10.48047/jocaaa.2016.20.02.01

ANALYSIS OF FLOW CONTROL ON AIRFOILS

P. Ravikanth Raju

Dept of Mechanical Engineering, Malla Reddy College of Engineering & Technology, Secunderabad - 500100, India

Abstract— **Flow separation is among the most significant problems in aerodynamics. At present, different kinds of surface modifications on airfoil are being studied to improve the maneuverability of the aircraft by delaying flow separation to higher angle of attack. The present work describes change in aerodynamic characteristics of an airfoil by applying certain surface modifications in form of dimples. At first surface modifications that are considered here are outward and Inward dimples on the airfoil model. A comparative study showing variance in flow separation of modified airfoil models at different angle of attacks can be done. The airfoil profile considered in the present study is NACA-2412 with uniform cross-section throughout the length of airfoil. ANSYS ICEMCFD and CFX are used for simulations. Results has been extracted using ANSYS CFD POST with Velocity streamlines, Mach plots etc.**

Keywords—dual bell nozzle; CFD; secondary injection.

INTRODUCTION

Airfoils are the streamlined bodies which produce lift and less drag in the form of wings for an aircraft. As the angle of attack increases lift increases till stalling angle after which flow separates and pressure drag increases. During maneuvers aircraft travels at high angles of attack. Considering this problem surface modifications like dimples are being considered in the given study to avoid flow separation. Till now these have been ignored because dimples help in reduction of pressure drag. In case of aerodynamic bodies pressure drag is very little in streamlined bodies compared to bluff bodies. An airfoil is streamlined body so dimples do not affect to its drag much at zero angle of attack, but as soon as airfoil attains some angle of attack, wake formation starts due to boundary layer separation. Application dimples on aircraft wing model works in same manner as vortex generators. They create turbulence which delays the boundary layer separation and reduces the wake and thereby reducing the pressure drag. This also assists in Lift of the aircraft. Most importantly this can be quite effective at higher angle of attack and also can change angle of stall to a great extent.

The purposes of aircraft flow manipulation, as Gad - el -Hak [1] explains, are : increasing lift, reducing drag and enhancing the mixing of mass, momentum and energy. In order to meet these objectives 1) the laminar-to-turbulent transition has to be postponed or provoked, 2) the flow separation has to be avoided or initiated, 3) the flow turbulence has to be prevented or encouraged. Stall is a condition in aerodynamics and aviation where the angle of attack increases beyond a certain point such that the lift begins to decrease. The angle at which it occurs is called the critical angle of attack or angle of stall. Flow separation begins to occur at small angles of attack while attached flow over the wing is still dominant. As angle of attack increases, the separated regions on the top of the wing increase in size and hinder the wing's ability to create lift. At the critical angle of attack, separated flow is so dominant that further increases in angle of attack produce less lift and vastly more drag.

Stalling is the strong phenomena during Landing because reduction in dynamic pressure has to be compensated by increasing the angle of attack. After passing the critical angle of attack means the wing is now unable to produce sufficient lift to balance weight, if this angle exceeds it leads to flow separation, thereby increase in drag, which reduces the L/D ratio [2]. On analyzing the golf ball, the aerodynamics present in the dimple over the ball results in experiencing drag force smaller than the smooth surfaced ball. In deep, dimples delay the flow separation point by creating turbulent boundary layer by reenergizing potential energy in to kinetic energy [3]. Modifying the aircraft wing structure by means of placing dimples will reduce the drag to considerable amount from the total drag and helps to stabilize the aircraft during stall. In this paper, over NACA 2412 airfoil at the effective location to delay flow separation point. Aerodynamic analysis for this airfoil is carried out using Computational Fluid Dynamics (CFD). Through this study we aim at making aircrafts more maneuverable by implementing dimples over the wing.

COMPUTIONAL METHODOLOGY

To analyze any problem in CFD the geometry of flow domain plays a major role. So for the present problem NACA 2412 airfoil with three different dimple geometries has been designed in ANSYS ICEM CFD as shown in figures below.

10.48047/jocaaa.2016.20.01.01

effects of dimples as a conventional vortex is studied computationally, dimples are quite effective at different angle of attack and also can change angle of stall to a greater extent [4].

PROBLEM DESCRIPTION

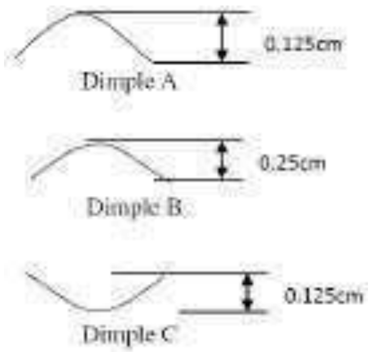


Figure 1 Dimple Configurations

In this work, the focus is on how to increase airfoil efficiency. For external wall bounded flow, like an aircraft or submarine exterior surface, boundary

layer manipulations become critical in an effort to enhance performance. It is known that flow separation leads to loss of lift (stall). If the boundary layer is turbulent, the flow is more resistant to separation, and more lift can be generated due to a delayed stall. When flow separates, a loss of lift and an increase in form drag cannot be avoided. The goal is to postpone separation so that lift is increased and drag is reduced. In order to verify the effect of dimples, the different shapes of dimples are analyzed by placing

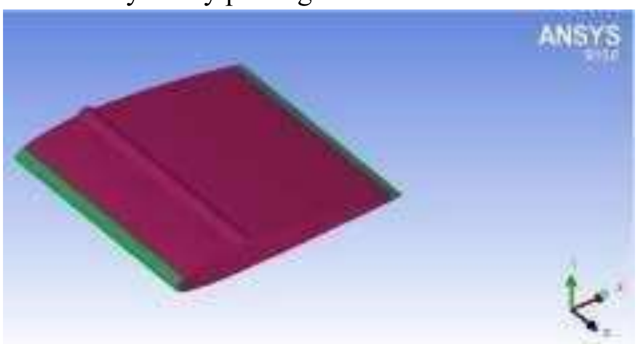
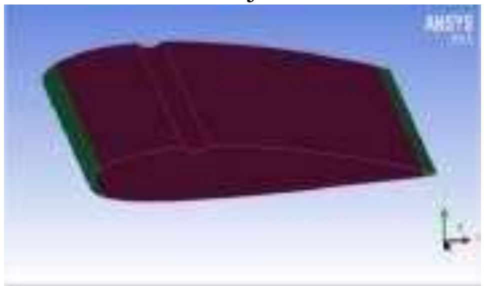


Figure.1. Outward Dimple



(a) Inward Dimple

Figure. 2. Airfoil with Dimple in ICEM CFD

After designing, meshing of flow domain is carried out ICEM CFD with Hybrid mesh. To capture the boundary layer effects i.e flow separation prismatic layers are used around the airfoil above which tetrahedra volumes are used as shown in figure below.

10.48047/jocaaa.2016.20.01.01

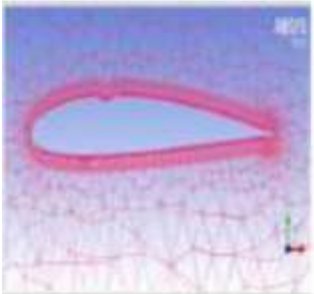


Figure. 3. Hybrid Mesh arund airfoil

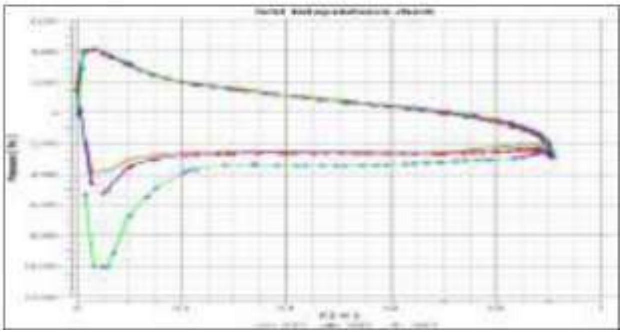
Solving the governing equations of fluid flows is done by ANSYS CFX with appropriate boundary conditions.

Table 1. Boundary conditions

S.No	Location	Boundary	Value
1	Inlet	Inlet	Velocity = 238 (m/s)
2	Outlet	Outlet	Pressure = 0 (Pa)
3	Domain	Wall	Free slip
4	Airfoil	Wall	No slip

I. GRID INDEPENDENCE STUDY& VALIDATION

Grid is the heart of CFD solutions and its accuracy and time for convergence depends on the number of volumes to be solved. Thus dividing the given fluid domain into small volumes which can give accurate solution and can capture flow physics is important. As the number of volumes increases computation time increases. So, for the present analysis over airfoil three grids are considered and best is chosen for all cases. Grid 1 is unstructured grid generated in ICEM CFD using robust octree method with 10,34,545 volumes and Grid with 15,23,617 volumes of more fine mesh and last Grid 3 of 12,42,897



volumes which is an Hybrid grid of unstructured grid as shown in figure with prismatic layers to capture boundary layer effects and flow separation are analyzed. The pressure distribution is shown in below figure.

Figure.4. Grid Independence check for NACA 2412 airfoil

From above results Grid 3 is considered as best for the present case of analysis.

The computational results of baseline were validated with that of experimental values in the *Theory of Wing Sections* [5]. The comparison of baseline lift coefficients is shown in Figure 5, and the drag coefficients are shown in Figure 6, as a function of α. The numerical results of the baseline lift coefficient are in very good agreement with those of experimental values

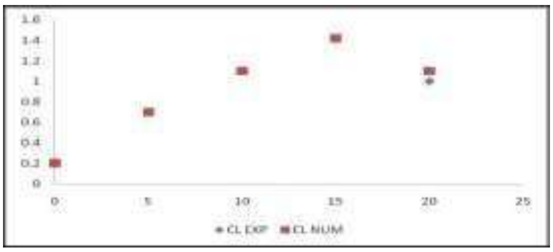


Figure. 5. Lift coefficient of 2412 airfoil as function of α

10.48047/jocaaa.2016.20.01.01

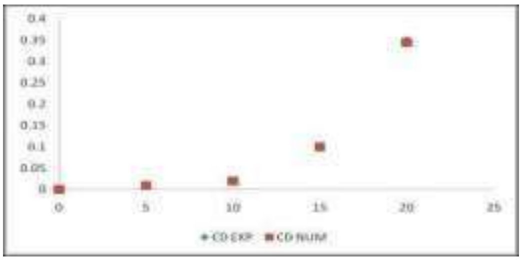
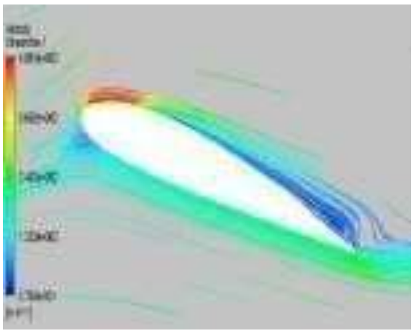


Figure. 6. Drag coefficient of 2412 airfoil as function of α

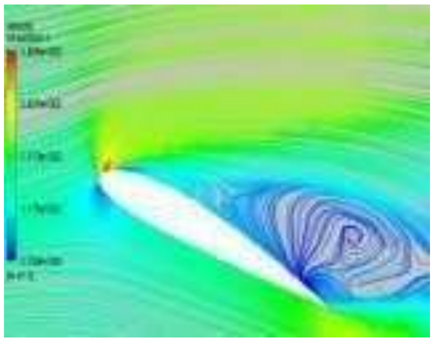
In above figure lift and drag coefficients of NACA 2412 airfoil has been calculated using ANSYS CFX and compared with results from theory of wings text book.



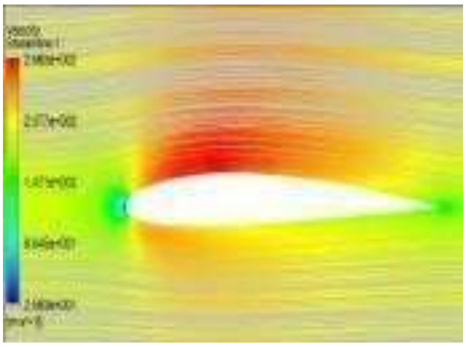
RESULTS AND DISCUSSIONS

A.Case 1 Base line Airfoil at M=0.6

In the present case the flow analysis is carried out at Mach 0.6 at various angles of attack as shown in (d) at 15 degrees AOA figure. Velocity streamline contour shows development of flow around airfoil and flow separation pattern.



(e) at



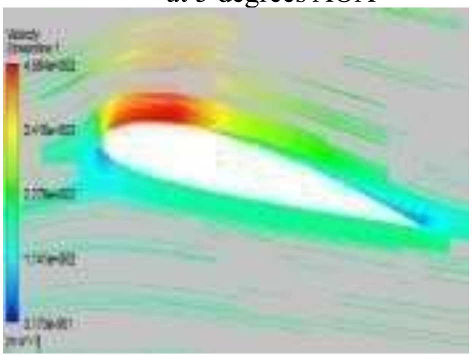
20 degrees AOA

at 0 degrees AOA

Figure.7. Velocity streamline of case1at various AOA

10.48047/jocaaa.2016.20.01.01

at 5 degrees AOA



at 10 degrees AOA

From the above figure it can be visualized that separation is taking place at 20% from the leading edge which is the crucial point. As the flow separates a wake region is formed with zero pressure regions leading to fall of lift which is known as stalling.

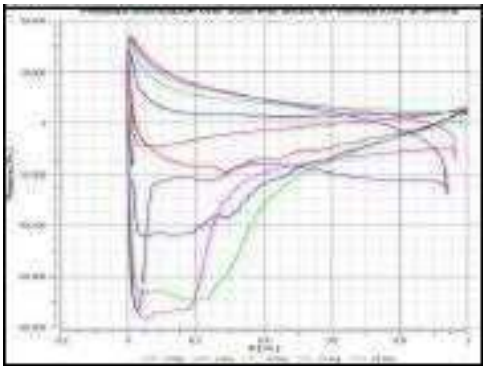


Figure.8. Pressure distribution over Base line airfoil for various AOA at M=0.6

As the angle of attack is increasing the adverse pressure gradient is moving towards leading edge as shown in the pressure distribution figure. Majorly flow behavior at 0 degrees angle of attack is shown the flow is tending to separate from trailing edge at 10 degrees angle of attack and it has been successfully separated at 15 degrees. As the speed is subsonic and very low this adverse pressure gradient cannot be altered by kinetic energy of flow which is leading to flow separation. This is very high at 20 degrees angle of attack as shown in figure.

Case 2 Dimpled Airfoil at M=0.6

In case 2 Dimple A configuration is used and the flow behavior at 20 degrees angle of attack is shown in figure 9.

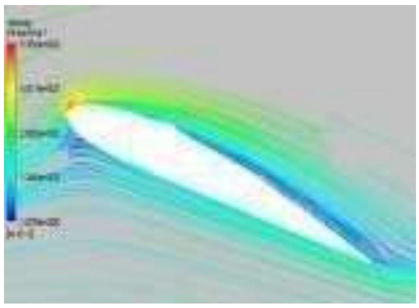


Figure.9. Velocity streamline of cas2at various 20 AOA

From the above figure separation of flow has taken but the vortex or eddy size is very less as shown in above figure. The above results of streamline pattern show a good improvement in flow

In case 3 Dimple B configurations is used and the flow in figure 11.

10.48047/jocaaa.2016.20.01.01

separation tendency compared to without dimple. The flow is almost attached to airfoil at 20 degrees angles of attack. Because of the turbulence created by dimple the kinetic of the flow within boundary layer has increased which has overcomed adverse pressure gradient.

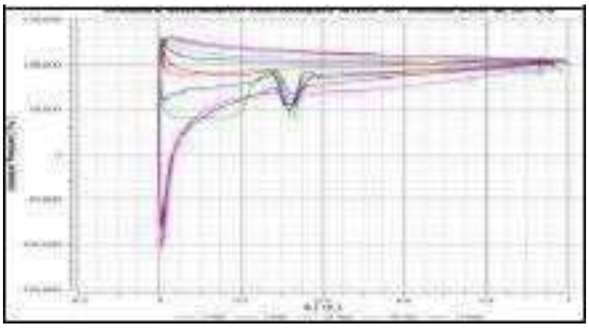


Figure.10. Pressure distribution over Dimpled airfoil for various AOA at M=0.6

Case 3 Dimpled Airfoil at M=0.6

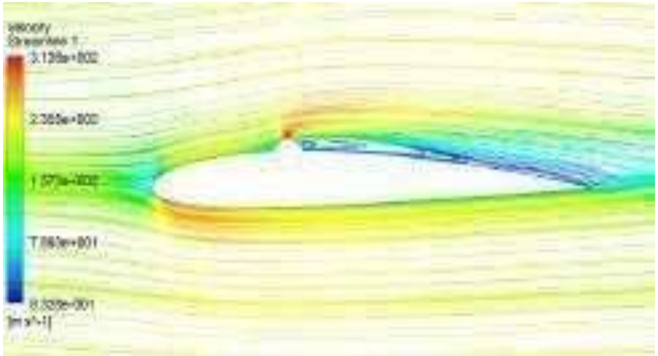


Figure.11. Velocity streamline of case3 at zero AOA

Increasing the amplitude of dimple has increased the tendency of flow separation at zero degrees. Thus further analysis for higher angles of attack is not carried out. So, an alternative for the present dimple is inward dimple and its analysis is carried out in following cases.

Case 4 Inward Dimple Airfoil at M=0.6

In case 4 Dimple C configurations is used and the flow behavior at 20 degrees angle of attack is shown in figure 12.

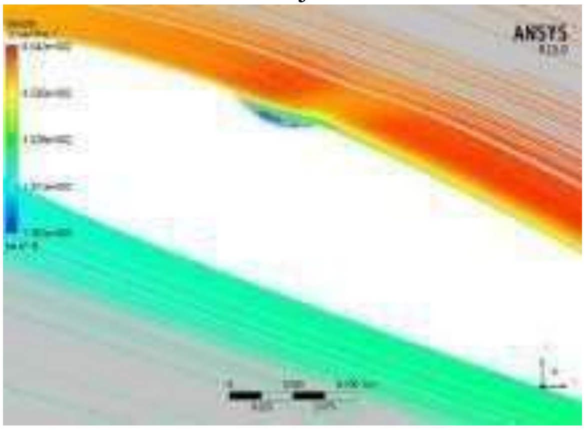


Figure.12. Velocity streamline of case4 at 20 degrees AOA

Inward dimple of same amplitude of 0.125cm has shown good results compared to other two cases of outward dimples.

10.48047/jocaaa.2016.20.01.01

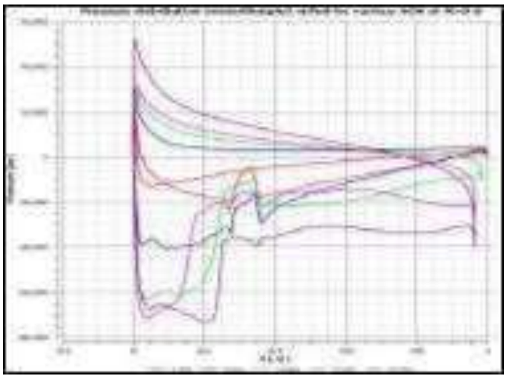


Figure.13. Pressure distribution over inward Dimpled airfoil for various AOA at M=0.6

I. CONCLUSIONS

Addition of dimples has proven to be effective in altering various aspects of the flow structure. With such significant flow structure the resultant lift and drag forces are also altered. Primarily the inward dimples are proved to be most suitable as proved by the results of this study. Downright, the total aerodynamic efficiency of the airfoil is improved by this complete idea. But for complete verification of its effect an experimental study is required. Also full scale testing focusing on aerodynamic force measurement is essential to determine the sustainability of the inward dimples to act as highly efficient vortex generators. The concept is very new and with the implementation of dimple matrix, it could be extremely beneficial in making an aircraft more maneuverable by changing flow characteristics. Also it increases the aerodynamic efficiency and therefore helps in improving the performance also. The idea will also assist in shorter take-offs at low speed.

REFERENCES

1. M. Gad-el-Hak, *Flow Control Passive: Active and Reactive Flow Management*, First ed., Cambridge University Press, 2000.
2. RENEAUX, J., ―Overview on drag reduction Technologies for Civil transport aircraft‖, European Congress on Computational Methods in Applied Sciences and Engineering ECCOMAS, 2004.
3. DAVIES, J.M., ―The aerodynamics of golf balls‖, J of Applied Physics, 1949, 20, (9), PP 821-828.
4. WU, J.Z. and WU, J.M., Vorticity dynamics on boundaries, Adv. in Appl Mech, 1996, 32, pp 119-275.IRA H ABBOTT and ALBERT E VON
5. DOENHOFF ―Theory of Wing sections‖ Dover Publications, 1959
6. ISBN 10: 0486605868 ISBN 13: 9780486605869.