FLOW SEPARATION CONTROL OF A NACA 4415 AIRFOIL BY PASSIVE TECHNIQUE

O. Khiat, K. Talbi

Dept. of Mechanical Engineering, University of Constantine01
Oussamaero@gmail.com

Abstract: The present work describes change in aerodynamic characteristics of an airfoil by applying certain geometric modification in form of a channel goes from the leading edge to the trailing edge on the upper surface. A computational fluid dynamics CFD is done to simulate the steady, transonic, compressible, 2-Dimensional turbulent flow over a NACA-4415 airfoil. At first the CFD analysis is done to the clean airfoil without modification for different angles of attack (0°, 5°, 10° et 15°), thereafter based on the results obtained the angle 15° “where the boundary layer separation is evident” is chosen to apply the geometric modification. The aim of this modification is to control the flow separation on the airfoil. The CAD model is prepared in Gambit and simulations are carried out in Fluent. The overall aim of the study is improved the aerodynamics performance of an airfoil by flow manipulation. The results justify the increase in the overall lift and reduction of the flow separation on the airfoil.

Keywords: airfoil, boundary layer control, CFD, Drag and Lift, flow separation.

I. INTRODUCTION

Flow control is one of the most important research areas in the fluid mechanics that has been investigated by many researchers. One of the important targets of the flow control is to control flow separation with respect to overall drag reduction. Flow separation occurs when the boundary layer moves far enough against an adverse pressure gradient that the speed of the boundary layer relative to the object falls almost to zero. The fluid flow becomes detached from the surface of the object, and instead takes the forms of eddies and vortices. Boundary layer separation is mostly an undesirable phenomenon. It can often result in a lot of negative effect, decreasing lift, increasing drag, amplifying the sonorous nuisances and the structure vibration is some of these effect. Separation delay and resulting separation zone shortening are of great interest in a number of industrial branches, e.g. turbomachinery, car and aircraft aerodynamics, etc.

Therefore, there is a strong tendency to delay the occurrence of flow separation. The separation control can lead to a lot of beneficial consequences, e.g. improving performances and maneuverability of vehicle, minimizing fuel consumption, Conservation of fossil oil reserves and decreasing in carbonic emissions, etc.

Fluid flow separation can be controlled by various ways such as motion of the solid wall, tangential blowing and suction, continuous suction and blowing etc. These are ancient methods. The modern techniques are the steady and pulsed jets, oscillatory fluid injection, dielectric barrier discharge plasma actuators, synthetic jet, vortex generator jets smart acoustically active surfaces etc. However, the surface and geometric modifications that are relatively classic techniques are the most used in the industry at present. As an example the passive vortex generators are the most frequently used modifications to an aircraft lifting surface to improve the maneuverability of the aircraft. Vortex generators create turbulence by
creating vortices which delays the boundary layer separation resulting in decrease of pressure drag and also increase in the angle of stall. It helps to reduce the pressure drag at high angle of attack and also increases the overall lift of the aircraft. In recent years many studies have been conducted on flow separation control. Prandtl (1904) was the first scientist who employed boundary layer suction to indicate its significant impacts on stream lines in 1904. He used suction on cylindrical surface to delay boundary layer separation. Boundary layer separation would be eliminated almost entirely by suction through a slot on the back of the cylinder. First experiments on flow separation control on an airfoil were done in late 1930’s to 1940. The effect of suction on boundary layer separation using slots on airfoil surface in wind tunnels was evaluated by NACA Langley memorial scientists. The first flight experiments in which seventeen suction slots were installed between 20 and 60 percent of the chord length was done. Employed airplane in this experiment was B-18 airplane A. L. Braslow (1999). The conventional passive vortex generator was first developed by Taylor (1947) to prevent boundary-layer separation in wind tunnel diffuser. The first systematic study of vortex generations and their effects on the boundary-layer was performed by Schubauer and Spangenberg (1950). Since then the vortex generators have been successfully applied to lifting surfaces in many aeronautical applications for control of flow separation and reduction of drag in a turbulent boundary layer, J.Casper (2003). Lin (2002) gave an in-depth review of boundary layer flow separation control by the passive vortex generators. More recently, Godard et al. (2006) conducted an experimental study to optimize the standard passive vane-type vortex generators over a bump in a boundary layer wind tunnel, which mimics the adverse pressure gradient on the suction side of an airfoil at the verge of separation. Two types of vortex generators configurations that produce co- and counter rotating longitudinal vortices were tested, and the counter-rotating device appears to be more effective. Iperas (2010) studied flow separation control on a NACA 4415 airfoil through different suction arrangements and increased the maximum lift coefficient value by 20 percent. Gene et al. (2011) studied on the numerical effects of suction and blowing on the NACA 2415 airfoil at transition zone. Although separation bubbles were not entirely eliminated in suction and blowing simulation, they either reduced or moved into the downstream. For synchronous suction and blowing, separation bubbles were exterminated completely, lift coefficient increased and drag coefficient decreased. They also showed the best results were obtained with the single suction jet, intermediate results were obtained with the multi jets and the worst results were obtained with the blowing jets. Yagiz et al. (2011) worked on drag optimization on Rae5243 airfoil in transonic conditions through suction. By optimum parameters selection they increased the lift coefficient, 3.17 percent, and decreased the total drag coefficient, 3.13 percent. In addition, Yousefi et al. (2012) reviewed the investigations on used methods in suction and blowing systems to increase or decrease drag and lift coefficient.

In this study, control of flow over a NACA4415 airfoil under Re=1×10^7 at different angles of attack is computationally investigated using geometric modification in form of a channel goes from the leading edge to the trailing edge on the upper surface. At first the CFD analysis is done to the clean airfoil without modification for different angles of attack (0°, 5°, 10° et 15°), thereafter based on the results obtained the angle 15° “where the boundary layer separation is evident” is chosen to apply the geometric modification. The aim of this modification is to control the flow separation on the airfoil. The purpose of the current study is to show the effect of this geometric modification on the behavior of boundary layer over an airfoil and compare the aerodynamics coefficients to the clean airfoil case.
II. GOVERNING EQUATIONS

For all flows, the solver solves conservation equations for mass, momentum and energy. Additional transport equations are also solved when the flow is turbulent. The equation for conservation of mass or continuity equation can be written as follows:

$$\frac{\partial}{\partial t} \rho + \frac{\partial}{\partial x_j} \rho U_j = 0$$

Conservation of momentum in an inertial reference frame is described by Equation 2:

$$\frac{\partial}{\partial t} \rho U_i + \frac{\partial}{\partial x_j} \left( \rho U_i U_j + P \delta_{ij} \right) = \frac{\partial}{\partial x_j} \sigma_{ij}$$

II.1 The k-ε turbulence model:

The simplest "complete models" of turbulence are two equation models in which the solution of two separate transport equations allows the turbulent velocity and length scales to be independently determined. The standard k-ε model falls within this class of turbulence model and has become the workhorse of practical engineering flow calculations in the time since it was proposed by Launder and Spalding (1974). Robustness, economy and reasonable accuracy for a wide range of turbulent flows explain its popularity in industrial flow and heat transfer simulations. It is a semi-empirical model and the derivation of the model equations relies on phenomenological considerations and empiricism. The modeled transport equations for k and ε in the standard k-ε model are:

$$\frac{\partial}{\partial x_j} \left( \rho U_j \frac{\partial k}{\partial x_j} \right) = - \frac{\partial}{\partial x_j} \left[ \left( \mu + \mu_t \right) \frac{\partial k}{\partial x_j} \right] + P_k - \rho \varepsilon$$

$$\frac{\partial}{\partial x_j} \left( \rho U_j \frac{\partial \varepsilon}{\partial x_j} \right) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \mu_t \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \frac{C_{1\varepsilon}}{k} P_k - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k}$$

The constants of the k-ε model are: $C_{1\varepsilon} = 1.44$, $C_{2\varepsilon} = 1.92$, $\sigma_k = 1.0$, $\sigma_\varepsilon = 1.3$.

III. NUMERICAL METHOD

In this study, the Fluent numerical code was used for simulation. Values for Reynolds number of flow and free stream velocity were $1 \times 10^7$, and the used fluid was air. The airfoil considered in this study is NACA4415 airfoil. The airfoil geometry, channel positions and dimensions are shown in figure 1 and 2 respectively. The chord length of the airfoil is 1m.
A computational fluid dynamics CFD is done to simulate the steady, Transonic, compressible, 2-Dimensional turbulent flow over a NACA-4415 airfoil. At first the CFD analysis is done to the clean airfoil without modification for different angles of attack (0°, 5°, 10° et 15°), thereafter based on the results obtained the angle 15° “where the boundary layer separation is evident” is chosen to apply the geometric modification.

The commercial RANS based code FLUENT was used for detailed calculations. The k-ε fully turbulent model that is offered by FLUENT was used for detailed calculations. In the FLUENT simulations, second order upwind discretization in space is used, and the resulting system of equations is then solved until convergence.

A C-type structured mesh used for the single airfoil is generated by the GAMBIT program is shown in Figure 3. The grid extends from -11.50 chords upstream to 21.00 chords downstream and the upper and lower boundary extends 12.50 chords from the profile. Different size grids are used to ensure grid independence of the calculated results. This is achieved by obtaining solutions with increasing number of grid nodes until a stage is reached where the solution exhibits negligible change with further increase in the number of nodes. Consequently, the grid size giving the grid independent results are selected and the total number of cells is adopted as 264,000 nodes.

The solutions in all cases, continued until lift and drag coefficient fully converged
Figure 3. Structured grid of single NACA 4415 airfoil for CFD analysis.

IV. RESULTS AND DISCUSSION

Figure 4. Velocity and Mach-number contour at the angle of attack 0°.

Figure 5. Pressure and pressure coefficient contour at the angle of attack 0°.
Figure (4) shows the velocity and Mach number contour around the NACA 4415 for the angle 0°. At the entrance velocity is uniform and equal to 146.13 m/s as it was imposed as condition at the domain limit. The leading edge velocity tends to zero. However, advancing to either side of the leading edge, we find a remarkable speed acceleration on the upper surface vis-à-vis the lower surface (especially on the first part of the upper surface. This is due to the strong favorable pressure gradient). A little further down the profile, the boundary layer thickens more quickly because it operates in a pressure gradient became unfavorable (in this region). Downstream of the profile, we get the wake phenomenon that is not yet enormous. The discrepancy in momentum is clearly observed at the trailing edge.

Figure (5) shows the pressure contours in the domain of study for the angle 0°. At the leading edge of the profile the pressure is at the maximum. This result is consistent with that of the velocity field; all the kinetic energy is converted into pressure energy. Advancing on both sides of the leading edge along the wall of the profile, a decrease in pressure is noticed greater on the upper surface than the lower surface in consistency with strong acceleration discussed above. A pressure increase is also observed at the trailing edge, although less than in the leading edge, so that the pressure gradient is negative along the second part of the profile.

Figure 6. Stream lines around the airfoil at the angle of attack 0°.

The observation of the Stream lines around the airfoil for the angle 0° (Figure 6) shows that there is no flow Separation (no velocity in the reverse direction) despite the adverse pressure gradient on the upper surface in the area near the trailing edge. As the Stream lines remains parallel. So the separation of the boundary layer will not take place for the angle 0° to confirm this result, the variation of the local skin friction coefficient is studied:

Figure 7. Skin friction coefficient variation at the angle of attack 0°.
In the figure (7) representing the coefficient of skin friction around the profile, it is noted that the friction coefficient is minimum in two positions; the first at the leading edge and the second at the trailing edge. Immediately downstream of the leading edge the skin friction coefficient on the lower surface is higher than that on the upper surface, but advancing on both sides of the leading edge along the wall of the profile the phenomenon is inverted that is to say that the coefficient of friction on the upper surface is higher than on the lower surface, as its value is controlled by the velocity gradient at the wall. It is clear that the friction coefficient does not vanish to 0 along the airfoil walls (except the at the leading edge point). Therefore there is no boundary layer Separation along the airfoil.

For the case of angles of attack = 5 ° and 10 ° the same procedure with the angle 0 ° was executed. The separation was less important.

Figure 8. Velocity and Mach-number contour at the angle of attack 15°.

Figure 9. Pressure and pressure coefficient contour at the angle of attack 15°.

In Figure (8) that shows the velocity magnitude contour and the Mach number contour for the angle of attack 15 °, it is observed that the fluid at very low speed or even zero or on the opposite direction to the flow imposed has colonized more than half of the upper surface.

Figures (10) representing the velocity vector fields, recirculation is observed near the trailing edge, and the disappearance of a preferred direction of advection. The emergence of this strong detachment was irretrievably accompanied by a rebalancing of pressure between the upper and lower surfaces in consistency with the results discussed above. Eddies with vast
dimensions appear on the upper surface near the trailing edge. These vortices have a detrimental effect on the performance of the airfoil because they disturb the flow completely and they are the main cause of an inevitable stall.

![Streamlines around the airfoil at angle of attack 15°](image1)

**Figure 10.** Stream lines around the airfoil at the angle of attack 15°.

![Velocity and Mach-number contour at angle of attack 15° modified](image2)

**Figure 11.** Velocity and Mach-number contour at the angle of attack 15° modified.

From the figure of skin friction coefficient (11) we can estimate that the detachment point position is located at: 0.49 m of the leading edge.

In the figures (12) we can notice that the accelerated fluid zone is located between the leading edge and the inlet of the canal. The fluid at very low speed, builds a small layer above the orifice in the opposite direction to the flow imposed colonizing the portion of the upper
surface after the orifice. The high pressure zone is a little narrower than the uncontrolled case, and the depression of the extrados is located between the leading edge and the inlet of the canal.

![Stream lines around the airfoil at the angle of attack 15° modified.](image)

The most important phenomena observed for the modified case are:

- Despite the modification done to the airfoil the recirculation near the trailing edge, and the disappearance of a preferred direction of advection are observed.
- Aspiration of the return flow is to say that the very low velocity fluid which is in the opposite direction to the flow imposed flows through the canal. While the flow on the upper surface above the canal becomes totally in the direction of flow imposed.
- Eddies that appears on the upper surface near the trailing edge are smaller than the non modified case without opening.
- The lift to drag ratio is improved by 7.2% for the modified case.

V. CONCLUSION

In this work, we presented the numerical simulation results of Boundary layer separation control via geometric modification in form of a channel goes from the leading edge to the trailing edge on the upper surface on a NACA 4412 airfoil at a Reynolds number of $1\times10^7$ and an angle of attack of 15 deg. where change in aerodynamic characteristics of the airfoil by applying this geometric modification is described.

The results proved the feasibility of this modification to improve aerodynamics performances. The boundary layer separation is reduced and the total aerodynamic efficiency of the airfoil is improved by applying this idea.

The concept is very new and, it could be extremely beneficial in making an aircraft more maneuverable by changing flow characteristics. Also it increases the aerodynamic efficiency and therefore helps in improving the performance also. The idea will also assist in shorter take-offs at low speed.

VI. REFERENCES


