OPEN ACCESS Freely available online

Journal of Aeronautics & Aerospace Engineering

Research Article

Drag Enhancement Techniques on Airfoil during Landing using Computational Fluid Dynamics

Ajin Branesh, Ajay Khartri, Fayez SK, Pranjul Agarwal

Department of Aerospace,Assistant Professor,Chandigarh University,India

ABSTRACT

This research aims to increase the drag on the airfoil during landing by optimizing the shape of a NACA 2412 airfoil for 80, 90, 100 m/s velocity for commercial aircraft. In this work, we deployed inward and outward rotating spherical section on the top of the airfoil section, and for the next case, dimples are placed throughout the top surface of the airfoil which will disturb the incoming airflow subsequently increase the drag during landing. As a substitute for the aircraft spoiler, this will reduce the additional structural weight. During the landing phase pilot flourish the drag by deploying spoilers. These spoilers disturbed the streamline flow over the wing and formed vortices at the wing-tip which help the aircraft to descend. Spoilers create additional drag to slow down the aircraft but it creates an induced loss of lift. As a result, stalling speed rise and cause strenuous landing. This work is done by importing airfoil coordinates into Ansys Workbench and simulating the NACA 2412 in ANSYS FLUENT using the k-epsilon turbulence model. Numerical analysis was performed for calibrating the coefficient of drag and lift along with the total drag force created. These can be highlighted by providing pressure, velocity, and Mach number contours for various Mach numbers.

Keywords: Airfoil, Computational Fluid Dynamics, Aircraft landing approach, Drag, Lift

INTRODUCTION

From the start of humanity, man has always dreamt of flying and on December 17, 1903, Wright Brothers gave humanity new wings and hoped for continuous endeavors in this field. ion airfoil and is related to the break-off of the skinny layer (called boundary layer), right at the wing surface. The way during which the flow separation develops to the instant of the complete separation occurrence is strictly hooked into various factors: an airfoil thickness, an airfoil type, an airfoil surface quality, the angle of attack, flow condition, and Reynolds number.[1] Mueller et al., [2] said about the importance of Reynolds number and physical phenomenon effects on an airfoil when it's designed for small aerial vehicles. Flow separation at the airfoil vanguard, transition of the separated shear layer to three-dimensional flow, and subsequently to turbulence. Zhang[3] highlights the geometrical effects on the airfoil flow separation and transition. The takeoff process of flight is often divided into two main stages - acceleration and takeoff. These stages are divided by other certain sub-stages. However, mainly experienced pilots perform the landing process. thanks to the complexity and danger of all actions at the time of landing, this stage is taken into account as the foremost danger consistent with statistics.[6] Actuation is performed via pneumatic vortex generators, impulsively activated to research the transient phenomena like the attachment process and, conversely, to transient re-separation occurring when the actuators are transitioned. [7] By using the variable droop leading-edge (VDLE), the local angle of attack near the leading-edge dynamically decreases when the general angle of attack gets overlarge, then the adverse pressure gradient is often reduced. The airplane engine and propeller combination are meant to supply thrust to beat drag. Their wings are designed to supply a lift to beat gravity.[9] The elimination of flow separation would permit higher angles of attack for several practical applications. Steady blowing on the suction side of the airfoil is found to be effective in controlling the physical phenomenon separation. Flow around NACA0012 and LA203A airfoils are analyzed within his present study. [13]

*Correspondence to: Ajin Branesh, Department of Aerospace,Assistant Professor,Chandigarh University,India; Tel No: 9344008713; E-Mail: branesh2803@gmail.com

Received date: May 13, 2021; **Accepted date:** October 16, 2021; **Published date:** October 27, 2021

Citation: Branesh A (2021) Drag Enhancement Techniques on Airfoil during Landing using Computational Fluid Dynamics. J Aeronaut Aerospace Eng 10: 275

An Airfoil had many practical applications within the branch of aerodynamics, which is why many efforts are given to enhance the performance of the airfoils. Literature suggests that using protuberances along the vanguard can improve airfoil performance remarkably.[14] This computational study is aimed toward studying the consequences of flap deflection angle on aerodynamic coefficients of NACA 2412 airfoil and thus predicting the suitability of using the airfoil with flap at various phases of flight.[15] The aerodynamic forces on golf balls were studied by dropping spinning balls through the horizontal wind flow of the B. F. Goodrich structure. The lift, L, and drag, D, were calculated from the drift of the balls, rotating at speeds, N, up to 8000 r.p.m.[16] Flow separation may be a source of aerodynamic inefficiency, by using vortex generators flow separation is often controlled. this is often of particular benefit to flows around bodies that are vulnerable to separated flows, like bodies in an aerodynamic lift.[18] Dimples behave as protrusions on the surface of the wing. These protrusions generate vortices that reduce the flow separation on the suction side of the wing. This delays and reduces the chord-wise physical phenomenon rate of growth. The add this paper describes the experimental analysis administered on the asymmetrical wing which can reduce the drag and delay the flow separation point over the side wing by using the dimple effect.[19] The surface of the NACA 2412 airfoil is modified by providing semi- circular outward and inward dimples on the lower surface. Initially, volume forcing is introduced to market transition to turbulence. After obtaining sufficient data from this forced case, the explicitly added disturbances are removed, and therefore the simulation runs further. With no forcing the turbulence is observed to self-sustain, with increased turbulence intensity within the reattachment region.[21] The flow characteristics around a symmetrical airfoil NACA 0012 at the incidence and a circular cylinder placed in tandem are studied experimentally at a Reynolds number of 1.5x105 supported the chord length of the airfoil C.It has been seen that the airfoil and therefore the cylinder has considerably suffered from one another. The variation within the flow structures consistent with the attack angle of the airfoil and therefore, therefore, the longitudinal spacing between the airfoil and the cylinder is revealed utilizing the flow visualization photographs.[22] The overall objective of this paper is to enhance the aircraft maneuverability by delaying the flow separation point at a stall and thereby reducing the drag by applying the dimple effect over the aircraft wing. This project includes both computational and experimental analysis of dimple effect on an aircraft wing, using NACA 0018 airfoil. From the literature, it had been found that several attempts are made on the enhancement of drag of airfoil by delaying flow separation. However, no significant studies are reported on CFD analysis of NACA 2412 airfoil with dimples as a surface modification over wings. the present study focuses on studying the consequences of surface modification over NACA2412 airfoil on aerodynamic performance through CFD analysis.

METHODOLOGY

Designing of NACA 2412 Airfoil

The NACA 2412 airfoil coordinates are imported to the Ansys workbench. The model of NACA 2412 airfoil is shown in below figure 1.

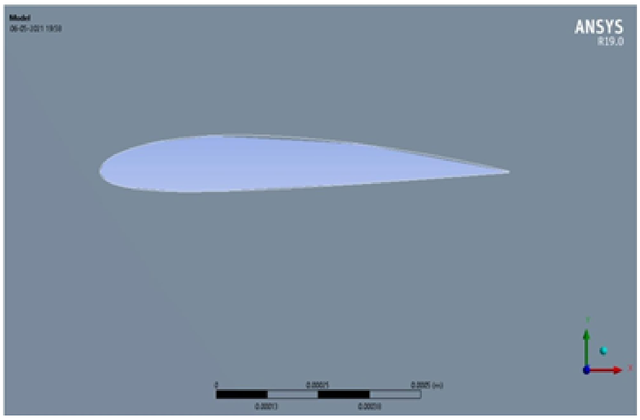


Figure 1: Geometrical model of Naca 2412 airfoil

Modifications to NACA 2412 airfoil

Thomas J. Mueller[1] explained that the stalling of an airfoil depends on its thickness It was reported that the fat airfoil (thickness> 14%) stalls from the leading edge whereas thin airfoils stall from the trailing edge. NACA 2412 being very thin airfoil stalls from trailing edge hence the surface modification in terms of inward and outward dimple outer dimple of diameter 25 mm at a lateral distance of 25 mm from the leading edge is provided. The specifications are given in table 2. The modified NACA 2412 airfoil is as shown in figure 2

TABLE 2: Specifications of wing model

Particulars	Specifications
Airfoil series	2412
Chord	100mm
Type of dimple	Semi-circular
Dia. Of dimple	25 mm
Dimple spacing	25 mm

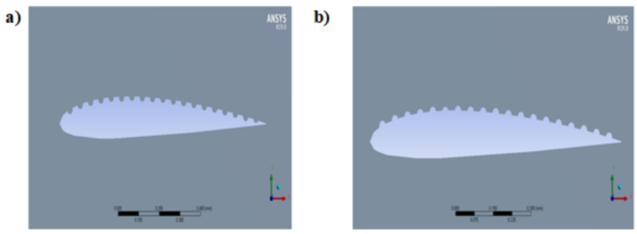


Figure 2: Modification Details (a) inward dimpled airfoil geometry (b) outward dimpled airfoil geometry

Computational Domain

To analyze the flow a computational domain is required hence a 2D planar fluid domain was generated. The domain was sufficiently large whose rectangle length is 20m and diameter is 30 m to simulate free streamflow. This large domain reduces the wall effects. The domain constituted of inlet, walls, airfoil, and outlet.

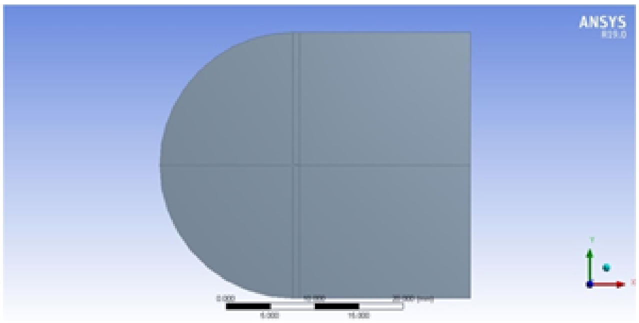


Figure 3 : Geometric shape of Fluid Domain

Meshing

Structured mesh with an edge sizing of 300 division along with bias factor 10 was generated in Ansys R19. The quality of the mesh was smoothening medium, the transition ratio is 0.272. The total number of Nodes 366717 and the Elements 365300. The meshed model of a Modified airfoil and computational flow domain is as shown in figure 4.

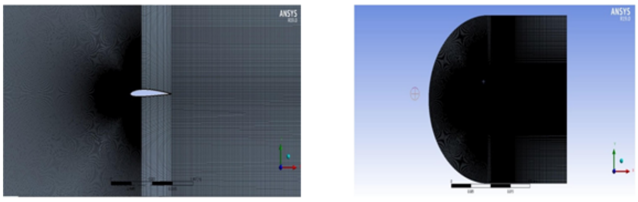


Figure 4: Meshing Details (a) Meshing in dimpled airfoil geometry (b) Meshing in Fluid domain

Boundary Conditions

The boundary conditions used for CFD analysis of airfoils with inward and outward dimples and without dimples at various velocity namely 80m/s, 90m/s, 100m/s. The solver model used in the study is the k ellipse turbulence model (k- ǎ), we have used pressure based model with energy equation on

Transport equations for k- model:

ȷu. ŷk = ŷ.[(Ẽ + ẼT/ůk) ŷk] + ẼTP(u) - ȷ

ȷu. ŷ = ŷ.[(Ẽ + ẼT/ů) ŷ] + C 1 ẼTP(u)/k - C 2ȷ 2/k
Where: P(u) = u: (ŷu + (ŷu)T), and

ẼT = ȷCεk2/ C 1=1.44, C 2=1.92, Cε=0.09, ůk=1.0, ů =1.3

Table 3: Boundary conditions

Parameter Condition/	Value
Flow velocity	80, 90 and 100 m/s
Solver used	Pressure based
Fluid condition	Ideal air fluid
Flow type	Steady state flow
Viscous model	Sutherland with three coefficient
Ambient temperature	300 K
Gauge preesure	101325(pascal)
Turbulent intensity	5%
Turbulent viscosity	Ratio=10

RESULTS AND DISCUSSIONS

Computational fluid dynamics simulations for various velocities ranging from 80, 90, 100 m/s were carried out, and the results of simulations of NACA 2412 airfoils with and without surface modifications were compared. The aerodynamic characteristics such as the coefficient of lift (CL), the coefficient of drag (Cd), and Aerodynamic efficiency (CL/CD) were determined and analyzed for maximum aerodynamic performance. The results obtained from the CFD simulation are as given in below.

WITHOUT DIMPLES				
VELOCITY	CL	CD	LIFT	DRAG
80	1.7536	0.76377	1.0757	0.46781
90	2.2933	0.9077	1.4047	0.55597
100	2.9157	1.0598	1.7859	0.64914
INWARD DIMPLES				
VELOCITY	CL	CD	LIFT	DRAG
80	1.4355	0.90382	0.87924	0.055359
90	1.8129	1.0973	1.1104	0.67207
100	2.2322	1.3063	1.3672	0.80009
OUTWAR D DIMPLES				
VELOCITY	CL	CD	LIFT	DRAG
80	1.234	1.0425	0.75585	0.63852
90	1.5159	1.2781	0.92852	0.78285
100	1.8151	1.5355	1.1117	0.94049

Pressure Contours

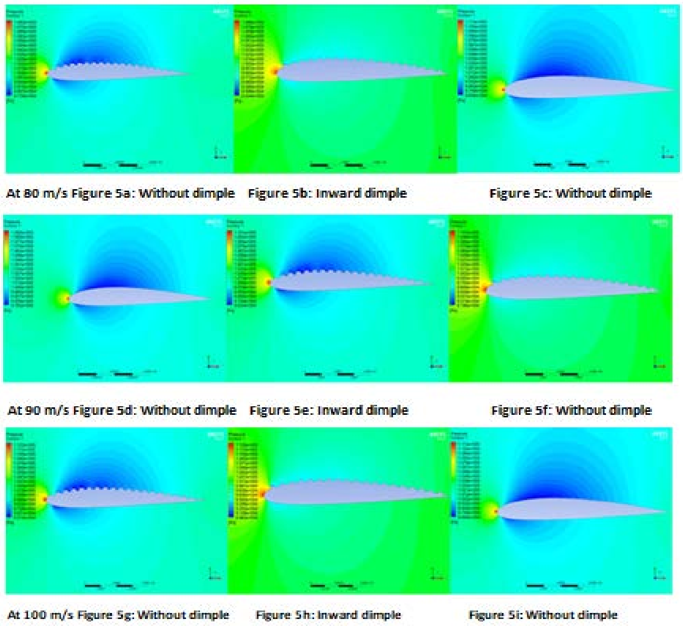


Figure 5: Pressure contours at different velocities

From the base case, the intensity of pressure is concentrated rapidly but for the modified inward dimples the pressure concentration is more widely dissipated which degrades the lifting capability. Additionally, for outward dimples there is no such impact on pressure. We are getting maximum pressure with unmodified airfoil at different velocities. Inward dimple acquiring least pressure while outwards dimples profile shows mild decrement compared to normal airfoil profile as we all know as velocity of air flourish pressure value also rose in the same manner as both properties are independent on each other.

VELOCITY CONTOURS

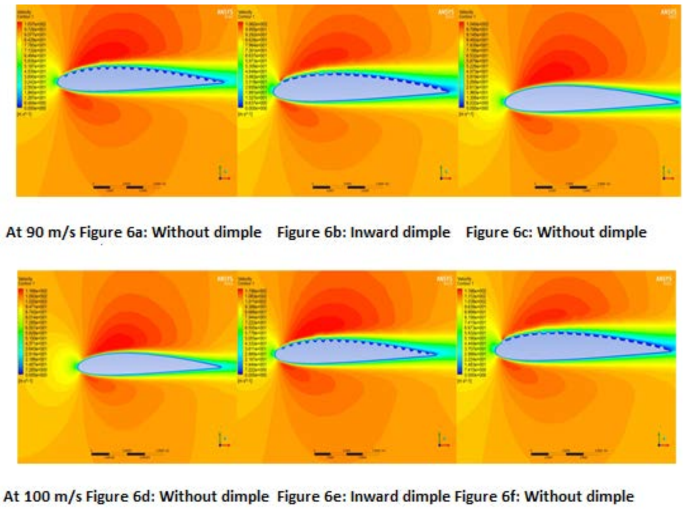


Figure 6: Velocity Contours at different velocities

From velocity contour we can say that maximum flow separation occurs in outward dimples due to its profile. As the incoming streamline flow gets disturbed over the top surface of the airfoil due to outwards semicircular shapes in the case of inwards dimples due to its shape velocity after striking the leading edge reduced compared to other two airfoil.

DENSITY CONTOURS

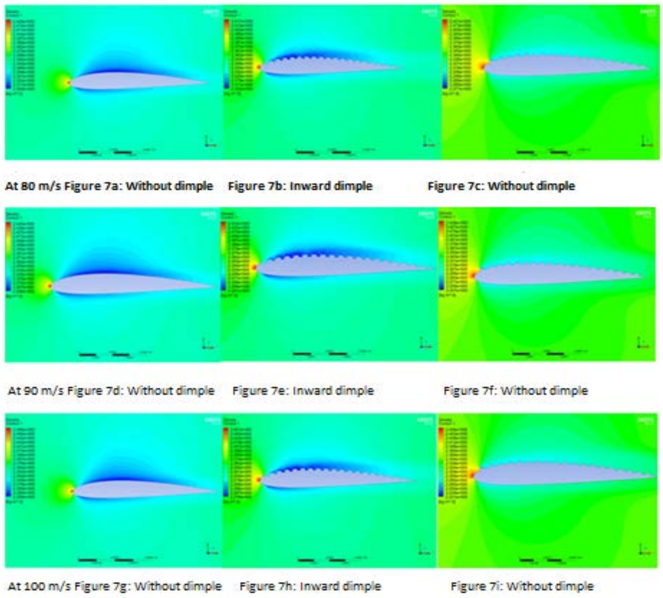
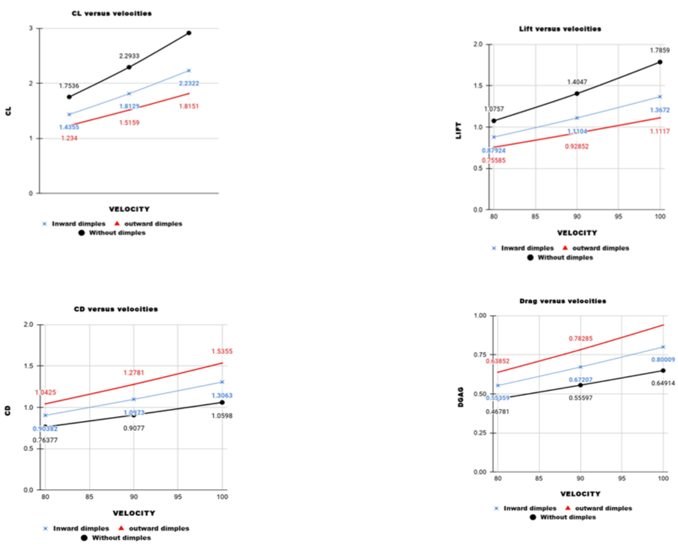


Figure 7: Density Contours at different velocities

Comparing with the base case for three different velocities it has been noted that for inward dimple, because of the flow disturbance imparted on to the flow field the molecules get denser while travelling on the top surface of the airfoil which provides a greater pathway for pressure increment.

COMPARITIVE GRAPH



CONCLUSION

The concept of surface modification through dimple is well worked, with the acute advantage of making an aircraft more efficient by changing flow characteristics. Implementation of inward and outward dimple over NACA 2412 airfoil has proven to be simpler in altering various aspects of the flow structure with varied lift and drag forces. the subsequent conclusions are drawn from the work.

- A comparative study between surface-modified and unmodified NACA 2412 airfoils for constant inlet velocity (80, 90, 100 m/s) showed that surface-modified airfoils introduced more drag compared to an unmodified airfoil.
- Among the surface modifications provided, the outward dimple has better aerodynamic performances. The outward dimple has proved to be more suitable than the onward dimple because it gives more drag as noted by the results of this study.
- While comparing with the graphs we can see that the outward dimple airfoil might be an alternative for a spoiler that exhibits vibration during the aircraft landing phase.
- So our main focus of the project is to increase the drag during the landing At the different speed
- Outwards dimples providing the maximum drag and also reducing lift up to a desirable limit which is essential for a smooth landing
- Inwards dimples also reducing the drag by maintaining more lift value which can be used for landing the aircraft on longer runner without using spoilers which will reduce the weight of the aircraft which will eventually save the fuel

REFERENCES

1. Robert Placeka, Paweł Ruchała, “The flow separation development analysis in subsonic and transonic flow regime of the laminar airfoil”, Transp. Res. Proc, 2018, Vol. 29, pp 323–329.
2. Blair G. Mclachlan and K. Karamcheti, “AN EXPERIMENTAL STUDY OF amfoil·SPOILER AERODYNAMICS”, National Aeronautics and Space Administration, NASA-CR-177328/ 19850006487 Ames Research Center
3. Thomas J. Mueller and James D. DeLaurier, "Aerodynamics of Small Vehicles", Annu. Rev. Fluid Mech. 2003, pp 89–111, DOI: 10.1146/ annual.fluid.35.101101.161102.
4. Wei Zhang, Wan Cheng, Wei Gao, Adnan Qamar, Ravi Samtaney, "Geometrical Effects on the airfoil flow separation and Transition", Comput Fluids, 2015, Vol. 116, pp 60-73.
5. Zare Shahneh A, "Computational Analysis of Generator Effect Tetrahedral Vortex on the Attenuation of Shock Induced Separation" Int. J. Appl Sci Technol, 2012, Vol. 2 No. 3.pp 34- 52
6. W.L. Siauw and J.P. Bonnet, "Transient phenomena in separation control over a NACA 0015 airfoil" Int.J. Heat Fluid Flow, 2017, Vol.67, pp 23-
7. Jianping Niu, JuanmianLei, Tianyu Lu, “Numerical research on the effect of variable droop leading-edge on oscillating NACA 0012 airfoil dynamic stall”, Aerospace Science and Technology, 2018, Vol.72, pp 476–485.
8. Livya G. Anitha P. Valli, “Aerodynamic Analysis of Dimpled Effect on Aircraft Wing”, Int.J. of Mechanical, Aerospace, Industrial, Mechatronics and Manufacturing Engineering,2015. Vol 9, No. 2, pp 350-353.
9. Sandeep Eldho James, Abhilash Suryan, Jiss J Sebastian, Abhay Mohan and Heuy Dong Kim “Comparative Study of Boundary Layer Control Around an Ordinary Airfoil and a High Lift Airfoil with Secondary Blowing”, Comput Fluids. 2017, Vol. 164, pp 50–63
10. Mahon S, Zhand X. “Computational analysis of pressure and wake characteristics on an aerofoil in ground effect”, ASME, 2005, Vol. 127, pp 290–298.
11. John E. Matsson, Connor A. McCain, Connor McGraw, “Aerodynamic Performance of the NACA 2412 Airfoil at Low Reynolds Number”, ASSE 23rd Annual Conference, Conference & Exposition, June 26-29 2016
12. Shivananda Sarkar, Shaheen Beg Mughal, “CFD Analysis of Effect of Variation in Angle of Attack over NACA 2412 Airfoil through the Shear Stress Transport Turbulence Model”, International Journal for Scientific Research & Development. 2017, Vol. 5 (2).
13. Md. Tariqul Islam, Amit M. E. Arefin, M. H. Masud, Monjur Mourshedd, “The Effect of Reynolds Number on the Performance of a Modified NACA 2412 Airfoil”, International Conference on Mechanical Engineering AIP Conf. Proc
14. Pracheta Dutta, Anup Acharya, Shahriar Alam and Saifur Rahman Bakaul, “Computational Study on Effect of Flap Deflection on NACA 2412 Airfoil in Subsonic Flow”, Applied Mechanics and Materials, ISSN: 1662-7482, Vol. 829, pp 9–14 DOI
15. Davies, J.M., The aerodynamics of golf balls, J of Applied Physics, 1949, Vol. 20, No. 9, pp 821–828.
16. Beves C C, Barber T.J, “Aerofoil flow separation suppression using dimples”, The Aeronautical Journal, 2011, Vol. 115, No 1168, Pg 335–344
17. Amit Kumar Saraf, Dr. Mahendra Pratap Singh, Dr. Tej Singh Chouhan, "Effect of Dimple on aerodynamic Behaviour of Airfoil", International Journal of Engineering and Technology. 2017, 9(3), pp 2268-2277.
18. Prasath. M. S, Irish Angelin S, “Effect of Dimples on Aircraft Wing”, GRDJE, April 2017, Vol:2, No. 5, pp 234–242.
19. Bhadri Rajasai, Ravi Tej, Sindhu Srinath, “Aerodynamic effects of a dimple on aircraft wings”, Int. J. Adv. Mech. Aero. Engg. October 2015, Vol 2. No. 2, pp 169–172.
20. M. Alam, N. Sandham, “Direct numerical simulation of ‘short’ laminar separation bubbles with turbulent reattachment”, J. Fluid Mech. 2000, Vol. 410, pp 1–28. DOI:
21. Mustafa sarioglu, “aerodynamic characteristics of the flow around an airfoil and a circular cylinder”, omer halisdemir university, volume 6, number 1, (2017), 191-20
22. Livya G. Anitha P. Valli, “Aerodynamic Analysis of Dimpled Effect on Aircraft Wing”, Int.J. of Mechanical, Aerospace, Industrial, Mechatronics and Manufacturing Engineering,2015. Vol 9, No. 2, pp 350-353