International Journal of Advancements in Mechanical and Aeronautical Engineering– IJAMAE
Volume 2 : Issue 2 [ISSN : 2372-4153]
Publication Date: 19 October, 2015

Aerodynamic effects of dimples on aircraft wings

Bhadri Rajasai, Ravi Tej, Sindhu Srinath

***Abstract*— This paper is concerned with analysis of the turbulent flow over dimpled aerofoil profiles. Dimples of varying aspect ratio are used to study the effects on the skin-friction drag and lift. An external flow study was performed using ANSYS FLUENT. Simulations for external flow configuration with and without dimples were carried out and analyzed in detail. The resulting pressure drop and drag were observed. The objective was to clarify whether or not dimples cause reduction of the skin-friction drag and if it would provide better lift.**

***Keywords*—Aerofoil, Aspect ratio, Drag, Dimple, Pressure-drop**

I. INTRODUCTION

Dimples on golf balls have been inspiring engineers in the field of vehicle aerodynamics considering its effect in reducing drag on spinning bodies. A golf ball with a dimpled surface can travel higher and further than a smooth surfaced golf ball when subjected to identical force. The dimples on golf balls induce turbulence at lower Reynolds number, providing extra momentum or energy to the boundary layer and causing delay in flow separation. This phenomenon causes smaller wake areas or swirling flow regions behind the ball, thus reducing the total drag.

Till now these have been ignored because dimples help in reduction of pressure drag. In case of aerodynamic bodies pressure drag is very little compared to bluff bodies. An airfoil is an aerodynamic 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. Past studies observed that dimples on an airfoil created extra turbulence to delay the boundary layer separation. Adding a dimple on a streamlined body might help delay the flow separation and reduce the size of the wake but it might also increase the friction drag as a trade-off. Hence, it is necessary to optimize the position and dimensions of the dimple relative to the size of the whole body.

In order to verify the effect of dimples, the following computational study has been made on 3-D inward dimpled wing sections extruded from NACA aerofoils. In order to have a comparative study, slices of the aerofoils with and without dimples of different aspect ratios were simulated.

II. MODELS

In order to carry out the study on the proposed idea, standard Aerofoil profiles have been selected on which the whole study was conducted. Simulations were carried out on the aerofoil – NACA 2412. The study assumes the use of incompressible and isothermal flow.

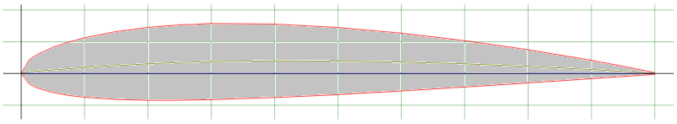

Fig. 1. Two dimensional NACA 2412 profile

The 3D model for the various cases used in the simulation were developed on Solidworks 2014, while the simulations were carried out in ANSYS 15.0 (FLUENT).

The 3D model was debossed with dimples of various aspect ratios and positions along the chord of the section.

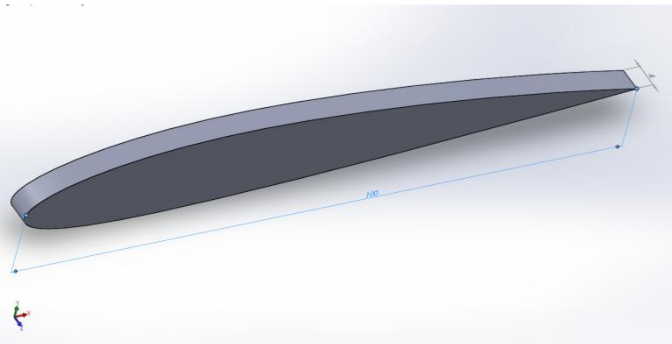

Fig. 2. Plain extruded NACA 2412 section

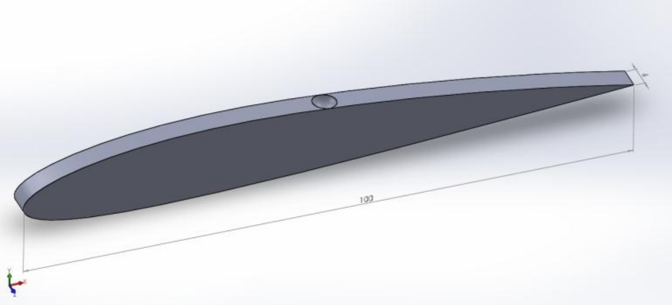

Fig. 3. Extruded NACA 2412 section with dimple at mid-chord with an aspect ratio of 0.7

Bhadri Rajasai
R.V.College of Engineering
India

Ravi Tej. G
R.V.College of engineering
India

Sindhu Srinath
R.V.College of Engineering
India

III. SOLVER MODEL

The SST (shear stress transport) k-ω turbulence model is used for the simulation using ANSYS FLUENT to predict the flow regime. The SST turbulence model is a widely used and robust two-equation eddy-viscosity turbulence model used in Computational Fluid Dynamics. The model combines the k-ω turbulence model and k-ε turbulence model such that the k-ω is used in the inner region of the boundary layer and switches to the k-ε in the free shear flow. The first transported variable is turbulent kinetic energy, k. The second transported variable in this case is the specific dissipation, ω. It is the variable that determines the scale of the turbulence, whereas the first variable, k, determines the energy in the turbulence

Following equations govern the flow profile around the models simulated:

Continuity equation:

$$\nabla . u = 0 \qquad (1)$$

Navier Stokes:

$$\rho u. \nabla u = \nabla .[-pI + (\eta + \eta_T)(\nabla u + (\nabla u)^T)] + F \qquad (2)$$

Transport equations for k-ω model:

$$\rho u. \nabla k = \nabla .[(\eta + \eta_T \sigma_k) \nabla k] + \eta_T P(u) – \beta_k \rho k \omega \qquad (3)$$

$$\rho u. \nabla \omega = \nabla .[(\eta + \eta_T \sigma_\omega) \nabla \omega] + a \omega \eta_T P(u)/k - \beta \rho \omega^2 (4)$$

Where: $P(u) = u: (\nabla u + (\nabla u)^T)$, and $\eta_T = \rho k/\omega$

$\beta_k = 0.09, \beta = 0.075, a = 0.55$

IV. MESHING AND SETUP

Triangular Meshing is used for the analysis. Each model is simulated at medium coarseness, with sizing properties such that the mesh is highly refined at the region around the Aerofoil section.

All simulations are carried out at angles of attack and aspect ratios, taking inlet velocity as $u_x = 20m/s$, $u_y = u_z = 0m/s$. Density of the air is taken as 1.2 kg/m3 and Dynamic viscosity as 1.2e-5 Pa-s. Domain cells are all tetrahedral shaped with all side walls of the airfoil given symmetrical boundary condition except for the inlet and outlet walls. Chord length of the airfoil is 100 mm and the span is kept constant at 4 mm in all simulations.

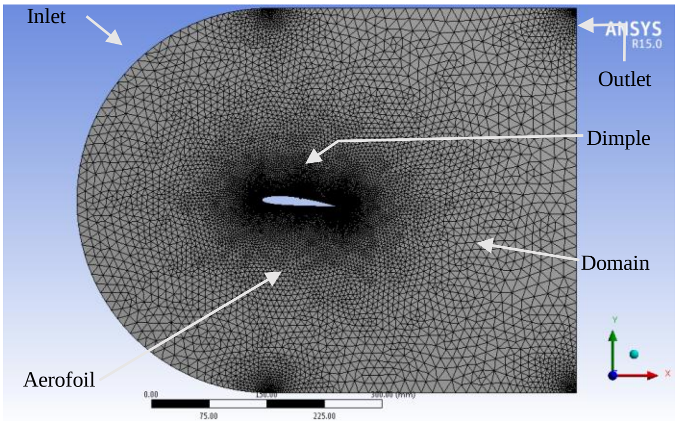

Fig. 4. Complete model with shown domain and boundaries to simulate airfoil in ANSYS FLUENT

All airfoil models are tested at 0, 5, 10, 15 and 20 degree of angles of attack. Vertical and horizontal forces i.e. Lift and Drag are calculated using ANSYS FLUENT post processing. Both lift and drag represented through dimensionless numbers – Coefficient of Lift (Cl) and Coefficient of Drag (Cd). The performance of each model is measured based on the aerodynamic efficiency i.e. Lift to drag ratio (L/D). Aerodynamic efficiency is one of the key parameters that determines the weight and cost of an aircraft. Roughly speaking, an aircraft's range is directly proportional to its aerodynamic efficiency without any increase in fuel usage.

V. RESULTS AND DISCUSSIONS

The study starts with CFD analysis of 3-D NACA2412 airfoils at different angles of attack, with inward dimples. Coefficient of drag is compared of these configurations along with one of plain airfoil.

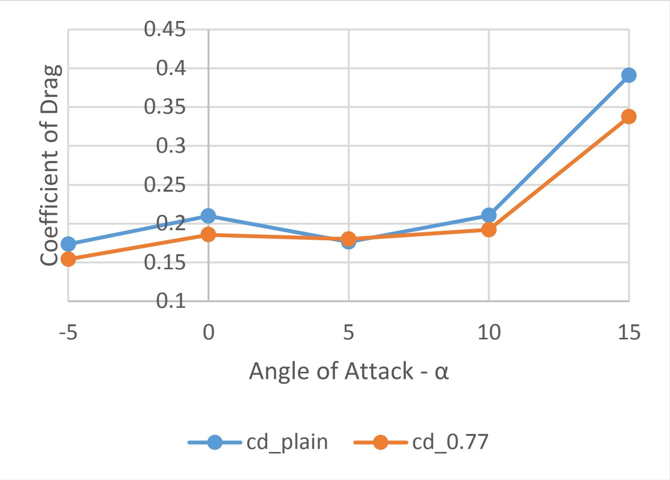

Fig. 5. Plot of Coefficient of drag versus angle of attack of plain and dimpled (aspect ratio 0.77) aerofoils

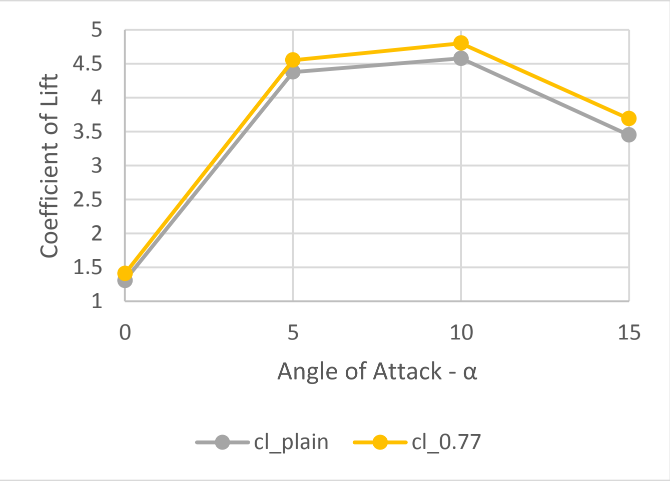

Fig. 6. Plot of Coefficient of lift versus angle of attack of plain and dimpled (aspect ratio 0.77) aerofoils

The study as seen in Fig.5 shows that as compared to plain aerofoils inward dimpled aerofoils produce lesser drag at a given velocity. Also, it is seen that the lift produced increases marginally, experimentally obtained significant increase in lift may be attributed to increase in propulsive velocity.

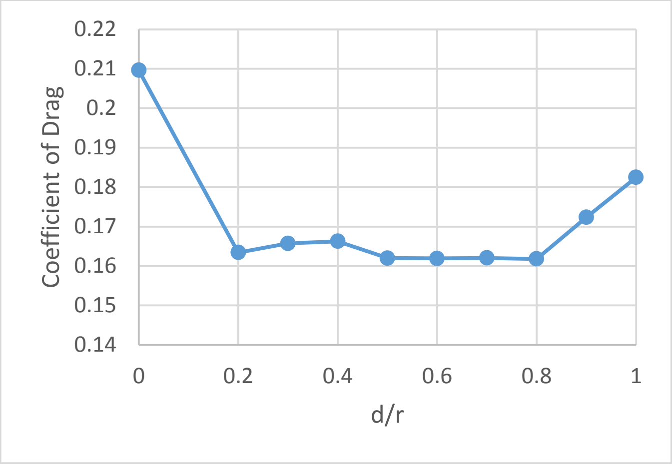

Fig. 7. Plot of coefficient of drag versus dimple aspect ratio

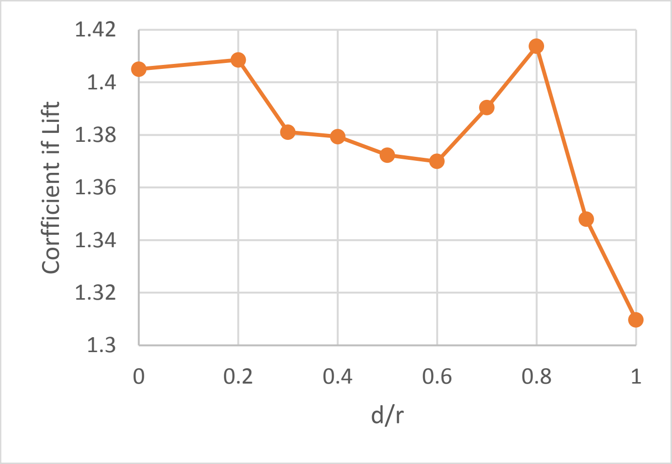

Fig. 8. Plot of coefficient of lift versus dimple aspect ratio

On varying the aspect ratio of the dimples, i.e. the ratio of depth to dimple surface radius, from figures 7 and 8, it can be seen that an aspect ratio of 0.8 is ideal for least drag and most lift for the given angle of attack.

Figures 9 and 10 show the variance of lift and drag coefficients for different positions of the dimple along the length of the chord for the Aerofoil at zero angle of attack. For the configuration, it is seen that the dimple is most effective while placed at 70% from the leading edge of the wing.

However, further studies revealed that the performance of the dimple is high when the dimple is optimally placed for each angle of attack, with the distance from the leading edge decreasing with increasing angle of attack. Results of the same have not been included here. Upon studying the pressure and velocity contours around the Aerofoil, the position of the optimum placement of the dimple can be found by realizing the region of velocity drop and pressure rise, owing to the flow separation that occurs at the given angle of attack

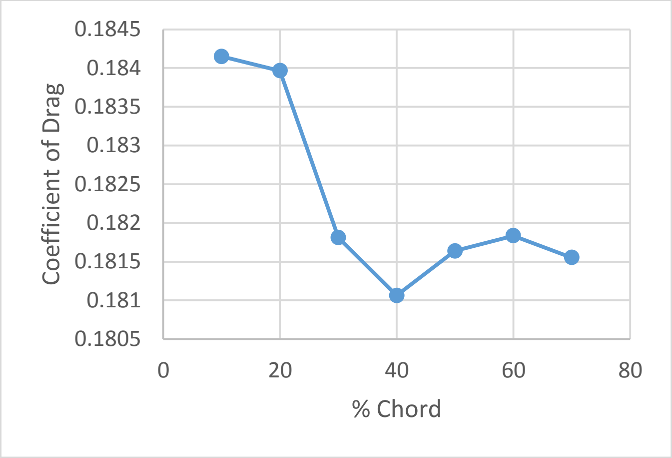

Fig. 9. Plot of coefficient of drag versus dimple position along chord length of aerofoil

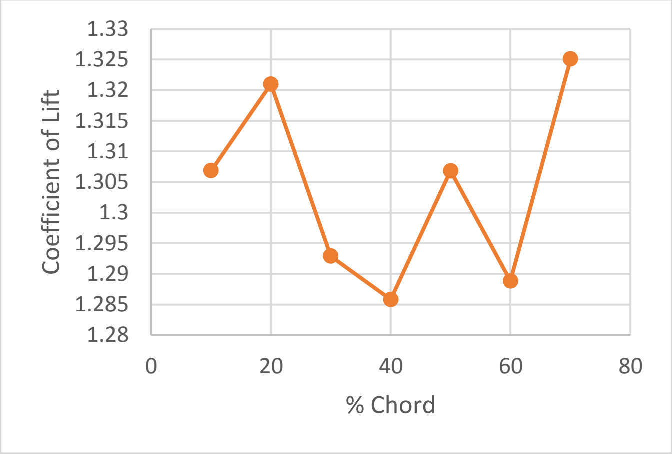

Fig. 10. Plot of coefficient of drag versus dimple position along chord length of aerofoil

VI. CONCLUSIONS

It has been proven that when the air flows along the surface of the aerofoil with dimple, there is an acceleration of the flow at the dimple surface and the boundary layer changes from laminar to turbulent. This transition results in delayed flow separation which reduces the drag. The presence of a dimple therefore increases the stall angle of the aircraft. This, if incorporated would be extremely beneficial in making an aircraft more maneuverable and increase the aircraft's fuel economy. The position and dimensions of the dimple affect the drag and lift characteristics. The total aerodynamic efficiency increases due to the reduced drag. However, experimental studies have to be performed. It is also necessary to determine the feasibility of generation of dimples on aircraft wings. The concept of presence of dimples on aircraft wings to reduce drag cannot be applied to all aerofoil profiles. Further investigation has to be carried to validate these results.

ACKNOWLEDGMENT

Acknowledge with thanks C.S.Prasad, Professor, Department of Mechanical Engineering, RV College of Engineering, Bangalore for guiding in the progress of our project.

REFERENCES

[1] Deepanshu Srivastav, “Flow Control over Airfoils using Different Shaped Dimples,” IPCSIT IACSIT Press, Singapore, vol. 33, pp 92-97, 2012.

[2] K.Manoj Kumar, P.Manivannan, “Experimental Study - Flow Characteristics of Dimpled Wing”, International Journal of Engineering Research & Technology (IJERT),ISSN: 2278-0181,pp 2018-2022, Vol. 3 Issue 4, April – 2014

[3] University of Illinois at Urbana-Champagne (UIUC) Airfoil database, http://www.ae.illinois.edu/mselig/ads/coord_database.html

[4] Jesse Sidhu, ”Analysis for Induced Delay of Boundary Layer Separation”,Phd Thesis, Henry Samueli School of Engineering

[5] Dr.P.PrabhakaraRao , Sri Sampath.V, “CFD Analysis on Airfoil at High Angles of Attack”, International Journal of Engineering Research ISSN:2319-6890)(online),2347-5013(print), Volume No.3, Issue No.7, pp 430-434 ,01 July 2014

[6] E. Livya, G. Anitha, P. Valli,” Aerodynamic Analysis of Dimple Effect on Aircraft Wing” World Academy of Science, Engineering and Technology International Journal of Mechanical, Aerospace, Industrial and Mechatronics Engineering, Vol:9, No:2,pp 350-353, 2015

[7] John D. Anderson; “Fundamental of Aerodynamics”, third edition.

[8] L J CLANCY, “Aerodynamics”, Indianedition 2006