Comparative Study of Dimple Effect on Three-Dimensional Cambered Aircraft Wing

Reshma Sett[1], Prathmesh Pravin Verekar[1], Mohammad Zuber[1,*]

[1] Department of Aeronautical and Automobile Engineering, Manipal Institute of Technology, Manipal Academy of Higher Education, Manipal, 576104, India

ARTICLE INFO	ABSTRACT
  	This paper focuses on aerodynamic analysis over a 3D cambered wing with dimple-like surface modification. Application of the golf ball dimples on airfoils is predicted to be a good alternative for drag reduction. A golf ball with dimples allows for smooth and controlled flight. Models of cambered wings based on NACA 2412 airfoil were created to investigate the dimple effect. A comparative study is carried out between wing models with inward and outward dimples at 50% of chord length locations. Simulations were conducted for two velocities – 30 m/s and 90 m/s and the angle of attacks varying from 0° to 20°. The study is carried out using different platforms for respective processes. Firstly, ANSYS WORKBENCH© and SOLIDWORKS© are used for the geometry creation of the wing model. Next, ANSYS FLUENT© is used for the pre-processing and solver setup stage. Then, TECPLOT©, MATLAB© and FLUENT© are used for the post-processing and analysis stage. The inward dimple showed better aerodynamics when compared with outward dimple and no dimple cases.

1. Introduction

The project is concerned about finding advantages of having dimple-like structures on the aircraft wing. The stall is an inevitable condition that every flight undergoes. Hence, many research works are going on in this field to develop new ideas to increase the stall angle. From previous studies, it is observed that a dimple on the wing structure helps to delay stall. Therefore, keeping these factors in mind, the project is undertaken to have a better insight on this area of research. There are various ways of delaying the flow separation over the wing surface, one of them being introducing surface modifiers on the wing. These devices are of two main categories – first, which require energy supply and second, which do not require the supply of energy [1]. Vortex generators are the devices which lead to delay in boundary layer separation. It has been found that these vortex generators are used in many aeronautical applications for reducing drag and controlling the flow separation in the boundary layer. According to principle, the vortices generated on the surface energise the flow, due

---

* *Corresponding author.*
E-mail address: mohammad.zuber@manipal.edu

to which the kinetic energy level is able to withstand higher pressure rise [2]. Presenting a dimple is considered as a potent strategy in increasing aerodynamic efficiency with higher stall angles. The ideal aerodynamic properties are achieved when the flow is attached. The disturbances generated by dimples lead to laminar-turbulent transition and help in controlling the flow pattern by overcoming adverse gradients [3]. In a real-world scenario, it is seen that a dimple over a golf ball results in a much lower drag as compared to a smooth surface. The question of interest here is if a golf ball has improved flight range, then could a similar effect be achieved for surfaces like an aircraft wing.

Aerodynamic efficiency and stall characteristics are the two important parameters in the study of flow over a wing. Many research works have been carried out in order to achieve better efficiency at optimal manufacturing cost. Stall characteristic for a flight is defined by its Cl versus α curve. A gradual decrease in Cl is preferred and denotes good stalling behaviour [4]. Whereas a steep, sudden decrease in Cl is unfavourable as it requires a very quick response from the pilot end. Studies also show that, new designs incorporated not only lead to improvement in aerodynamic properties but also has an impact on the fuel efficiency. Thus, being advantageous to the industry [5]. A better flow pattern also affects the fuel efficiency positively. As known both the parameters are interrelated. Suitable engine power significantly affects flight endurance and range [6]. The shape and structure of the body plays a vital role in determining the efficiency of the transport as fluid flows past it. The energy used by the vehicle to overcome the aerodynamic loads is greatly affected by the shape of the conveyance [7]. Roughness on the wing surface produces momentum when air flows over it, which leads to a transition of flow from laminar to turbulent. The attached flow properties are also improved due to the irregular flow of air around the dimple region [3]. Considering the material to be used for construction of surface modifiers, generally aluminium is preferred. Strength of aluminium metal is found to be highest when used as alloy with other materials. The alloy AA6061 is one of the most commonly used by aircraft builders [8]. Vortex generators are vane-like structures placed on the wing at specified locations to energise the flow. The height of these structures is generally taken to be half of that of the boundary layer thickness [9]. Separation of flow is a widely studied topic across the domain of aerodynamics. Different types of techniques are applied and tested to observe the pattern of flow around the trailing edge and to study the improvement in lift coefficient. It is seen that modification like use of thick trailing edge airfoil has led to an increase in efficiency by 39.495% with an inclination of airfoil equal to 15° [10]. When air flows along the surface, separation bubbles are formed at the dimple region signifying acceleration of the flow [11]. Previous studies show that, efficiencies differ depending on the shape, number and position of the surface modifiers used. The appearance of vortices is due to shear flow instability. These vortices affect the flow over the wing surface and cause disturbances in transverse and longitudinal directions [1]. There are mainly two types of vortex generators – passive and active. The passive ones eliminate separation by a notable amount, reducing the size of separation zone by around 80%, while the active generators are more powerful than passive generators, with no separation zone in the averaged results [2]. The flow around the wing depends on various factors including inlet velocity, nature of the fluid flowing, roughness and shape of the wing used [12]. Hence it is seen that having surface modifications greatly affect the flow around the aircraft. It is observed that skin friction drag is one of the major contributors in aircraft drag that is linked with higher shear stress and causes flow separation [13]. Use of square and triangle shaped dimples have demonstrated an expansion in lift and enhanced aerodynamic efficiency by improving the maximum lift coefficient or reducing the drag coefficient [14]. Another significant observation studied by researchers is that as the flow passes the dimple region, high momentum is created in the near-wall region, resulting in an increase in skin friction. This skin friction has a prominent role in delaying the flow separation. It is also noticed that higher

maximum lift coefficients are attained when the surface modifier structures are placed near the separation point [15].

2. Methodology

CFD is used as the research tool for carrying out the analysis process. It is one of the widely accepted methods for estimating the aerodynamic coefficients and other related parameters without having manufacturing costs. It bridges the gap between theory and experiments. CFD is used for various applications to identify the complex flow like blood vessels, vehicle aerodynamics, cricket ball and more [5,16-18]. The process of CFD involves a set of Navier-Stokes equation to describe the fluid flow. Partial differential equations are solved numerically using finite volume approach [19]. For solving RANS family of turbulence model is used. It is one of the most economical solvers used widely in the field of research and gives accurate results [20]. Pressure based solver is used along with K-omega SST viscous model for simulation. From previous research works carried out, it is noticed that this viscous model yields accurate values of lift coefficient at low angle of attack [21]. The process gives lift coefficient, drag coefficient around the 3D wing and shows pressure-velocity contours near the dimple region. The results obtained from the CFD process using the mentioned boundary conditions reveals that the solution techniques being used for analysis are realistic in nature.

Figure 1 demonstrates the various stages involved in the computational analysis process. The process starts with geometry generation with the use of suitable modelling tools. Once the model is created and imported it is put to meshing. Meshing generates number of grid points that ensured accuracy of the simulation. Next, the boundary conditions are set on the corresponding surfaces and the system solves the process internally. As and when the results are observed to be converging, the iterations are stopped. Lastly the results obtained are analysed with the aid of graphical and contour representation.
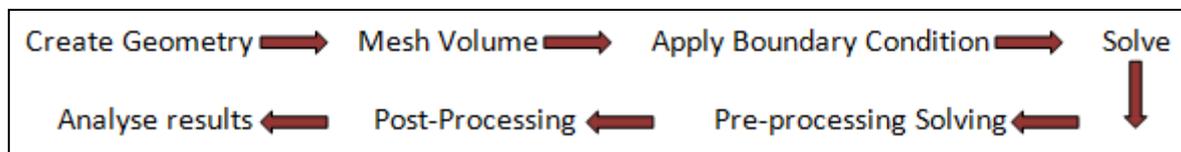
Fig. 1. Steps involved in CFD processing

2.1 Geometry and Domain Creation

The geometry for NACA 2412 3D Wing is created using SOLIDWORKS©. X and Y coordinates of NACA 2412 is imported to SOLIDWORKS© for modelling of the airfoil. Chord of the airfoil is taken as 1 m. Aspect ratio of the cavity/protrusion is taken to be equal to 0.5. A cuboidal shaped domain is created. The size of the domain is taken to be around 9 times the chord of the airfoil on X direction, 5 times the chord on Y direction and same as the span length in the Z direction. Table 1 gives the fluid domain specification. Table 2 lists the airfoil specifications.

Table 1
Domain Specifications

Direction	Dimensions
+X direction	6 m
+Y direction	5 m
+Z direction	1.68 m
-X direction	9 m
-Y direction	5 m
-Z direction	0 m

Table 2
Airfoil Specifications

Property	Value
Airfoil	NACA 2412
Chord	1 m
Span	1.68 m
Effective area	1.68 m^2
Cavity depth/ height	0.0253 m
Cavity radius	0.053 m

Figure 2 shows the three dimensional model created for inward dimple and Figure 3 shows the inward dimple dimensions. Similarly for outward dimple in Figure 4 and Figure 5 displays the 3D model along with dimensions for an outward dimple wing. Smooth curved surfaced dimples are created. Nine inward and outward dimples are situated equidistant from each other on each of the two wings, as shown in Figure 2 and Figure 4 respectively. The dimple-like structures are located at 50% of the chord, throughout the span length. Figure 3 and Figure 5 shows magnified view of the dimples, giving detailed information about the aspect ratio of the surface generated. The radius of the dimple is maintained at 0.053 m whereas the depth for inward dimple and height for outward dimple is taken to as 0.0253 m.
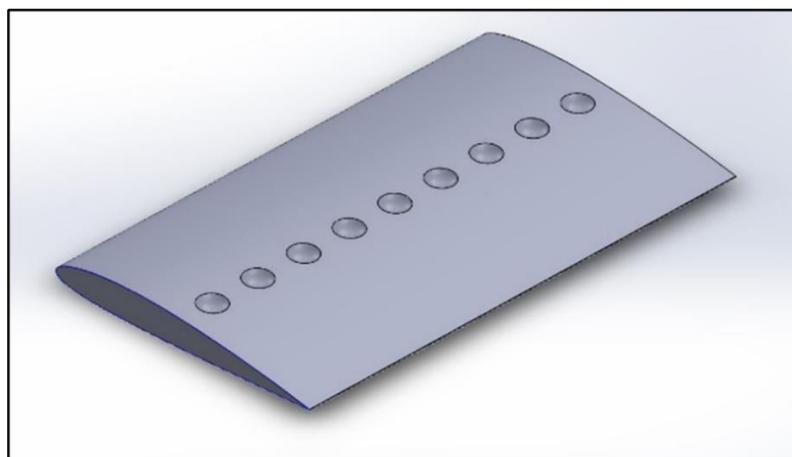
Fig. 2. 3D Model for inward dimple
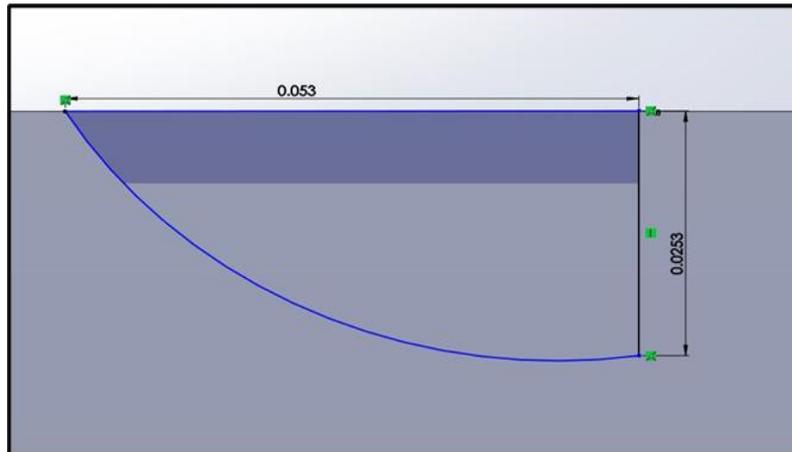
Fig. 3. Inward dimple dimensions
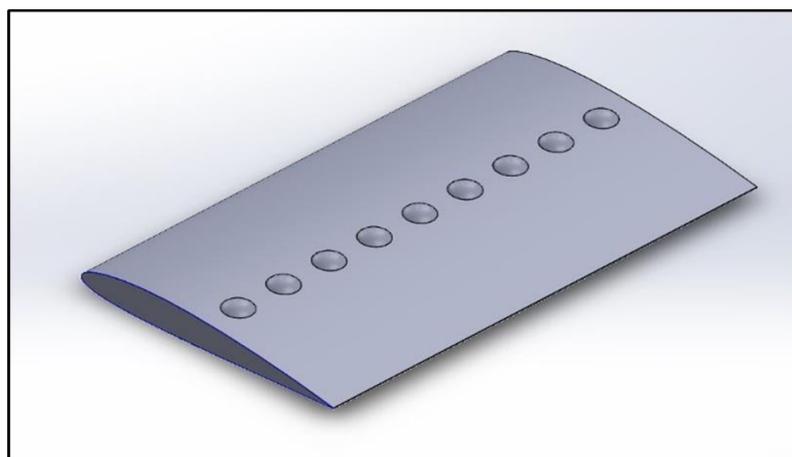
Fig. 4. 3D Model for outward dimple
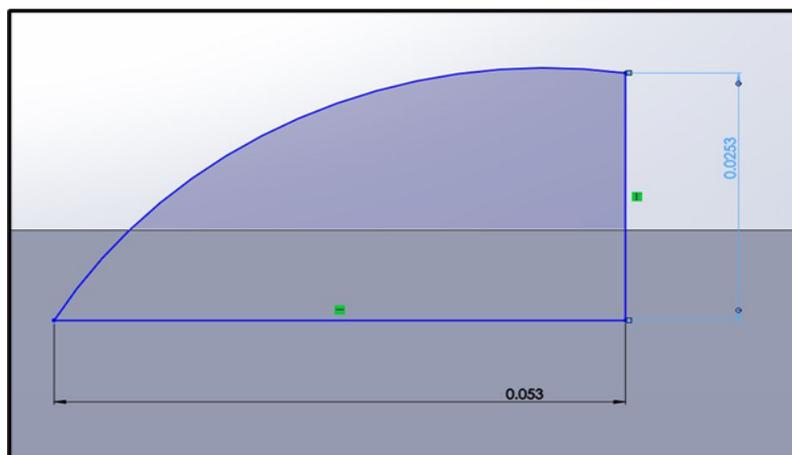
Fig. 5. Outward dimple dimensions

2.2 Solver Setting

ANSYS Fluent is used for simulation. Inlet velocity is 30 m/s, and the angle of attack varies from 0° to 16° for validation. Whereas for comparing the dimpled and plain wing cases, the inlet velocities are taken to be 30 m/s and 90 m/s to vary the Reynolds number. The angle of attack is varied from 0° to 20°.

Table 3 describes about the boundary conditions used for the analysis process. A pressure based solver type is preferred with SST model as it provides more accuracy. Density, pressure and viscosity are maintained at the standard values for sea level. The residual value set controls the behavior of the numerical process and decides upon the convergence.

Table 3

Solver setting

Specifications	Parameters used
Solver used	Pressure based
Relative specification	Absolute
Model used	Viscous model – K-Omega (SST)
Boundary condition	Velocity inlet, pressure outlet
Method used	SIMPLE
Residual monitors (convergence criteria)	upto 10^{-6}
Initialization type	Standard initialization (from inlet)
Density	$1.225 kg/m^3$
Viscosity	$1.789*10^{-5}$ kg/ms
Operating pressure	101325 Pa
Gauge pressure	0 Pa
Spatial discretisation	Second order upwind

3. Results and Discussions

This section presents the results for the validation, the comparison between the plain wing and dimple wings, analysis of various contours and the effect of the Reynolds number on dimple wings.

3.1 Validation

The simulation methodology is validated by comparing the three-dimensional model of the NACA 2412 airfoil to the experimental wind tunnel results [22]. The results were in good agreement with the experimental data. The lift coefficient showed an error range of 3%- 8%, as shown in Figure 6.
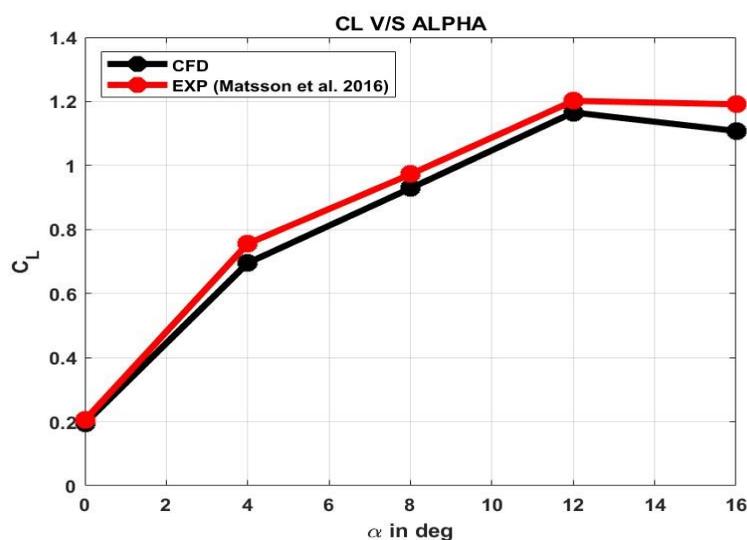
Fig. 6. Validation

3.2 Plain Wing vs Dimple Wing

Stall phenomenon leads an aircraft into a state of chaos and this need to be delayed to have better flight condition. The dimple-like structure on the plain wing increases the stall angle, as shown in Figure 7. The stall angle for the plain wing was 16°, whereas, for both inward and outward dimples, it was 20°.
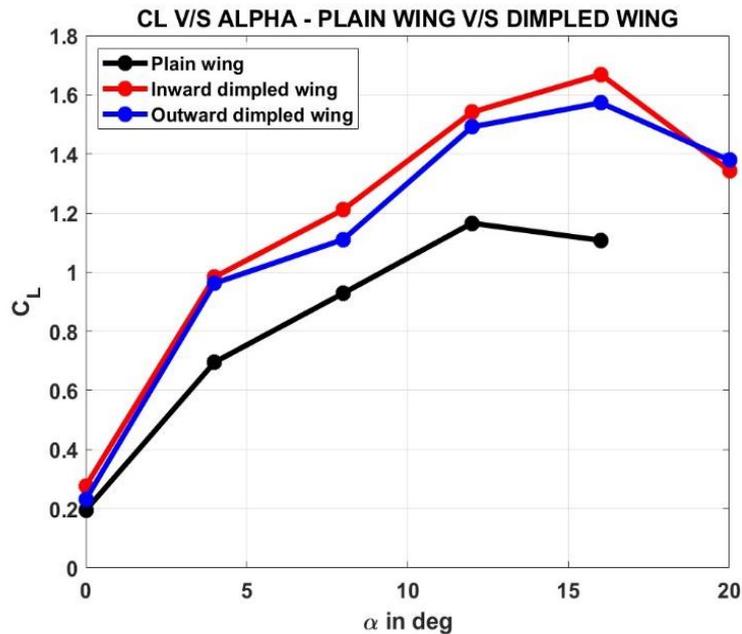
Fig. 7. Coefficient of lift vs angle of attack

3.3 Analysis of Contours

Streamtraces, static pressure contours and velocity contours are shown in Figure 8 to Figure 10.

3.3.1 Plain wing

From stream traces it is observed that flow is well aligned along the surface of the airfoil as it flows from the leading edge towards the trailing edge. As the angle of attack is increased from 0° to 16°, separation of streamlines is noticed near the trailing edge. From pressure and velocity contours, it can be concluded that the highest pressure is seen at the stagnation point. It reaches a minimum value as it moves over the surface and increases again towards the trailing edge. With an increase in angle of attack, the maximum pressure point shifts downward. Similarly, reverse patterns are observed for velocity contours that is minimum at the location of maximum pressure. This phenomenon takes place in accordance with the principle of Bernoulli's Theory.
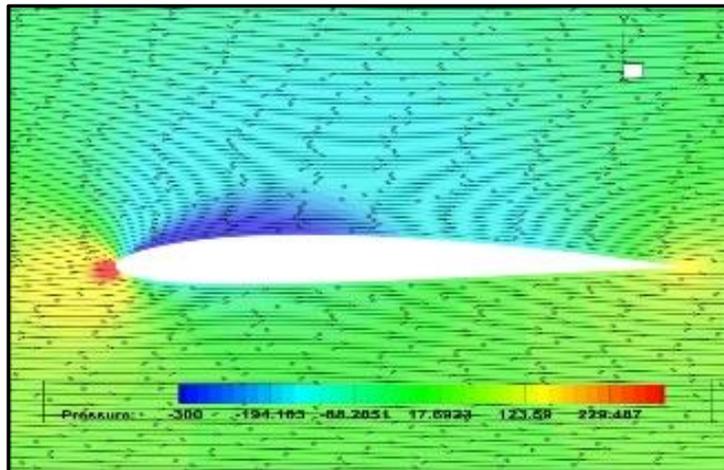
Fig. 8. Streamtraces – plain wing
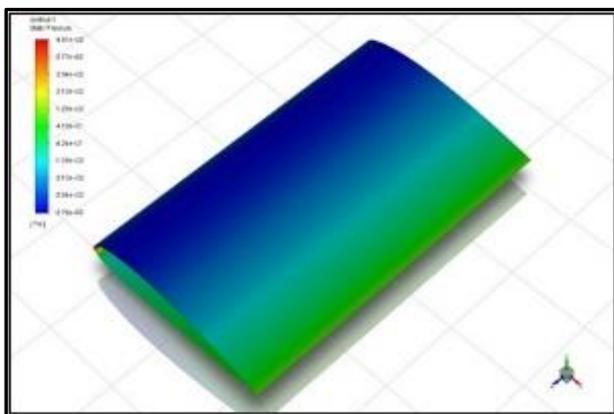
Fig. 9. Static pressure contour- plain wing
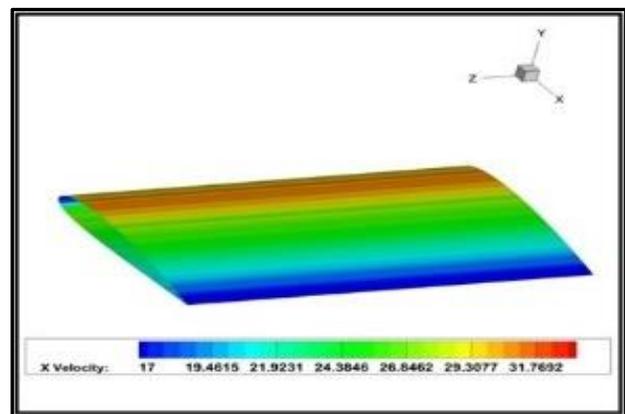
Fig. 10. Velocity contour- plain wing

3.3.2 Wing with inward dimple

From the stream traces in Figure 11 and Figure 12, it is seen that flow circulates in the dimple region forming vortex inside the inward dimple cavity, resulting in the suction of flow (low-pressure region), thus delaying the flow separation. It is visible from Figure 13 that maximum pressure observed at the stagnation point. At higher angle of attacks flow starts separating towards the trailing edge. Consequently, the velocity contour in Figure 14 shows minimum value at the stagnation point in accordance with the Bernoulli's principle.
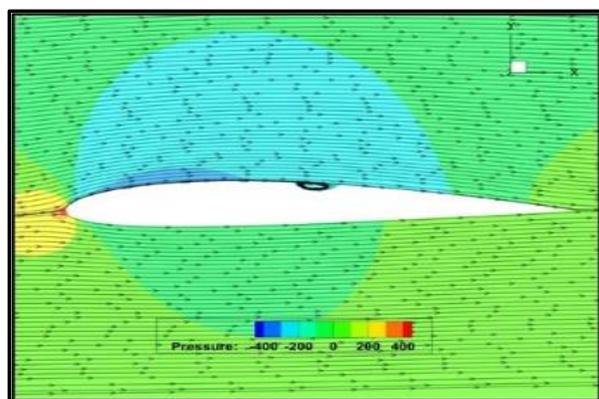
Fig. 11. Streamtraces – inward dimple
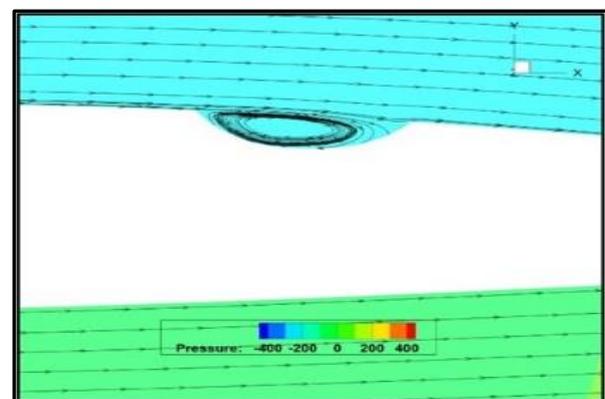
Fig. 12. Magnified view- inward dimple
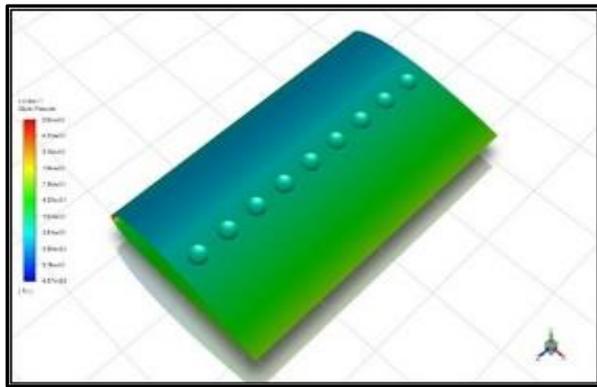
Fig. 13. Static pressure contour- inward dimple
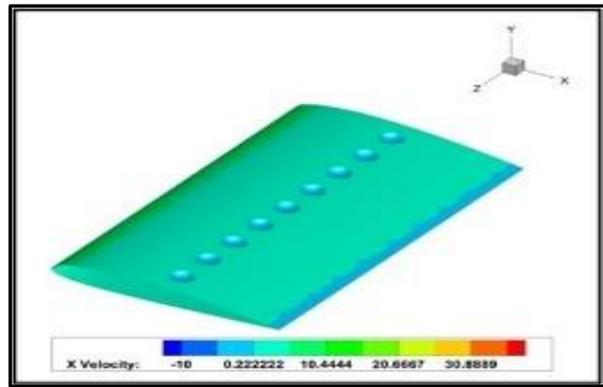
Fig. 14. Velocity contour- inward dimple

3.3.3 Wing with outward dimple

The flow moves smoothly over the dimple region following the usual streamline pattern as shown in Figure 15 and Figure 16. Similar observation is done for outward dimple, there is a low-pressure region formed at the tip of protrusion which results in the suction of flow and delay in flow separation. From Figure 17, pressure contour, and Figure 18, velocity contour, it can be observed that maximum pressure and minimum velocity is attained at the stagnation point at the leading edge of the wing. Also due to the protruded surface a suction region is created and a very low value of pressure is seen at that particular area.
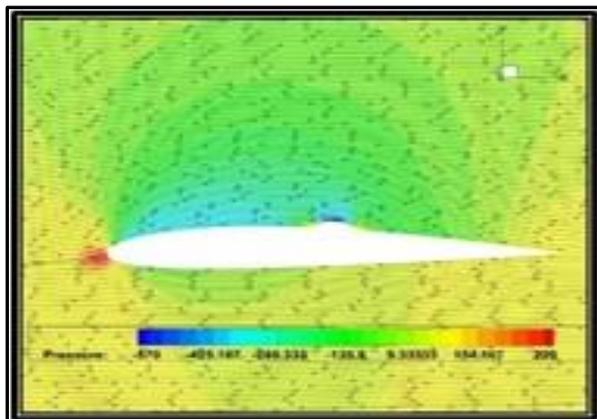
Fig. 15. Streamtraces- outward dimple
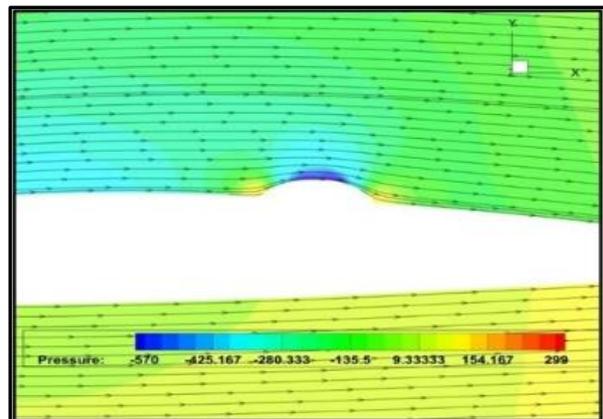
Fig. 16. Magnified view- outward dimple
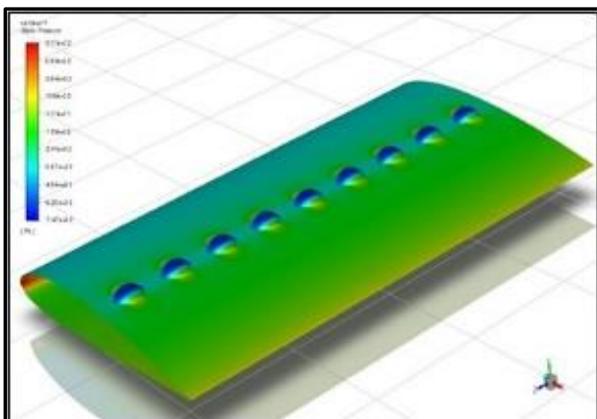
Fig. 17. Static pressure contour-outward dimple
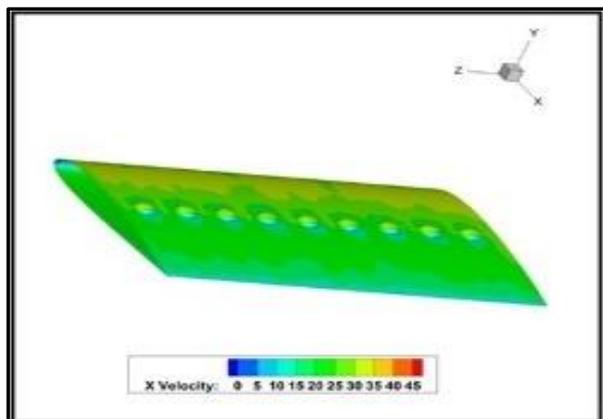
Fig. 18. Velocity contour- outward dimple

3.4 Varying Reynolds Number

The Reynolds (Re) number is varied using two different inlet velocities, 30 m/s and 90 m/s, and the corresponding Reynolds numbers are 2030387 and 6091160. It is observed from Figure 19 that the higher Reynolds number generates more lift for an inward dimple. At the same time, the effect of the Reynolds number on the outward dimple is not distinct.
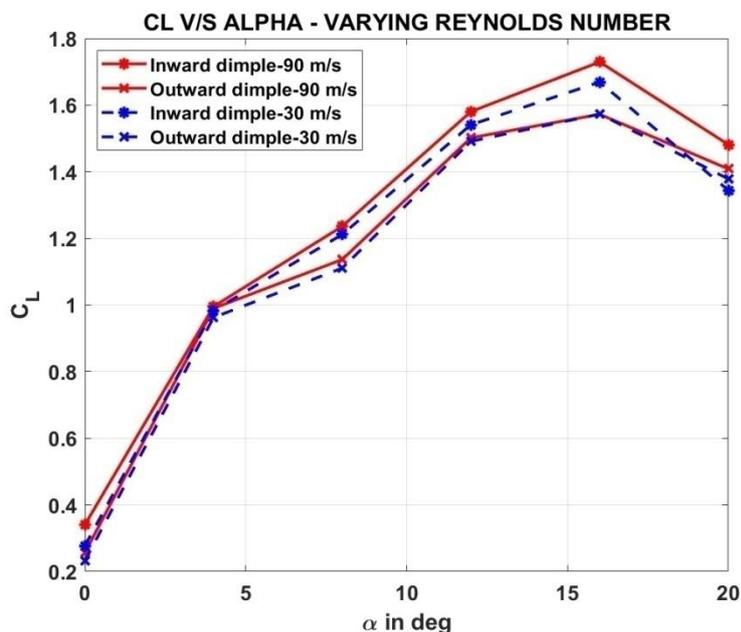
Fig. 19. Varying Reynolds number

4. Conclusion

Dimple surface modifications on the surface on the wing have resulted in better aerodynamic characteristics. It can be inferred that as vortex are generated in and around the dimple surface it energizes the flow, hence resulting delay in flow separation. Also, the amount of lift generated is found out to be much higher than normal plain 3D wing. Therefore, the new modified wing was proven to be beneficial. From the analysis performed the following points can be concluded

i.   Introduction of dimple-like structure – both inward cavity type and outward protrusion type lead to delay in the stall. The stall angle was 20° for both inward and outward dimples.
ii. From stream traces, it is observed that the dimple-like cavity delays the flow separation. For an inward dimple, the vortex formed inside the dimple causes the flow to reattach to the wing.
iii. Considering inward and outward dimples, inward dimples give a higher lift than the latter.
iv. For an increase in Reynolds number, the lift coefficient also increased for the inward dimple case.

5. Future Scope

As this technique of introducing dimples on the surface of the wing is seen to delay the stall and have better aerodynamic characteristics, this research work can be extended in future by altering the shape and size of surface modifier used. The geometry of the dimple can be changed by altering the aspect ratio and the same model can be analysed again noting the variations. Also wing with dimples placed at various locations can be modelled and tested against varied inlet (Re) numbers. Taking the manufacturing process into account, study can be further carried out on the choice of materials appropriate for construction of the modified wing and environmental effects on the cavity/protruded area. Scope of this research work widens by moving onto experimental tests after carrying out computational analysis of the modified wing.

Acknowledgement

References

[1] Zanin, B. Yu, V. V. Kozlov, and A. M. Pavlenko. "Control of flow separation from a model wing at low Reynolds numbers." *Fluid Dynamics* 47, no. 3 (2012): 403-410. https://doi.org/10.1134/S0015462812030143

[2] Shan, Hua, Li Jiang, Chaoqun Liu, Michael Love, and Brant Maines. "Numerical study of passive and active flow separation control over a NACA0012 airfoil." *Computers & Fluids* 37, no. 8 (2008): 975-992. https://doi.org/10.1016/j.compfluid.2007.10.010

[3] Merryisha, Samuel, Parvathy Rajendran, and Sher Afghan Khan. "CFD Simulation of NACA 2412 airfoil with new cavity shapes." *Advances in Aircraft and Spacecraft Science* 9, no. 2 (2022): 131-148.

[4] Houghton, Edward Lewis, and Peter William Carpenter. *Aerodynamics for engineering students*. Elsevier, 2003.

[5] Kamal, Muhammad Nabil Farhan, Izuan Amin Ishak, Nofrizalidris Darlis, Daniel Syafiq Baharol Maji, Safra Liyana Sukiman, Razlin Abd Rashid, and Muhamad Asri Azizul. "A review of aerodynamics influence on various car model geometry through CFD techniques." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 88, no. 1 (2021): 109-125. https://doi.org/10.37934/arfmts.88.1.109125

[6] Otkur, Murat. "Altitude Performance and Fuel Consumption Modelling of Aircraft Piston Engine Rotax 912 S/ULS." *Journal of Advanced Research in Applied Sciences and Engineering Technology* 23, no. 1 (2021): 18-25. https://doi.org/10.37934/araset.23.1.1825

[7] Nazaruddin, Nazaruddin, Syafri Syafri, and Yudi Saputra. "Body Shape Selection of" Bono Kampar" For Urban Concept Student Car Formula to Fulfill Indonesian Energy-Saving Standards ("KMHE") with Aerodynamic Analysis." *CFD Letters* 12, no. 12 (2020): 104-114. https://doi.org/10.37934/cfdl.12.12.104114

[8] Abd El-Hameed, Afaf M., and Y. A. Abdel-Aziz. "Aluminium Alloys in Space Applications: A Short Report." *Journal of Advanced Research in Applied Sciences and Engineering Technology* 22, no. 1 (2021): 1-7. https://doi.org/10.37934/araset.22.1.17

[9] Seshagiri, Amith, Evan Cooper, and Lance W. Traub. "Effects of vortex generators on an airfoil at low Reynolds numbers." *Journal of Aircraft* 46, no. 1 (2009): 116-122. https://doi.org/10.2514/1.36241

[10] Jaafar, Reham K., M. A. Abdelrahman, Mina G. Mourad, Adnan A. Ateeq, and M. Moawed. "Modified Trailing Edge Impact on the Aerodynamic Performance of Wind Turbine Airfoil." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 91, no. 2 (2022): 133-144. https://doi.org/10.37934/arfmts.91.2.133144

[11] Livya, E., G. Anitha, and P. Valli. "Aerodynamic analysis of dimple effect on aircraft wing." *International Journal of Aerospace and Mechanical Engineering* 9, no. 2 (2015): 350-353.

[12] Asral, Asral, Kaspul Anuar, and Agung Soegihin. "Aerodynamic Analysis of Unnamed Aerial Vehicle Serindit V-2 Using Computational Fluid Dynamics." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 93, no. 1 (2022): 83-93. https://doi.org/10.37934/arfmts.93.1.8393

[13] Ali, Jaffar Syed Mohamed, Mohd Farid Amran, and Nurul Aziz Mohamad. "Experimental Study on the Effect of Boundary Layer Control on the Aerodynamics Characteristics of NACA 0021 Aerofoil." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 94, no. 1 (2022): 129-137. https://doi.org/10.37934/arfmts.94.1.129137

[14]   Devi, P. Booma, and Dilip A. Shah. "Computational Analysis of Cavity Effect over Aircraft Wing." *International Journal of Aerospace and Mechanical Engineering* 10, no. 4 (2016): 803-806.

[15]   Zhen, Tan Kar, Muhammed Zubair, and Kamarul Arifin Ahmad. "Experimental and numerical investigation of the effects of passive vortex generators on Aludra UAV performance." *Chinese Journal of Aeronautics* 24, no. 5 (2011): 577-583. https://doi.org/10.1016/S1000-9361(11)60067-8

[16]   Khader, Shah Mohammed Abdul, Adi Azriff, Cherian Johny, Raghuvir Pai, Mohammad Zuber, Kamarul Arifin Ahmad, and Zanuldin Ahmad. "Haemodynamics behaviour in normal and stenosed renal artery using computational fluid dynamics." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 51, no. 1 (2018): 80-90.

[17]   Kalliyath, Shaheen Ayyoobi, Mohammad Zuber, Satish Shenoy, and Anand Pai. "Drag and Lift Characteristics of Different Handlebars and Front-Fairing Combinations Used in Two-Wheelers: A CFD Approach." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 93, no. 1 (2022): 136-147. https://doi.org/10.37934/arfmts.93.1.136147

[18]   Kalburgi, Sagar, Ashwini Rathi, Mukund Narayan, Laxmikant G. Keni, K. N. Chethan, and Mohammad Zuber. "Computational fluid dynamics study of cricket ball aerodynamics associated with swing." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 75, no. 2 (2020): 125-136. https://doi.org/10.37934/arfmts.75.2.125136

[19]   Hoffmann, Klaus A., and Steve T. Chiang. *Computational Fluid Dynamics. 4th ed*. Engineering Education System, 2000.

[20]   Hang, Leong Chee, Nur Athirah Nadwa Rosli, Mastura Ab Wahid, Norazila Othman, Shabudin Mat, and Mohd Zarhamdy Md Zain. "CFD Analysis on Propeller at Varying Propeller Disc Angle and Advance Ratio." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 96, no. 1 (2022): 82-95. https://doi.org/10.37934/arfmts.96.1.8295

[21]   Khan, Sher Afghan, Musavir Bashir, Maughal Ahmed Ali Baig, and Fharukh Ahmed Ghasi Mehaboob Ali. "Comparing the effect of different turbulence models on the CFD predictions of NACA0018 airfoil aerodynamics." *CFD Letters* 12, no. 3 (2020): 1-10. https://doi.org/10.37934/cfdl.12.3.110

[22]   Matsson, John E., John A. Voth, Connor A. McCain, and Connor McGraw. "Aerodynamic performance of the NACA 2412 airfoil at Low Reynolds Number." In *2016 ASEE Annual Conference & Exposition*. 2016.