOA

4. Conclusion

All of the above-mentioned configurations have been subjected to analysis. The findings for the various settings have been provided in tables and has been represented graphically.

We discovered that the configuration of the aircraft wing with tubercles that have wavelength (50mm) and amplitude (10 mm) has offered the best performance in terms of maximum Angle of attack and coefficient of Lift with a considerable amount of Drag, and then this tubercle added wing was further modified by adding dimples at the point where the boundary layer began to separate.

On Analysis we found that the aircraft wings with tubercles and dimples (together), are more efficient than the aircraft's plain wing. The wing's performance has also increased, witha remarkable improvement in the lift force.

5. Future work

1) In the future, the analysis can be replicated by altering the wavelength and amplitude as well as the forms (shapes, geometry, patterns) of the tubercles.
2) Structural analysis of the model can be done in the near future.
3) The Geometry of the model can be modified by adding struts to the wing and analysis can be performed.

6. References:

1) D. S. Miklosovic, M. M. Murray, L. E. Howle, and F. E. Fish, "Leading-edge Tubercles Delay Stall on Humpback Whale (Megaptera novaeangliae) Flippers", Physics of Fluids, Vol. 16, No. 5, May 2004.
2) F. E. Fish, P. W. Weber, M. M. Murray, and L. Howle, "The Tubercles on Humpback Whales' Flippers: Application of Bio-Inspired Technology", Integrative and Comparative Biology, Vol. 51, No. 1, pp. 203-213, May 15, 2011.
3) J.Andreas, K.Theobald, B.Kerschgens, "CFD simulations of the flow around a humpback whale's pectoral fin", 17th biennial conference on the biology of marine mammals - Capetown, South Africa 29th Nov. -3rd Dec.
4) Ahmed Farouk Abdel Gawad Transaction On control and mechanical systems,"Utilization of Whale-Inspired Tubercles as a Control Technique to Improve Airfoil Performance", Vol. 2, No. 5, PP.212-218,May,2013.

5) Vishal Kaushik, Manoj Mahore, Sandeep Patil, “Analysis of Dimpled Wing of an Aircraft” , IJEDR 2018| Vol 6, Issue 3.

6) K.L.Hansen , R.M.Kelso and B.B.Dally, “An Investigation of Three Dimensional Effects on the Performance of Tubercles at Low Reynolds Number”, 17th Australasian Fluid Mechanics Conference Auckland, New Zealand 5-9 December 2010

7) Yin Yin Htay Aunga*, Myat Myat Soeb, Aung Myat Thuc. “Effect of Attack angle on Aerodynamics Analysis of Different Wind Turbine Wings using Numerical Simulation,” American Scientific Research Journal for Engineering, Technology, and Sciences (ASRJETS).

8) Hardeep singh laal, Jaithra khanna, harsh khatri, Sahil mhapankar, parmeswar paul, Anjali A vyaas. “ Effect of tubercles on aerodynamic performance of NACA 0015,” IRJET volume 6 issue 3, 2019.

9) A. Gross and H. F. Fasel. “ Numerical Investigation of Different Wind Turbine Airfoils,” AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition, Oriando, Florida, 2011.

10) D. Hartwanger and A. Horvat. “ 3D Modelling of a Wind Turbine Using CFD,” Engineering Simulation: Effective Use and Best Practice, Cheltenham, United Kingdom, 2008

.