iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 5 **Issue:** IX **Month of publication:** September 2017

DOI: http://doi.org/10.22214/ijraset.2017.9055

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor:6.887
Volume 5 Issue IX, September 2017- Available at www.ijraset.com

CFD Analysis of Dimple Effect on Airfoil Naca 0015

Somanagoud Biradar[1], Anandkumar S Malipatil[2]
[1] M Tech Student, Assistant Professor[2], Dept of Thermal Power Engineering, VTU Regional Centre Kalaburagi, Karnataka, India.

Abstract: The main goal of an aircraft aerodynamics is to improve the aerodynamic behaviour of the aircraft. The airfoil model contains dimple on the surface have less drag than the model contains no dimple. Providing the dimples on the surface of the airfoil i.e. cross section of the aircraft wing will generate turbulence by creating vortices which reduces the boundary layer separation from laminar to turbulence resulting in decrease of pressure drag. The main objective is to reduce the take off distance of the aircraft by minimizing the drag force. The project includes CFD analysis of vortex effect on the 3D model of airfoil NACA 0015. Dimple shape may be semi-sphere of 15 mm in different configuration i.e. inward and outward and 1 m chord selected for the analysis under the 17 m/s of flow velocity at zero degree angle of attack. Dimple may be provided at different positions of the airfoil from the leading edge and analysis were performed coefficient of lift, coefficient of drag, velocity contours and pressure contours are calculated and compared with the plane model.
Keywords – Airfoil, Lift coefficient, drag coefficient Pressure contours, Velocity contours.

I. INTRODUCTION

An aircraft is a basically machine which is capable of fly in aerospace by gaining support from the air. The relation between aircraft and fluid (air) is termed as aerodynamics which deals with the effects of forces, moments and motion of the airplane through the space. The aero dynamical forces associated with the aircraft are lift and drag, they exists due to the movement of the airplane in aerospace. The lift created by the air flowing over the blade creates the upward force called as thrust against the weight of the plane which pulls down. The drag generated by the air resistance to the front of the airplane will overcome by the thrust created by the propeller. In level flight and at constant speed drag force is exactly equal to the thrust force and lift force exactly equal to the weight or gravity force. NACA Airfoil series is developed by national advisory committee for aeronautics. In this project the airfoil chosen for analysis is NACA 0015 which is a symmetric airfoil. The geometry of the airfoil is explained by series of digits following by the NACA. The first digit indicates the maximum camber as percent of chord, second digit indicates the position of the maximum camber measured from the leading edge in percentage of chord and third and fourth digits including denotes maximum thickness of the airfoil in percentage of chord length. The NACA0015 is a symmetrical airfoil with a 15% thickness to chord ratio. Symmetric airfoils are used in some applications including aircraft vertical stabilizers, fixed wings, rotary and some submarine fins. A 3D wane cross section is analyzed at low velocity for lift, drag and vector characteristics and also pressure and velocity contours. Presently various types of surface modifications are studied to increase the maneuverability of the aircraft. Dimples on surface or the vortex generators are most commonly used type of surface modifications. These dimple effect creates the turbulence effect by creating the vortices, which delay the boundary layer separation which result in lower in the pressure drag. For example golf ball with dimple effect travels more distance than the smooth surfaced ball when subjected to same force which is mainly because of the reduction in drag force due to the dimple effect. The dimples generates the turbulence at lower value of Reynolds number, which provides extra energy to the boundary layer and resulting late in flow separation from laminar to turbulence, thus reducing the drag force therefore meanwhile decrease in the takeoff distance of the aircraft.

In our project surface modifications considered are inward dimple on both top and bottom surfaces of NACA0015 airfoil. Further iterations are carried out for zero angles of attack with two numbers of dimples in different dimensions. A 3 dimensional computational fluid dynamics analysis is carried out using Spalart Almaras model, after this among various results the better of there is chosen.

II. METHODOLOGY

A. Geometry generation

Airfoil model can be generated using CAD software solid works using the coordinate points taken from the UIUC airfoil database coordinates. Chord length of the airfoil has taken as 1m and spam length of 100mm. Dimples are given on the surface of the airfoil at different length from the leading edge with a depth of 15 mm.

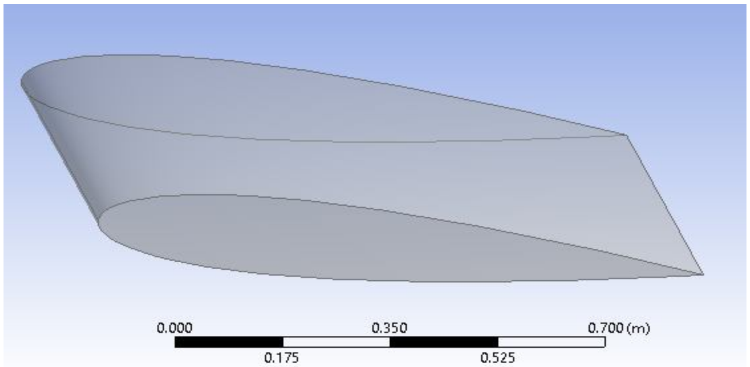
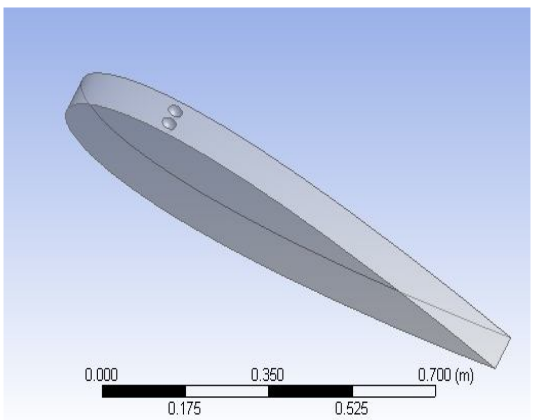


Fig 1: Airfoil NACA 0015

Airfoil length	1000mm
Max thickness	15 % of chord
Angle of attack	0 degree

Table: Model dimensions



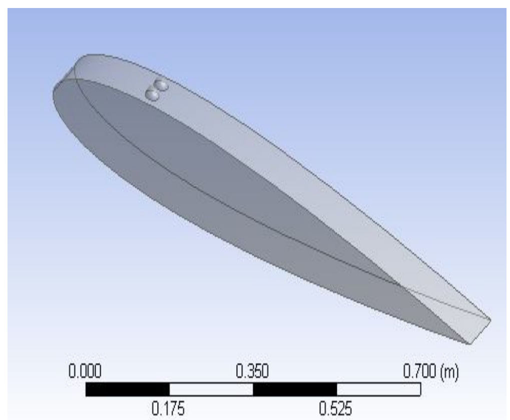


Fig 2: Airfoil model with inward dimple at 206mm from leading edge   Fig 3: Airfoil model with inward dimple at 206mm from leading edge

B. Meshing of flow domain

To start with, we have connected a mapped confront coinciding control to the geometry. Next, we connected edge estimating controls to the greater part of the edges of the work and refinement to all edges of the airfoil. Keeping in mind the end goal to dissect liquid stream, stream spaces is part into littler sub areas. The administering conditions are connected and settled inside each of these sub areas. The lattice around the aerofoil is appeared in underneath figure in which fitting exactness is increments as we go towards the aerofoil.

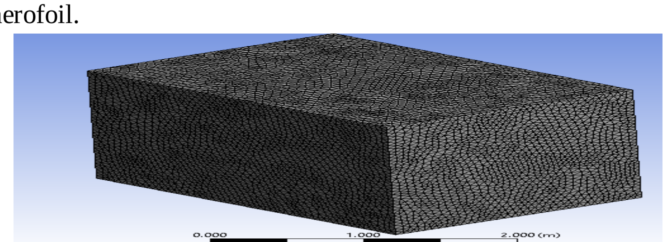


Fig 4: meshed flow domain

C. Turbulence model

It is somewhat difficult to choose which model is best for the analysis, nevertheless, with the help of previous works on the airfoil, in this analysis we have chosen Spalart-Allmaras turbulence model for the simulation. It is a two equation model. Flow is simulated by using equations of conservation of mass and momentum. Finite volume method is used to convert governing equations to algebraic equations.

D. Boundary Conditions

Boundary conditions are incorporated in ANSYS FLUENT. This study has three boundary conditions they are inlet, outlet and wall boundary**.**At the inlet, velocity specification method is taken as magnitude and normal to boundary. Velocity magnitude is taken as 17 m/s. At the wall boundary, the walls are assumed to be adiabatic with no slip condition. The wall is taken as stationary wall and shear condition as no slip. Wall roughness is taken as 0.5. At the outlet pressure is taken as zero

III. RESULTS AND DISCUSSION

In this study, numerical analyses were performed at 17 m/s wind velocity. Coefficient of Lift and drag of NACA 0015 airfoil having dimple at different length from the leading edge at zero angle of attack can be measured. The lift and drag coefficients are obtained as numerical with FLUENT programs for the same conditions. The top bottom and left right boundaries are kept at a distance of 1 chord from airfoil. A mesh independent study was performed to verify that the solution would not change subsequent additional refinements.

Here we are provided dimple on the surface of the airfoil whose effect will be obtained from the analysis. Their is reduction in drag with iterations irrespective of the position of the dimple and nature of the dimple that is inward or outward, but in the 3D model with no dimple their is more drag compared with the airfoil having the dimple.

Here is no considerable change in the lift coefficient value with the dimple on the airfoil it is almost zero at zero angle of attack.

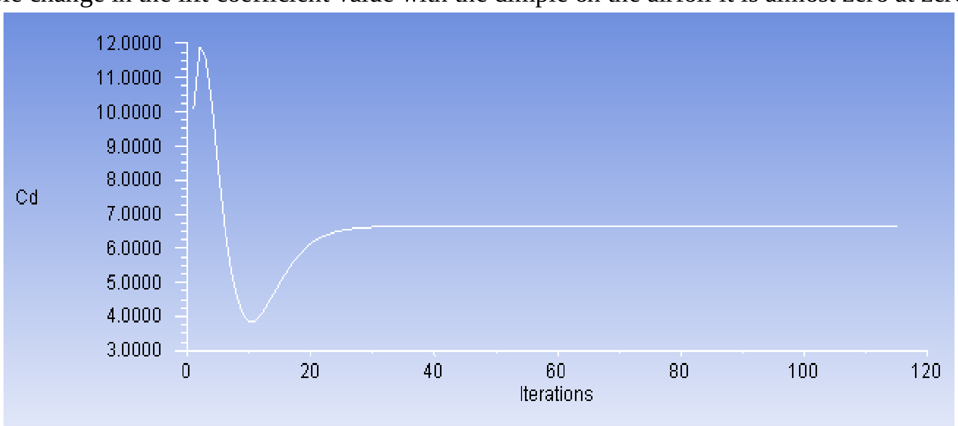


Fig 5: Coefficient of drag with iteration of model with no dimple

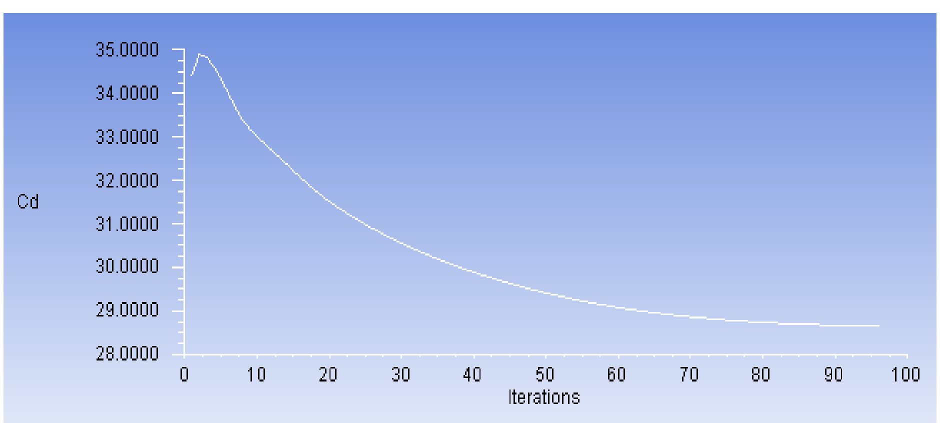


Fig 6: Coefficient of drag with iteration with inward dimple at 216 mm

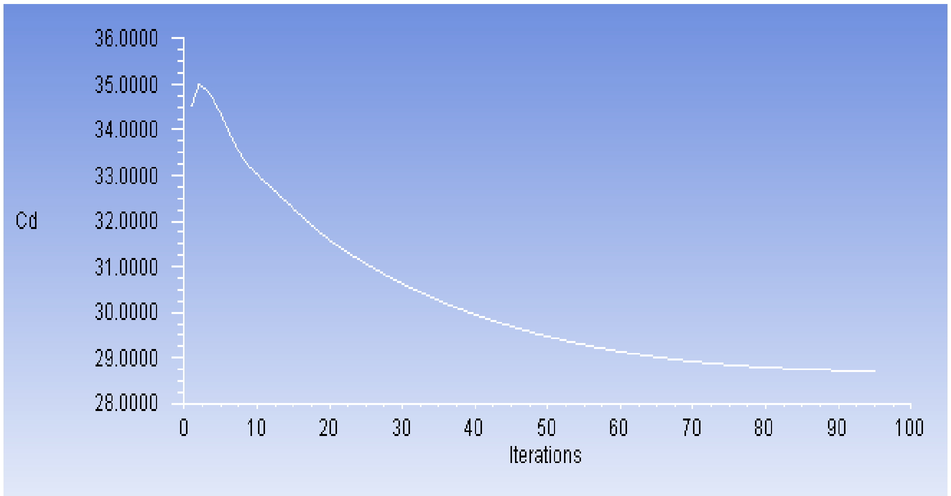


Fig 7: Coefficient of drag with iteration with outward dimple at 206 mm

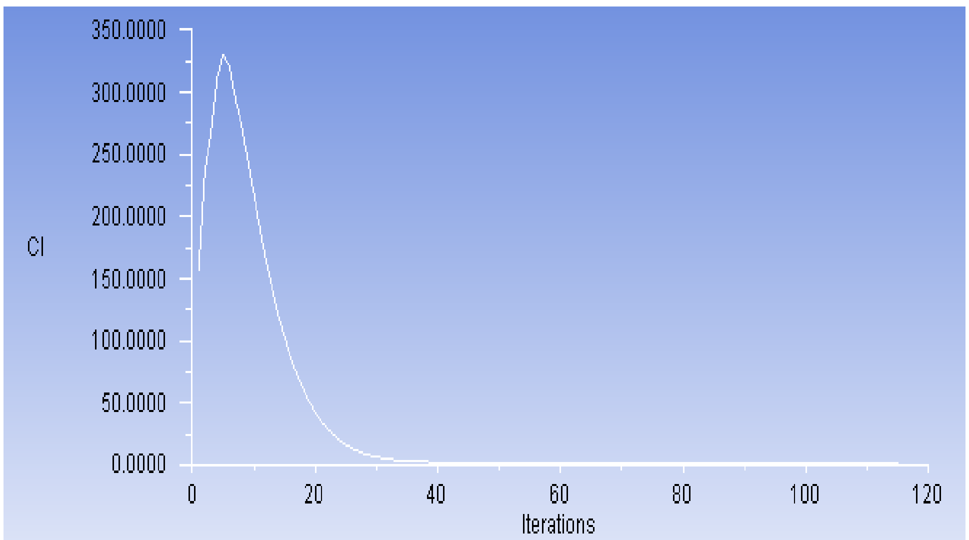


Fig 8: Coefficient of lift with iteration of airfoil 3d model

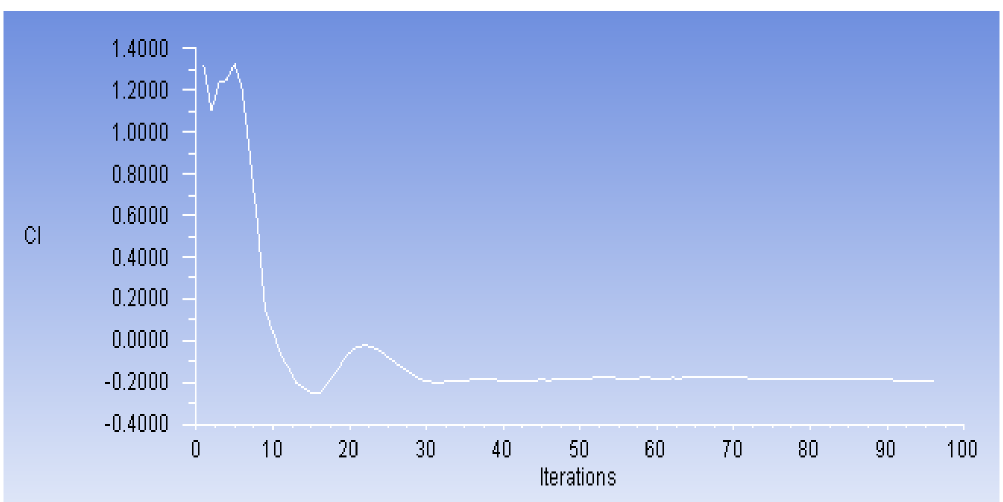


Fig 9: Coefficient of lift with iteration with inward dimple at 206 mm

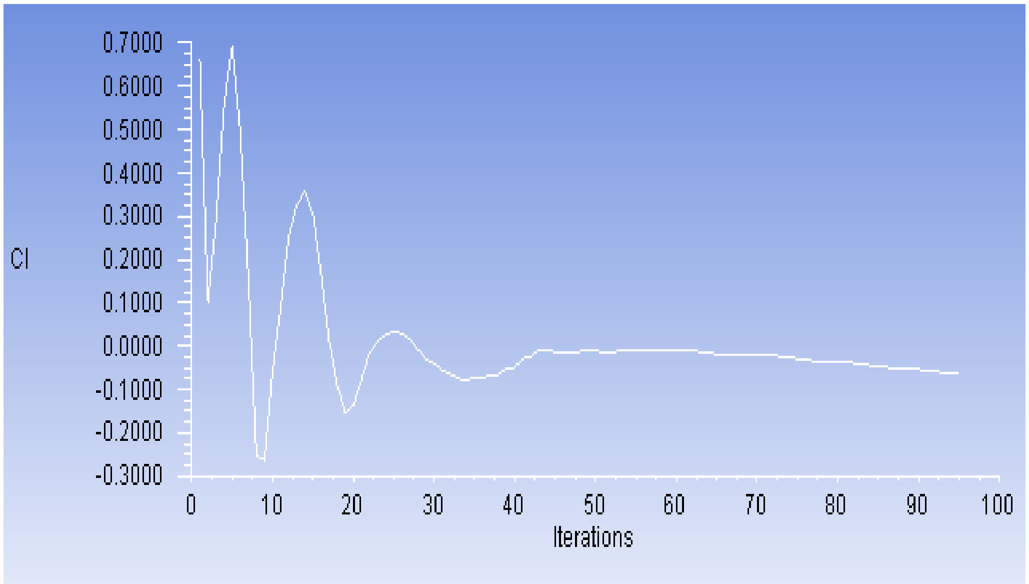


Fig 10: Coefficient of Lift With Iteration With Outward Dimple At 206 mm

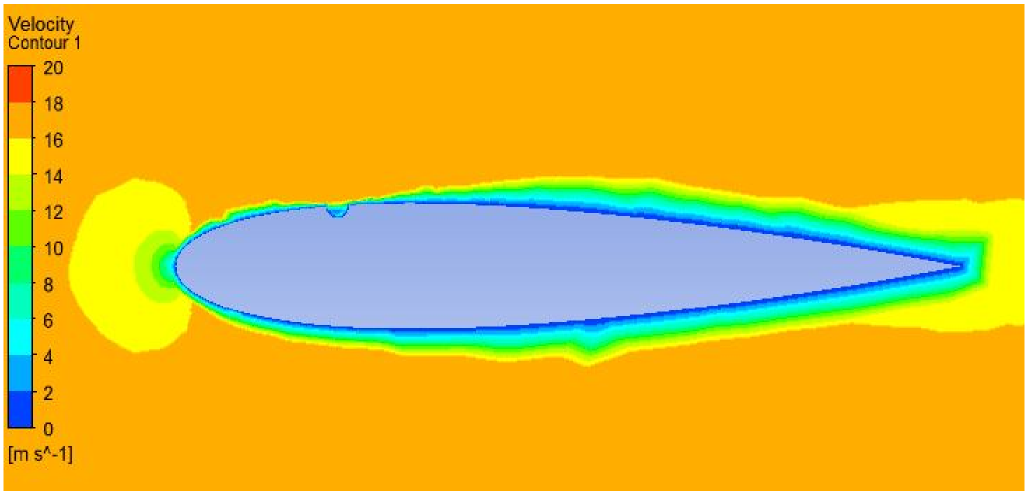


Fig 11: Velocity contour with inward dimple at 206 mm

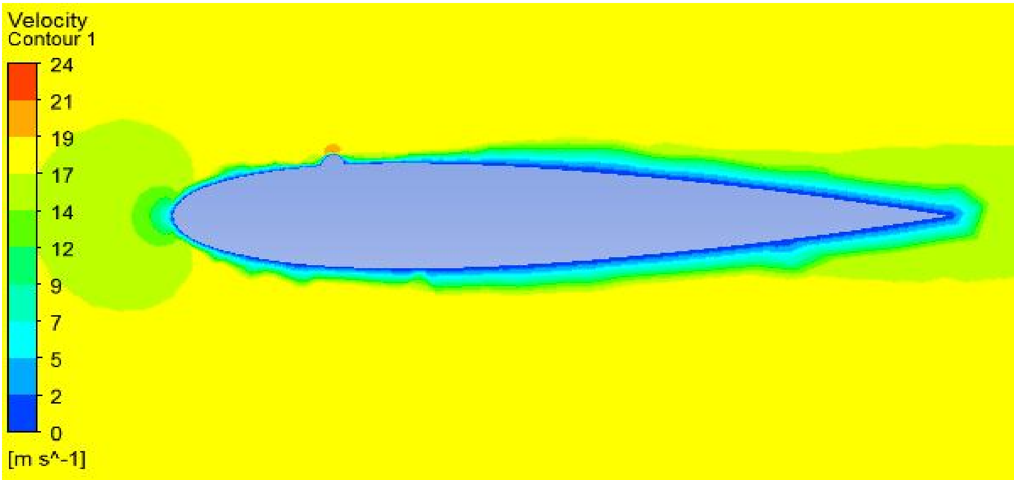


Fig 12: Velocity contour with outward dimple at 206 mm

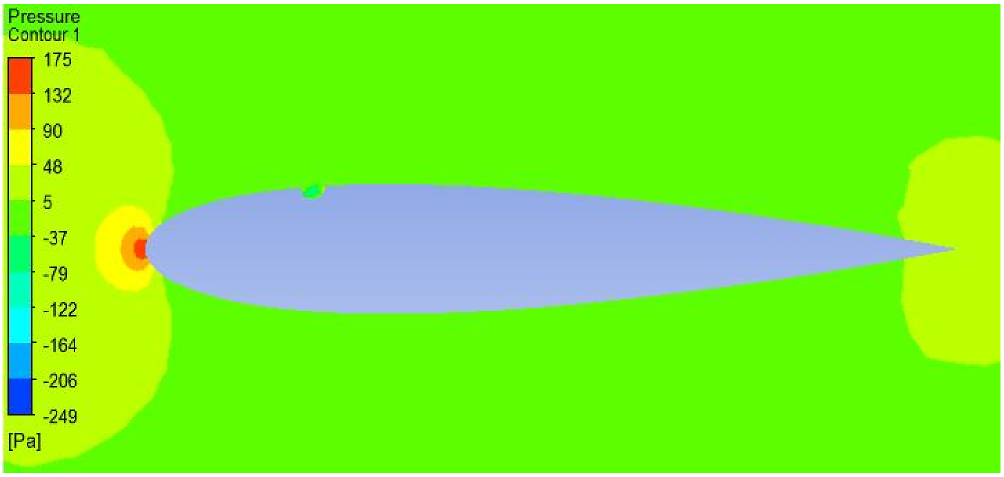


Fig 13: Pressure contour with inward dimple at 206 mm

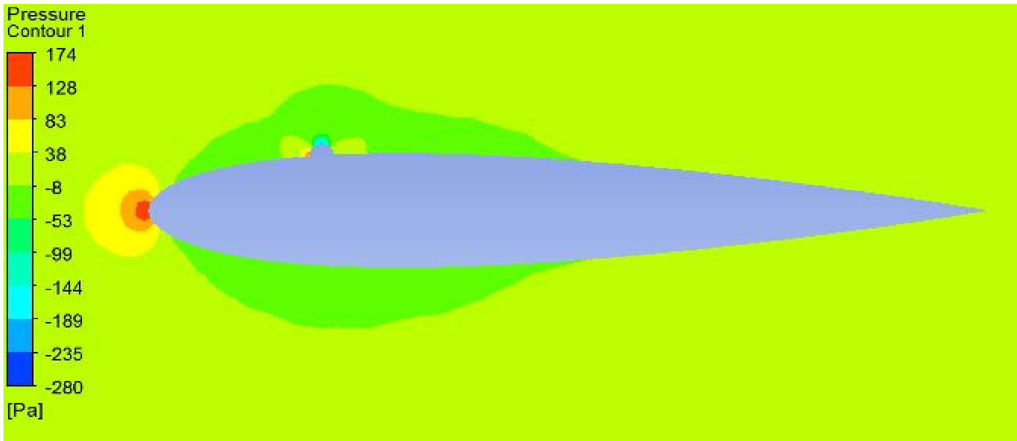


Fig 14: Pressure contour with outward dimple at 206 mm

IV. CONCLUSION

A. Analyses were carried out for zero AOA for 8 different positioned configuration of NACA 0015 airfoil and analysis report were compared with the plain airfoil.

B. Lift coefficient is found to be zero at zero angle of attack

C. It is found from the analysis that in all cases drag coefficient is more in the dimple model than the model having no dimple.

REFERENCES

[1] E.Livya, G.Anitha and P.Valli, "Aerodynamic Analysis of Dimple Effect on Aircraft Wings, World Academy of Science, Engineering and Technology, International Journal of Mechanical, Aerospace, Industrial, Mechatronics and Manufacturing Engineering Vol:9, No:2,2015.W.-K. Chen, Linear Networks and Belmont, CA: Wadsworth, 1993, pp. 123–135.

[2] V B Swamy, "CFD analysis of airfoil NACA 0015" IJMET, Vol 8, Issue 2, Feb 2017, pp. 210-219.

[3] Badri Rajasai, Ravi Tej, Sindhu Srinath, "Aerodynamic effects of dimples on aircraft wings", Proc of the fourth international conference on advances in mechanical, aeronautical and production techniques MAPT 2015.

[4] Deepanshu Srivatsav, "Flow control over airfoils using different shaped dimples", International conference on FDTT 2012, IPCSIT vol.33 IACSIT Press, Singapore.

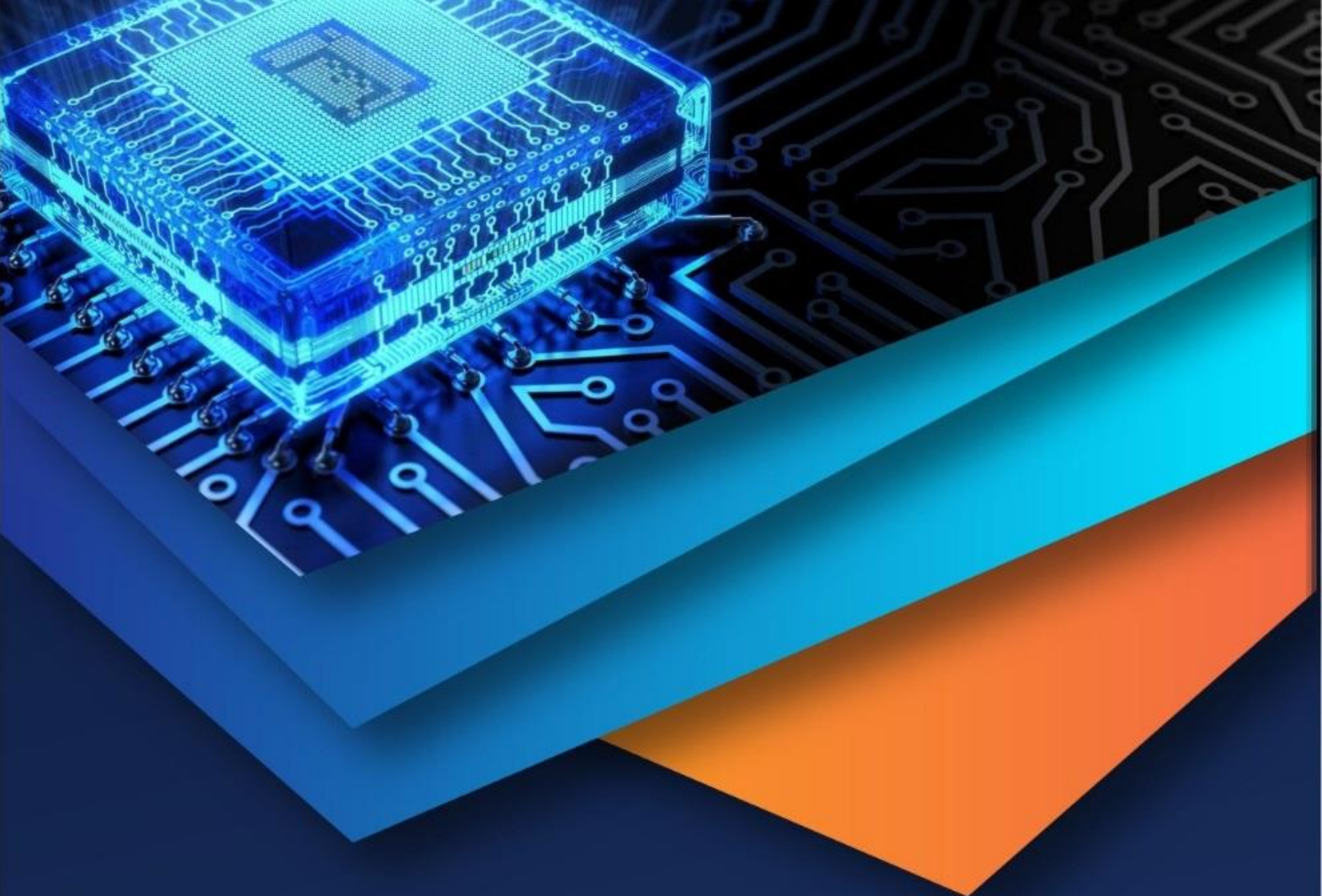
[5] V Soundarya, Anil B A, Venugopal S and Gowreesh S S, "aerodynamic analysis of dimple effect on airfoil", IJEAT, ISSN: 2249-8958, Volume-5, Issue-4, April-2016.

[6] Saarang S, Mahamuni, "A review on study of aerodynamic characteristics of dimple effect on wing", IJAME Volume-2-No.4, July 2015.

[7] UIUC Airfoil Data Site, http://m-selig.ae.illinois.edu/ads/coord_database.html.

[8] John d Anderson, "computational fluid dynamics the basics with applications" TATA McGraw Hill Publications 1995, Applications of CFD page no,[6-22].

[9] Abdulnaser Sayma, "computational fluid dynamics", 1st edition 2009 bookboonpublications, Prediction methods -page no,[8-12].



10.22214/IJRASET

INDEX COPERNICUS

45.98

IMPACT FACTOR: 7.129

TOGETHER WE REACH THE GOAL

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY