IOP Conf. Series: Materials Science and Engineering **197** (2017) 012080 doi:10.1088/1757-899X/197/1/012080

Numerical simulation of dimples in airfoil using MATLAB

P. Booma Devi[1] and Dilip A Shah[2]
[1, 2] Department of Aerospace Engineering,
Hindustan University, Chennai, India.

E-mail: [1]boomajeeva@gmail.com, [2]aero@hindustanuniv.ac.in

ABSTRACT**:** The Aircraft wing is a point of important research which poses greater challenge in terms of aerodynamic efficiency. The flow separation control method is addressed in classical aerodynamics methods. This study focuses on influence of dimples on controlling the flow and also increasing the aerodynamic efficiency. The periodic process of placing the cavities on the wing starting from root to tip controls the flow separation. The linear variation of characteristic curve provides the information about the flow separation and control of flow on upper surface of the airfoil.These different shapes are utilized viz., Square, Rectangle and Triangle. The numerical simulation is carried out in using MATLAB package. Preliminary analysis on the flow separation is carried out focuses on laminar flow separation, which has the influence on the overall lift generation and drag generation.

Keywords: Flow separation, dimples, laminar flow, lift and drag.

1. INTRODUCTION:

The laminar flow under low Reynolds number aerodynamics is of greater significance for civil aircraft. Typical performances of these airfoils are characteristics by the aerodynamic efficiency. It is a well-known fact that being aerodynamic problem have greater factor for low Reynolds number. In general, the range of the aircraft angle of attack to be aerodynamic efficiency and also the overall weight characteristics of the aircraft. However for low Reynolds number less than 5×10^5. The non-linear features are responsible for the flow separation. Hence at lower speed, the greater increase in angle of attack is mandatory to produce the constant lift. The phenomenon of laminar flow separation complies with the loop found in the lift curve which further explains the reasons contributing towards flow separation. The application of various models creates region of vortices by reducing the pressure drag of the cavities.

In general, the interaction between the forces on the motion of aircraftin air is noted as aerodynamics. Lift to drag ratio is one of the important aerodynamic parameter which determines the total weight and the cost of the aircraft.[1]

In general, the range of aircraft is directly proportional to its aerodynamic efficiency at a constant fuel consumption. Increase in the efficiency of a wing for commercial usage results in reduction of the operational cost. Stalling is a phenomenon associated with flow seperation on the

IOP Conf. Series: Materials Science and Engineering **197** (2017) 012080 doi:10.1088/1757-899X/197/1/012080

upper wing surface. It is know that C_L is directly proportional to Angle of Attack. Hence at lower speed,high Angle of Attack is required to have a constant lift.[2]

The concept of introducing cavities in the wing creates turbulence which in turn delays the flow seperation(boundary layer). This reduces the region of vortices, there by reducing the pressure drag. Cavities are also called dimples, which are effective with different Angle of attack and also effectively increases the stalling angle. This can be verified by introducing different geometrical cavities over the NACA 0012 airfoil, at the flow of seperation point.[3]

Accelerated flow on the dimple surface converts the laminar flow into turbulent flow. This continuous flow translation results in the reduced dreg and also increases stalling angle. The flow separation over NACA0012 airfoil is observed and results indicate enhanced lift coefficient.[4]

To reduce the flow separation, boundary layer is energized using either leading edge slot or dimple effect. This study aims to validate, document and support the incrasein lift to drag ratio, reduction in the pressure drag and increase in stalling angle. Care is taken to ensure proper pressure coefficient distribution over the airfoil surface.[5]

The introduced cavities converge the laminar to turbulent which as the result of flow translation. The symmetrical of NACA0012 airfoil helps this transition to reduce the drag curve increase the stalling angle. This paper conducts numerical simulation on this airfoil, which incorporates dimples, using the MATLAB package. The subsequent results are then plotted for various aerodynamic parameters.

2. RESEARCH METHODOLOGY - COMPUTATION SCHEME

The unsteady and incompressible flow which is in 2D can be expressed in Cartesian form without taking in account the gravity and body force.

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial u_i}{\partial x} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{\partial \rho}{\partial x_i} + v \frac{\partial u_i}{\partial x_i \partial x_j} - \frac{\overline{\partial u_{i\, v_j}}}{\partial x_i} \tag{2}$$

Where

U_i ----- the mean velocity
γ ---- Kinematic viscosity
ρ ---- Density
P ---- Pressure
$U_i^{'} U_j^{'}$ -----Reynolds stress

2.1 TURBULENCE MODEL:

The SST k-ω turbulence model is a two- equation turbulence model which can predict precisely the adverse pressure gradient flows and airfoil flows. This effectively combines the accurate combination of turbulence.

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left(\gamma_k \frac{\partial k}{\partial x_j}\right) + \widetilde{H_k} - I_k \quad (3)$$

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_i}(\rho \omega u_i) = \frac{\partial}{\partial x_j}\left(\gamma_\omega \frac{\partial \omega}{\partial x_j}\right) + \widetilde{H_\omega} - I_\omega + G_\omega \quad (4)$$

Where,

$\widetilde{H_k}$ ---- the generation of turbulent kinetic energy due to mean velocity gradients.
H_ω ----- the generation of ω, $\gamma_k$and γ_ω represent the effective diffusivity of k and ω.
I_k andI_ω ----- the dissipation of k and ω due to turbulence.
G_ω ----- the cross-diffusion term.

The simulation representing laminar flow correction can be represented in terms of Reynolds number as follows

$$\mu_t = \alpha^* \frac{\rho k}{\omega} \quad (5)$$

$$G_\omega = \alpha \frac{\omega}{k} . \tau_{ij} \frac{\partial u_j}{\partial x_j} \quad (6)$$

$$Y_k = \rho \beta^* \, k\omega \quad (7)$$

The above equations are converted into MATLAB codes. Dimples of volume, $6x10^{-4}m^3$is made along the entire wing span of 3m long for both the square and triangle dimples.

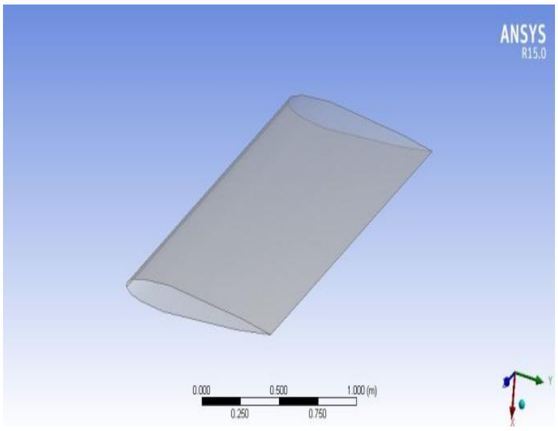
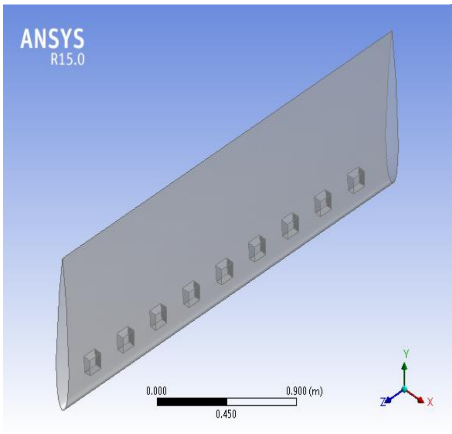


Figure 1: plain wing section of NACA-0012

In figure 1 shows the base or plain airfoil that is NACA 0012 which is imported model from CATIA V5R20.

IOP Conf. Series: Materials Science and Engineering **197** (2017) 012080 doi:10.1088/1757-899X/197/1/012080



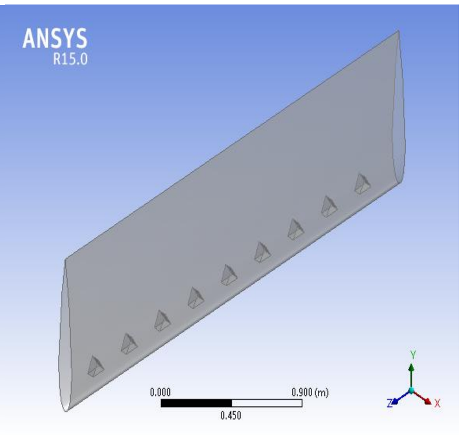


Fig 2: Square cavity pattern over the wing section Fig 3: Triangle cavity pattern over the wing section

In fig2, the dimples are arranged in a pattern on the surface of aerofoil at a distance of 0.1m from leading edge. The domain length is considered as 20 times the chord length of the aerofoil. For a square dimple the area is $0.01m^2$. NACA-0012 airfoil with chord 1m and span 3m is considered for the analysis. In fig3, for the same location of a triangular dimple is considered at the same location as the square one. The base dimensions of the triangle dimples are 0.03m long and 0.03m height, with one of its sharp edges facing the trailing edge. The depth of the dimples is 0.06m.

3. PRESENTATION OF RESULTS:

The simulation results for NACA0012 show the time dependent lift curve of various angles of attack. The simulation result agrees with the previous numerical work. The angle of attack is varied from 0^0 to 20^0 and it observed that the slope of the curve is closure to 2π. The inward cavities which as placed are wing span reduce the drag and the cumulative average of drag reduction is considerably low. For overall lift co-efficient the triangular cavities found to be 1.45, which is closure to 1.47 found using previous numerical results. The both cases the stalling angle is increases to 15^0 and 18^0. This is established in correlation with low Reynolds number using standard SST model.The factors which affect the boundary layer are fluid viscosity at adverse pressure gradient. As the angle of attack increases the flow pattern remains until the stalling angle. Hence the overall characteristics agree with the simulated MATLAB analysis.

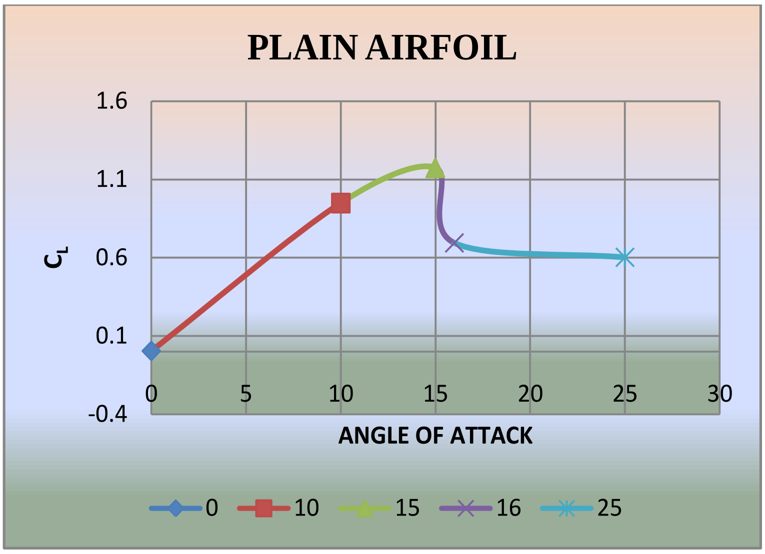


Figure 4: Co-efficient of lift Vs Angle of attack for plain NACA0012 airfoil

In fig 4, the above simulated results show positive increase the co-efficient of lift and the information recording the angle of attack which is at 15^0.

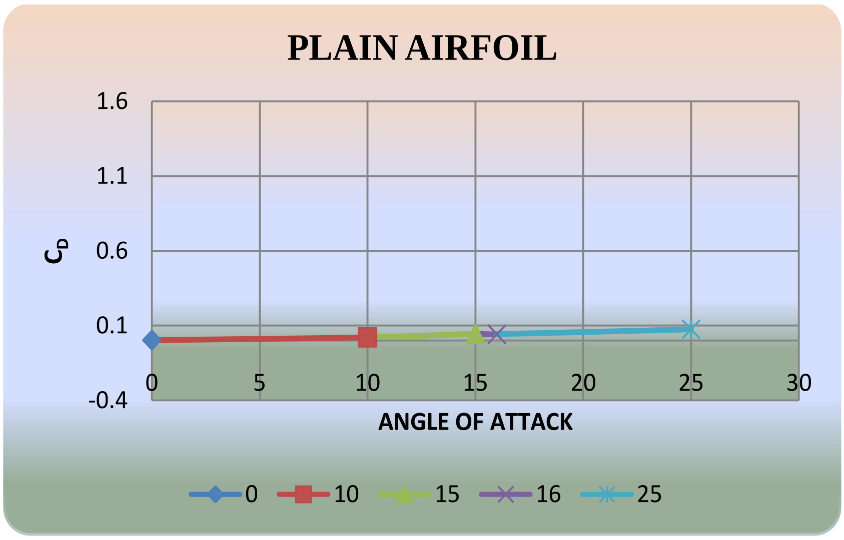


Figure 5:Co-efficient of Drag Vs angle of attack for plain NACA0012 airfoil

In fig 5, the subsequent drag profile for the plain airfoil indicates that for varying angle of attack the drag polar increases with increase in lift.

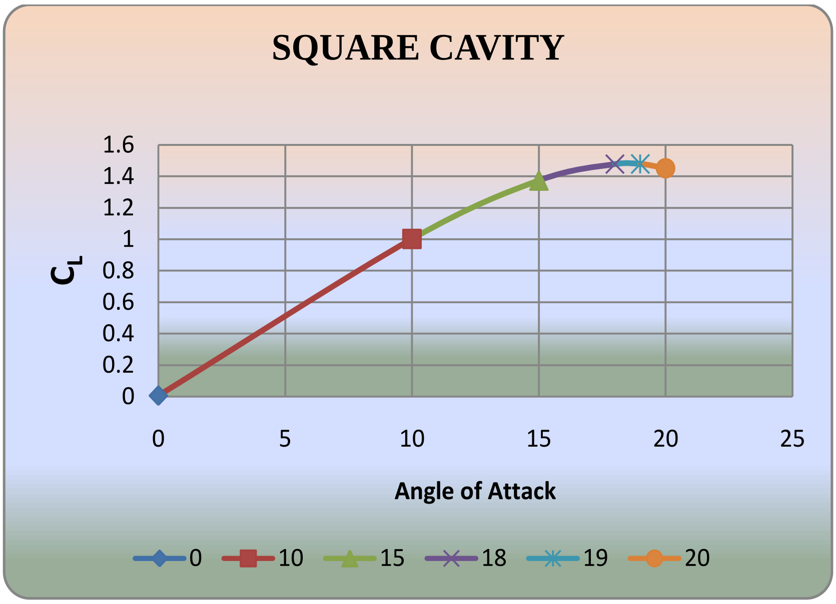


Figure 6:Co-efficient of Drag Vs angle of attack for square dimple

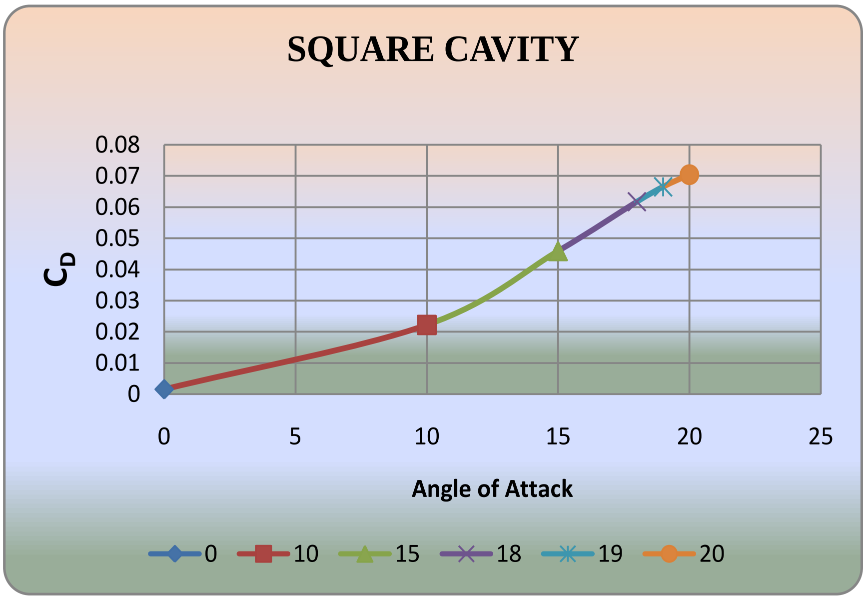


Figure 7: Co-efficient of lift Vs angle of attack for square dimple

In fig 6 and fig 7, the above graph for square dimple represents increase in stalling angle from 15^0 to 18^0. The drag polar explains the drag profile which is correspondingly less comparative plane airfoil.the drag polar for such a cavitiy is parbolicaly less than the angle of attack profile . this shows that the total drag acting on the airofoil is maintained at optimum level in order to provide the required aerodynamic characteristics. the angle of attack profile also increases from 15 degree to 18 degrees.

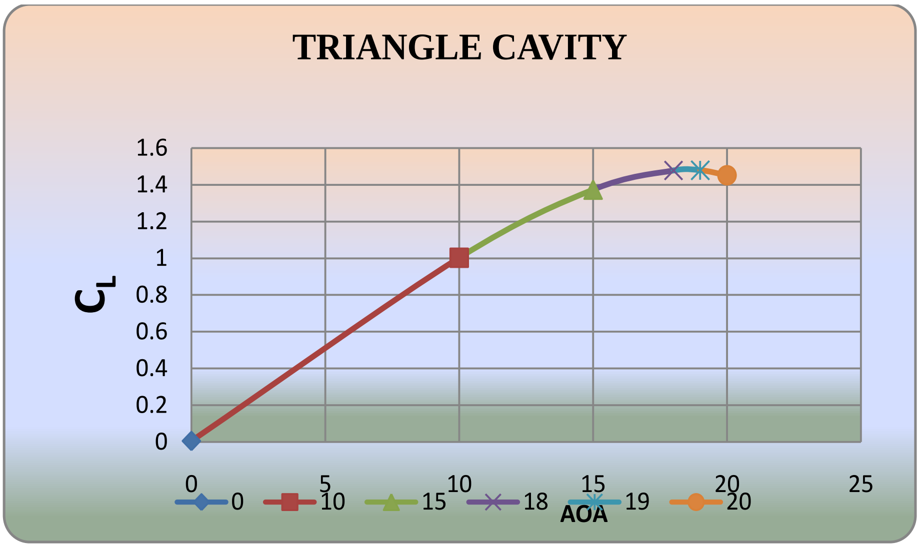


Figure 8:Co-efficient of lift Vs angle of attack for triangular dimple

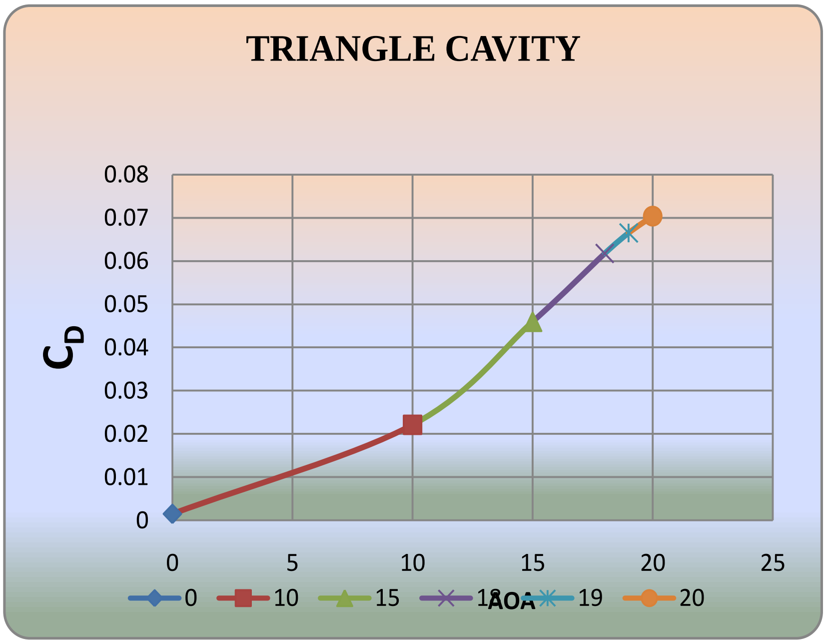


Figure 9:Co-efficient of Drag Vs angle of attackfor triangular dimple

In fig 8 and fig 9, the above graph for triangle dimple represents increase in stalling angle from 15^0 to 18^0. The drag polar explains the drag profile which is correspondingly less comparative plane airfoil.the drag polar for such a cavitiy is parbolicaly less than the angle of attack profile . this shows that the total drag acting on the airofoil is maintained at optimum level in order to provide the required aerodynamic characteristics. the angle of attack profile also increases from 15 degree to 18 degrees.

IOP Conf. Series: Materials Science and Engineering **197** (2017) 012080 doi:10.1088/1757-899X/197/1/012080

The above set of simulation using MATLAB agrees with the previous work and hence it is very clear that the equation taken for evaluating the effect of dimples is inperfect match with the analytical results.

4. CONCLUSION:

1. The non-linear charactersied of symmetrical airfoil NACA 0012 are characterised by the inward cavities which placed root to tip.
2. At very low reynolds number the laminar flow separation occurred as the angle of attack increased periodically.
3. The overall stalling angle exhibits and increase of 18^0 from 15^0.
4. The numerical simulation also indicates the dimple airfoil configuration gives better aerodynamic efficiency compare to plain aerofoil.
5. Furthur more the periodical flow separation on the upper surface of airfoil reduces the stalling effect on the dimple case.
6. The intensity of the cavity increases the lift co-efficient and decreases the drag c0-efficient.

REFERENCES

1. Srivastave.D, “Flow controls over airfoils using different shaped dimples” BITS Bilani, k.k Birla Goa, India-403726, 2012 pp 92-97.
2. Livya.E, Anitha.G and Valli.P “ Aerodynamic analysis of dimple effect over aircraft wing” international journel of mechanical aerospace industrial and mechatronics engineering, Vol-9. No 2, 2015 pp 350-353.
3. Md.AmzadHossain and Md.Nizzam Uddin ,“Experimental study of aerodynamic characteristics of airfoils using different shaped dimples” ISSN (e):2319 The IJCE Vol-4.No1,2015 pp13-17.
4. “Aerodynamic effects of dimples on aircraft wing” Intl. Conf. On Advances in Mechanical, Aeronautical and Production Techniques - MAPT 2015, ISBN: 978-1-63248-072-9.
5. Makwana.P.B,Makadiya.J.J, “Numerical simulation of flow over airfoil and different techaniquies to reduce flow seperation alongwith basic CFD model: A review study” ISSN: 2278-0181, 2014 PP 399-404.
6. Gad-el-Hak, Flow Control Passive: Active and Reactive Flow Management, First ed., Cambridge University Press, 2000.
7. D.J. Butter, “Recent Progress on Development and Understanding of High Lift Systems,” in Improvement of Aerodynamic Performance Through Boundary Layer Control and High Lift Systems, Brussels, Belgium, pp. 1.1-1.26,1984.
8. J.D. Anderson, Fundamentals of Aerodynamics, McGraw Hill Series, Third Edition, pp. 41-47, 2001.
9. A.M.O. Smith, “High-Lift Aerodynamics,”Journal of Aircraft vol. 12, pp. 501-530, 1975.

IOP Conf. Series: Materials Science and Engineering **197** (2017) 012080 doi:10.1088/1757-899X/197/1/012080

10. R.H. Liebeck, Smyth, D.N., "Study of slat-airfoil combinations using computer graphics," Journal of Aircraft,vol.10, April, pp. 254-256, 1973.
11. M. Sajben, et al., "A New, Passive Boundary Layer Control Device," AIAA-1976-700, presented at the American Institute of Aeronautics and Astronautics and Society of Automotive Engineers, Propulsion Conference, 12th, Palo Alto, Calif, 26-29 July, 1976.
12. F.G. Howard and W.L. Goodman, "Drag Reduction on a Bluff Body at Yaw Angles to 30Degrees," Journal of Spacecraft and Rockets, vol. 24, pp. 179-181, 1987.