ISSN 2229-5518

# A Review on Aerodynamic Behavior of Airfoil when Surface Modified

Amit Kumar Saraf, Dr. Mahendra Pratap Singh, Dr. Tejsingh Chouhan

**Abstract**— There are limitations of engines of aircraft to decrease drag and increase efficiency. Efficiencies requirement is increasing day to day. Airfoil plays an important role in efficiency of aircraft. Modification in the shape of airfoil is also important in aircraft. General airfoil behavior has been analyzed by many researchers. In this paper airfoil characteristics given by many researchers are studied while surfaces of airfoils were modified. Modification on surface of airfoil gives significant changes in result. Without consuming much amount of fuel in aircraft drag can be decreased with simple modification on airfoil surface..

**Index Terms**— Airfoil, CFD, Aerodynamic behavior, Coefficient of Lift & Drag,angle of attack,flow separation,Dimple .

## 1 Introduction

IN early days with the help of powerful engines effort was made to reduce drag and increase lift and velocity of airplane. Importance of aerodynamics came in role in the twelfth century. Modification in airfoil shape plays an important role in aerodynamics. Various types of airfoil series were developed. Every series has its own aerodynamics characteristics. Such approach is still used. The flow separation on airfoil increases pressure drag. During flight, increases or decreases in lift will cause incremental starting or stopping vortices, always with the effect of maintaining a smooth parallel flow at the trailing edge. At a low angle of attack, the rear surfaces have an adverse pressure gradient but not enough to cause significant boundary-layer separation. As the angle of attack is increased, the upper-surface adverse gradient becomes stronger, and generally a separation bubble begins to creep forward on the upper surface. At a certain angle α = $15^0$ to 20°, the flow is separated completely from the upper surface.

## 2 Literature Review

### 2.1 Review Stage

NACA4315 model was used to analyze the aerodynamic properties. Bumps were used at upper surface on trailing side. The bumps were created 80% from leading edge. Regular and bumped airfoils were compared. It shows that stall angle increased due to controlling on flow separation [13].

Airfoil four digit nomenclatures tell camber and thickness of airfoil. From this, we can find airfoil series. This helps to change the camber of airfoil and location of maximum thickness [2].


- *Amit Kumar Saraf is Phd Scholar in Jagannath University Jaipur, Rajasthan India. (Phone: +919928396705; e-mail: amitsaraf11@rediffmail.com)*
- *Dr.Mahendra Pratap Singh, is with Jagannath University Jaipur, Rajasthan India. He is now with the Department of Mechanical Engineering (e-mail: mpsingh_76@yahoo.co.in)*
- *Dr. Tejsingh Chouhan is with the Mechanical Engineering Department, Rajdhani Institute of Technology & Management,Jaipur Rajasthan, India (e-mail: tejsingh7@gmail.com)*


At various Reynolds no aerodynamic property of smooth and rough airfoil were compared. It was observed that aerodynamic losses were increased at high Reynolds no and losses were decreased at low Reynolds no. in rough airfoil. Separation bubble becomes weak due to roughness of surface at low Reynolds no [3].

NACA00012 airfoil was tested under different turbulent models i.e. [Spalart-Allmaras, Realizable k-ε and k-ω shear stress transport (SST)].These turbulent models were compared and validated with experimental data. It was found that k-ω shear stress transport (SST) gives the best result for given airfoil. 80000 cells were taken for simulation. Air velocity was taken constant. Before solving it the main important work was to find out the transition point. Transition point should be modeled to get more accurate result. Here commercial CFD software was used. According to this if much amount of nodes are used, result will be more accurate, but huge amount of nodes take much time in computation. Here C type grid topology was used and 80000 quadrilateral cells were taken. Transition point was determined by hit and trial method. If the value of simulated $C_D$ is greater than experimental value, it means that transition point chosen is wrong, turbulent region is larger. SO author have to shift it in right side and accordingly he can determine transition point. In result there was a disagreement between the data at near stall. The predicted drag coefficient was higher than experimental data. This is because the actual airfoil has laminar over the half of leading side. Turbulent model consider turbulent boundary layer throughout its length [12].

Two row and eight rows dimples on flat surface were investigated and observed that dimples were effective to convert laminar into turbulent at low Reynolds no. Multiple rows increases strength of mixing flows. The analysis was also done on sphere where it was found that drag were decreased in dimpled sphere compared to smooth sphere [4].

There were various experiments going on with various methods to reduce drag and to improve efficiency of wind turbine, airfoil of flight vehicle etc. A detailed study was done on airfoil to reduce drag on the trailing side. Since one of the main reason of drag or pressure drag is the formation of turbulent on the trailing side. Roughness was created in turbulent region of the smooth airfoil. This result in decrease in drag and gives better aerodynamic efficiency. Experimental work was done using a wind tunnel and it was found that lower roughness gives the better result than upper roughness compared to smooth airfoil. Triangular roughness was gives better result than others and roughness was placed on 60% from the leading edge on the lower surface. The drag was reduced at negative angle of attack $-20^0,-15^0,-10^0$ when the roughness at the lower side of the airfoil, but at the positive angle of attack $20^0,15^0,10^0$ the drag was reduced when the roughness at the upper side of the airfoil. CFD analysis was also done and found that lift and drag altered. [1].

Surface modification was done by using inward and outward dimples using 2-D CFD with k-ω turbulent model. CFD analysis was done on various types of shapes of dimples and tested in 2-D and on 3-D CFD analysis. The result shows that dimples affect the drag much more, since it delays boundary separation and reduce wake formation by increasing turbulent. The study was done on NACA-0018 airfoil and flow condition taken subsonic. CatiaV5 R18 was used for CAD model and simulation was done on Comsol 3.4 and Comsol 4.2a. The result shows that there is increase in lift and decrement in drag. Vortex generators also used to create turbulence by creating vortices. Turbulence result shows that it delay in boundary layer separation, hence it leads to increase in stall and decrease in pressure drag and therefore increase in lift. Dimples on airfoil work as a vortex generator. At stall condition flow separation dominates and decrease in lift. Here 2-D Model were computed with inward and outward dimples and it was found that outward dimple perform better at positive angle of attack and is suitable for further study. On the basis of this a new matrix of outward dimple can be made. In complete study it was to determine the sustainability of dimples to act as vortex generator [26].

Computational fluid dynamics was used to study of airfoil. Other researcher's drawbacks were the motivation to do study of passive device to improve the lift and decrease in drag of airfoil. Most efficient position of the dimple and cylinder was found out. The dimple model shows good result than cylinder. Modeling was done in CATIA V5 R20, preprocessing was done on ANSYS ICEM CFD 14.0 and post processing done on ANSYS FLUENT 14.0. If the location of dimple or cylinder was at the optimum position, then it gives better outcomes. NACA 4412 studied and model prepared in CATIA V5 R20. Both models were analyzed but it was found that dimple model gives better result. Fine meshing give more accurate result, but computer memory and time are the limitations. Result showed that dimple airfoil gives the better result compared to cylinder [10].

Wing was designed with dimples on the upper surface of the wing. Analysis was done using ANSYS. It was found that skin frictions increased but drag reduced in such an amount that effect of skin friction can be neglected [20].

Turbulence was created by using dimples due to generation of vortices. This result in decreased pressure drags and improves maneuverability. Different shapes of dimples were used. A comparative study was done between inward and outward dimple. A combination of two (semi spherical followed by square) dimple with constant height and depth ratio studied. Inward dimpled shape gives better lift coefficient in both single and compound dimples [11].

Maximum thickness location of the airfoil has been optimized by using Genetic Algorithm on NACA 0012 airfoil. For changing the angle of attack microcontroller has been designed. Airfoil characteristics have been studied i.e pressure, drag, lift and moment under subsonic condition. Mat Lab was used to validate the results. Eight different best designs have been considered out of various designs according to Genetic Algorithm. Composite material was used for wind tunnel test. Design no.6 gives the best results out of the eight designs. Result shows that 1% hike in lift and 26% fall in drag observed at Mach no 0.7. There was 28% hike in L/D at Mach No.0.4. Design no six fails in stability due to much variation in momentum as angle of attack increased [19].

NACA 2412 dimpled and smooth 3d printed airfoil was tested at various angles of attack. Velocity varies 5, 10 m/s with different angle of attack that varies from $6^0$ to $18^0$. The complete evaluation was at low angle of attack. Printed 3D model of airfoil was cured with acetone vapour bath for smoothness of surface.3-D printer was used to save time & money. After experiment it was observed that dimpled airfoil give increase in stall then smoothed airfoil. Stall angle of attack is increased by $2.1^0$ [8].

A low pressure turbine blade was investigated to examine different dimple pattern. Three different pattern were tested with experimental and computational basis. Dimple pattern were located at 65% and 76% on the axial cord of the airfoil. One pattern of single row with 2.22 and 4.44 cm spacing was investigated. Second pattern of double row staggered with 4.44 cm spacing was investigated. All patterns were also investigated on CFD tool. In this spacing in single row pattern was 2 cm and in double staggered row was 4 cm investigated.CFD software FLUENT v6.1, Gridgenv15 and FIELDVIEW v9.1 was used. All experiments were performed at 25K, 45K and 100K Reynolds no. It has been observed that loss efficiency was increased by 35% at 25K. Induced turbulent did not play a significant role in flow separation. Experimental & computation result were approximately identical [6].

On the bluff body roughness stimulates turbulence hence leads to delay in separation of boundary layer. Since old golf ball runs faster than new golf ball because of roughness on old ball. Inspiring from these phenomena some researcher made golf ball with dimple. Chear take a car model with different dimples on it. Flow was considered turbulent and simulation done with k-ε turbulent model. Aim was to reduce drag on car. Dimple ratio (depth to diameter) was designed on the car body. Five different dimple ratio used and simulation was done on ANSYS simulation tool. With the addition of dimples turbulent kinetic energy generates within the dimple and manages the flow separation. $C_D$ was reduced by 1.95% when the DR is 0.4 used [9].

Vortex generator was provided on NACA 0015 airfoil to improve lift and decrease drag. Vortex generator placed on 10% from the leading edge. Experimental model developed for analysis. VG height, space between them was also considered. The vortex generator height plays an important role in analysis. The optimized geometry improves result by 14% in lift and 16% in drag [15].

Advanced computational fluid dynamics models are focused

ISSN 2229-5518
which are capable of capturing the micro scale flow physics associated with passive vortex generator. VG are used on wind turbine blade. So many research has been performed on vortex generator the device ranges from 10%-50% of the boundary layer thickness [28].

Whenever airplane stall and suddenly lose lift required, in that case dimples on airfoil used to increase lift. Material used for this purpose was balsa wood 4 feet long. Comparison was done on dimpled and without dimpled airfoil 15 times. Second airfoil was again tested to validate data. Result showed that dimpled airfoil performed well [21].

A high lift airfoil at low Reynolds no has been designed and the experiment has been performed and validated at NASA Langley Research Center's Low Turbulence Pressure Tunnel. It was observed that airfoil become more cambered when the pitching moment increases. Wind tunnel model S1223 was tested at Reynolds no 2 X 105. Experiment was done using VG and Gurney Flap [24].

Aerodynamics characteristics were measured and flow separation is determined.NACA0015 model airfoil is used and C-Programming is used. Cp and x/c graph was analyzed and found that at $12^0$ AOA there is no flat area and at an AOA $20^0$ it seems that value of Cp was approx smooth, showing boundary later separation. It is clear from this paper that flow separation cannot be controlled or delayed with conventional ways [16].

Dimpled surface augmentation and grit is used on NACA0011 and NACA16611.Surface augmented dimple taken at 50% chord, 23% chord and 8% chord from leading edge to trailing edge. A grit tape was also used. Different test was done and finally it was observed in every test that from 8% chord to the trailing edge had reduced separation [27].

Vortex generators used to delay flow separation. The main factors considered was chord wise position, size, aspect ratio, inclination of VG and inter and intra spacing. In this research VG are placed at x/c=0.15, x/c = 0.20 and x/c = 0.3 from leading edge. Stalling angle is increased when VG moved towards leading edge [25].

Air jet vortex generator also used to suppress the flow separation. A passive vortex generator jet flow control system was used from leading edge attachment line fed the upper surface air jet. In this we do not require any active energy input. Air jet vortex generator (AJVG) located at 12% of chord. NACA 23012C model is compared with active and passive AJVG. The result shows that span wise array of passive air jet vortex generator can effectively delay separation [22].

In this paper active flow control system was used. The vortex generator jet added to the main wing to suppress the separation. Slatless high lift airfoil was used with a flap. VG location was on the pressure side close to leading edge. Two test were performed with constant mach no. Reynolds no varies during study. It was found that lift is inversely proportional to Reynolds no. Dynamic blowing results higher lift [7].

NACA 0012 airfoil used to study transient growth in laminar separation bubble. Laminar separation bubble formed due to separation of flow, the flow convert from transition to turbulent and reattached to surface. This LSB can affect the performance of airfoil. It was observed that flow is periodic with temporal frequency of 1.27. In two dimensional flows it was observed that monotonic energy increases with interval of time and it can reach up to around six order of larger [18].

Design of Natural Laminar Flow on airfoil can be used to increase in the percentage of laminar flow and to decrease drag by using Boundary Layer Mixing Devices. Vortex generators were used as a boundary layer mixing device in current uses throughout aircraft industries. Vortex generators are passive flow control device and do not require any additional power to operate. In order to reduce laminar bubble several airfoil section were designed. Code was modified to design airfoil at the upper and lower surface with the help of vortex generator [17].

A car model was used and dimple ratio (depth to diameter) also designed on the car body. In this paper 5 different dimple ratio were used. Simulation was done on ANSYS simulation tool [9].

Sphere is a bluff body and dimple pattern on the spherical body was studied by understanding the flow of hydrodynamics. Experimental and numerical investigation was performed. By using various turbulent models, K-w turbulent model gives good results and k-ε is not useful for curved body. Geometry of dimple also affects significantly aerodynamic characteristics. Formation of small separation bubble leads to delay of the flow separation [5].

With the help of nose flap, slas, variable camber can increase lift coefficient but these are mechanically complex. The addition of roughness or vortex generator is the simplest technique to increase lift coefficient. Vortex generator can be taken at 10% of chord from leading edge, its height and length is about 1% of chord and 2-3% of chord respectively [14].

Riblets are also used to reduce skin friction, ultimately reduced drag up to 6-10%. Use of riblet is as a passive flow control technique. Though roughness reduces drag but increases skin friction so riblet is the best method to control skin friction [23]..

## 4 CONCLUSION

Though surface modification has been done in previous studies but in some studies there is a gap and there may be further scope of modification to increase the efficiency of airfoils. This study shall be useful in aircraft industry and wherever the airfoils are used. This study has huge scope as per previous researches. Combination of two types of modification is still not done. The main objective of this review is to analyze the behavior of airfoil when its surface is modified. Computational fluid dynamics is carried out for analysis of flow aiming to enhance aerodynamic characteristics by reduction in drag. By modification on surface, airfoil characteristics of aerodynamic will change. Thus it is predicted that lift is increased and stall angle of the airfoil is increased and drag will be reduced with modification.

## REFERENCES


[1] A. Dhiliban, P., & Narasimhan, P. (2013). AERODYNAMIC PERFORMANCE OF REAR ROUGHNESS. The Eighth Asia-Pacific Conference on Wind Engineering (pp. 193-200). CHENNAI, INDIA: Research Publishing, Singapore. ISBN: 978-981-07-8011-1.

[2] Anderson, J. D. (2012). Fundamental of Aerodynamics. In J. D. Jr, Fundamental of Aerodynamics (Fifth Reprint ed., pp. 300-301). New York: Tata McGraw -Hill, New Delhi India.

[3] Bai, T., Liu, J., Zhang, W., & Zou, Z. (2014). Effect of surface roughness on the aerodynamic performance of turbine blade cascade. Propulsion and Power Research , 3 (2), 82-89.

[4] Beratlis, N., Balaras, E., & Squires, K. (2014). Effects of dimples on laminar boundary layers. Journal of Turbulence , 15 (9), 611-627.

[5] Bogdanovic, J. B., Jovanovic, & Stamenkovic, Z. M. (2012). Experimental and Numerical Investigation of Flow Around a Sphere With Dimples For Various Flow Regimes. Thermal Science , 1-14.

[6] Casey, J. P. (2004). EFFECT OF DIMPLE PATTERN ON THE SUPRESSION OF BOUNDARY LAYER SEPARATION ON A LOW PRESSURE TURBINE BLADE. Wright-Patterson Air Force Base, Ohio: AIR FORCE INSTITUTE OF TECHNOLOGY,Ohio.

[7] Casper, M., Scholz, P., & Radesiel, R. (2011). Separation Control on High Lift Airfoil Airfoil Using Vortex Generator Jet at High Reynolds Numbers. 41 st AIAA Fluid Dynamic Conference and Exhibit (pp. 1-11). Honolulu, Hawaii: American Institute of Aeronoutics and Astronautics,Inc.

[8] Chavarin, A., Magtoto, A., & Lee, D. (2014). Experimental Comparison of Lift between a NACA. Design Laboratory Project, University of Calofornia Irvine , 1-16.

[9] Chear, C., & Dol, S. (2015). Vehicle Aerodynamocs: Drag Reduction by Surface Dimples. International Journal of Mechanical, Aerospace, Industrial and Mechatronics Engineering , 9 (1), 202-205.

[10] Dr.P.PrabhakaraRao, & Sampath, S. (2014). CFD Analysis on Airfoil at High Angles of Attack. International Journal of Engineering Research , 3 (7), 430-434.

[11] E.Livya, Anitha, G., & Vali, P. (2015). Aerodynamic Analysis of Dimple Effect on Aircraft Wing. International Journal of Mechanical, Aerospace,Indusrial and Mechatronics Engineering , 9 (2), 350-353.

[12] Eleni, D. C. (2012). Evaluation of the turbulence models for the simulation. Journal of Mechanical Engineering Research, ISSN 2141-2383 , Vol. 4(3), 100-111.

[13] Faruqui, S. H., AlBari, M. A., & Emran, M. (2014). Numerical analysis of role of bumpy surface to control the flow separation of an airfoil. Procedia Engineering ,10th International Conference on Mechanical Engineering, ICME 2013. 90, pp. 255 – 260. Bangladesh: Elsevier.

[14] G.W.Gyatt. (1986). Development and Testing of Vortex Generators for Small Horizontal Axis. U S Department of Energy. Lewis Research Center: National Aeronoutics and space Administration.

[15] H.Tebbiche, & Boutoudj, M. (2014). Optimezed Vortex Generatoros in the Flow Separation Control around a NACA 0015 profile. Proceeding of 9th International Conference on Structural Dynamics, EURODYN 2014 (pp. 3219-3226). Porto,Portugal: ISSN:23119020, ISBN:-9789727521654.

[16] Islam, M., Hossain, M. A., & Uddin, M. N. (2015). Experimental Evaluation of Aerodynamics Characteristics of a Baseline Airfoil. American Journal of Engineering Research , 4 (1), 91-96.

[17] Kerho, M., & Kramer, B. (2003). Enhanced Airfoil Design Incorporating Boundary Layer Mixing Devices. 41 st AIAA Aerospace Science Meeting & Exhibit (pp. 1-18). Reno NV: American Institute of Aeronautics and Astronautics Inc.

[18] Loh, S., Balckburn, M. H., & Sherwin, J. S. (2014). Transient Growth in an Airfoil Separation Bubble. 19th Australasian Fluid Mechanics Conference (pp. 8-11). Melbourne, Australia: Australian Fluid Mechanics Conference.

[19] MANIKANDAN, G., & RAO, M. A. (2011). Effect of Maximum Thickness Location of. INTERNATIONAL JOURNAL OF ADVANCED ENGINEERING SCIENCES AND TECHNOLOGIES , 3 (2), 122-133.

[20] Mohanasaravanan, P. (2015). Flow Analysis around The Dimple Wing on Aircraft. International Journal of Engineering Research-Online , 3 (2), 402-406.

[21] Patacsil, M. B. (2010). Dimples on Wing. California: California State Science Fair.

[22] S.A.Prince, Khodagolian, V., & Singh, C. (2009). Aerodynamic Stall Supperation on Airfoil Section Using Passive Air Jet Vortex Generator. (F.Coton, Ed.) AIAA Journal , 47 (9), 2232-2242.

[23] Saravi, S. S. (2013). A REVIEW OF DRAG REDUCTION BY RIBLETS AND MICRO-TEXTURES IN THE TURBULENT BOUNDARY LAYERS. European Scientific Journal , 9 (33), 62-81.

[24] Seling, M. S., & Guglielmo, J. J. (1997). High Lift Low Reynolds NumberAirfoil Design. Jornal of Aircraft , 34 (1), 72-79.

[25] Sorensen, N. N., Zahle, F., & Bak, C. (2014). Prediction of the Effect of Vortex Generators on Airfoil Performance. Journal of Physics: Conference Series 524. 524, pp. 1-11. Denmark: IOP Publishing.

[26] Srivastav, D. (2012). Flow Control over Airfoils using Different Shaped Dimples. International Conference on Fluid Dynamics and Thermodynamics Technologies , Vol-33, 92-97.

[27] Vento, J. L. (2012). Analysis of Surface Augmentation of Airfoil section via Flow visualization Techniques. San Luis Obispo: California Polytechnic State University, San Luis Obispo.

[28] Xue, S., Johnson, B., & Chao, D. (2010). Advanced Aerodynamic Modeling of Vortex Generators for Wind Turbine Application. European Wind Energy Conference (pp. 1-10). Warsaw Poland: European Wind Energy Conference