FLOW SEPARATION CONTROL FOR THE IMPROVEMENT OF THE LEFT OF AIRFOIL USING RIBLETS

DR. EMADSHUKRI WINDI*

*College of Engineering, University of Babylon, Iraq

ABSTRACT

This paper deals with the experimental and numerical simulation of the two dimensional, incompressible, steady air flow past an airfoil with riblets. The analysis involved the airfoil’s aerodynamic performance which meant obtaining velocity, pressure distribution and lift coefficient curves as a function of the angle of attack for the condition of adding riblets to the surface. The results size chosen for this study is (h=0.5 mm and h=1mm, S=1mm, and W=1mm, for α =0, 5°, 10°, 15°, 20°). The computational work has been performed by Comsol 3.5 software using k-ε model to describe the turbulent flow process. The simulations were held at a Reynolds number of (Re_c=7.4 x10^6). Results show improvement in lift and drag forces and pressure coefficient and also the location of the separation and reattachment using riblets as well as velocity contours and streamlines for both conditions at different angles of attack. Analysis the characteristics of the airfoil after modification shows that it improves significantly the lift of the airfoil reaching values up to 27% when compared to the base surface without riblets.

INTRODUCTION

In recent years there has been an increased interest in the flow control field, especially in aerodynamics, with the purpose of increasing lift and decreasing drag of airfoils. Wings suffer from flow separation at high angles of attack due to viscous effects, which in turn causes a major decrease in lift and increase in drag.

Flow control involves passive or active devices; passive control devices are those, which are not energy consumptive. They mainly affect the flow by the geometry of the airfoil. In contrast, active control devices use energy such as surface suction or injection, to effect a beneficial change in wall bounded or free-shear flows. Whether the task is to delay/advance transition, to suppress/enhance turbulence or to prevent/provoke separation, useful end results include drag reduction, lift enhancement, mixing augmentation and flow-induced noise suppression [12].
Flow separation occurs when the boundary layer travels 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. In aerodynamics, flow separation is mostly an undesirable phenomenon and boundary layer control is an important technique for flow separation problems on airfoils and in diffusers. Flow separation can often result in increased drag, particularly pressure drag which is caused by the pressure differential between the front and rear surfaces of the object as it travels through the fluid.

Drag reduction is one of the basic scientific and technological issues for large transport airplane development. Within the airplane’s cruising drag, friction drag is an important component, especially for subsonic airplanes, with surface friction drag accounting for almost 50% of the total drag [2]. Transition-delaying compliant coatings were rationally optimized using computational fluid dynamics [3]. By controlling the flow, the fuel burned might be decreased almost 30 percent as reported by Braslow (1999) [4]. As a result, the pollutant emissions are reduced. In addition, lower fuel consumption will reduce the operating costs of commercial airplanes at least 8% (Braslow, A.L., 1999) [4].

Research on drag reduction methodologies relevant to flight vehicles has received considerable attention during the past 2–3 decades. Riblets, or wall grooves manufactured in a surface, are attractive drag-reducing devices because of their low production cost and easiness of maintenance. They can be successfully installed on wings of an aircraft, hull of a submarine or internal walls of a gas pipeline by adding special plastic films with sub-millimeter scale riblets which are available commercially[5]. There has been continuous and focussed activity around the globe concerning development of new techniques for skin friction drag reduction and attempts have progressed broadly in two directions: methods for delaying laminar-turbulent boundary layer transition and methods for altering or modifying the turbulent structure of a turbulent boundary layer.[1].

Choi [6] investigated the effects of longitudinal pressure gradients on a flat plate with machined riblets (1.5 mm high and 2.5 mm pitch) for two values of pressure gradient parameter of 3.1 and 0.16 at low speeds; the emphasis in the study was on the structure of near-wall turbulence but not on drag reduction. Based on measurements of mean velocity, stream wise turbulence intensity, wall shear stress fluctuations, he suggested that the effectiveness of riblets in reducing skin friction may remain under pressure gradients. The experiments carried out by Pulvin and Truong [7] in a channel flow showed maximum
The purpose of this investigation is to study the effect of streamwise riblets on the aerodynamics characteristics and on controlling the flow over a specific airfoil. The case studied is the flow field over a subsonic airfoil with and without riblets for different angles of attacks. The investigation is accomplished experimentally and numerically in order to study the effect of Riblets as a flow separation control and its ability to postponed flow separation on the airfoil and the percent of drag reduction achieved.

**EXPERIMENTAL WORK**

In the experimental work, the lift and drag coefficient is being investigated by installing a vertical airfoil section inside the wind tunnel. A wind tunnel is essentially a large Venturi with air driven by a turbine fan. The wing section is located at the throat of the Venturi. Since the fan is downstream from the Venturi, and the airstream is settled before the throat, little turbulence exists before the airfoil. An overview of the experimental system is shown in Figure 1.

![Figure 1 Experimental System Overview](image)

A wing section in an air stream develops lift, because the pressure on one side of the section can be made different from that on the other side. The lift coefficient $C_L$, defined by [11]:

\[
C_L = \frac{FORCE_{Lift}}{\frac{1}{2} \rho \cdot u^2 \cdot AREA_{WingPlane}} \tag{1}
\]

And the drag coefficient is given by
Drag and lift depends strongly on the angle of attack $\alpha$ between the mean chord line and the direction of the airflow. The direction of the lift force is always perpendicular to the direction of the airflow. The force component parallel to the airflow is called drag. The sum of the force vectors is the total force. Small rectangle strips tapes have been installed on the surface of the NASA 1412 airfoil with a dimension, of height $h=0.5$ and 1mm, spacing $S=2$ mm, and width $w=2$ mm, as shown in fig. 2 and 3. The airfoil contains 11 pressure taps. Pressure tap #11 is at the leading edge, while No.1 is located downstream of the airfoil. The chord length of the section is 4 inches while the width is 9 inches and maximum thickness is 0.5 inch. The air speed is set at 50 mph (Re=7.4 x10$^6$) and the angle of attack set to be to 0°, 5°, 10°, 15°, and 20°.

The pressure can be measured using The DSA3200 series pressure acquisition systems which represent the next generation of multi-point electronic pressure scanning contains 2-16 single sensing element pressure transducers which is connect to the computer for capturing the pressure data from the DSA. Then Pressure measured $\Delta P = \rho_m g \Delta H$ where

$\rho_m$ =density of manometer bank fluid

$g$ =acceleration due to gravity (m/s$^2$);

$\Delta H$ =difference between each pressure tap reading and upstream tap reading (m);

$\Delta P$ = difference in pressure between the tap and upstream (Pa);

\[ C_D = \frac{\text{FORCE}_{\text{drag}}}{\frac{1}{2} \rho_u u^2 \times \text{AREA}_{\text{WingPlane}}} \]  

(2)

NUMERICAL SOLUTION

In order to verify the proposed idea, an airfoil profile needed to be selected on which whole study will be based. This is a conceptual study which assumes an Incompressible and
isothermal flow. All the simulations are carried out on a symmetrical airfoil NACA 1412 (chord length-4 inch)[10 ].

**k-ԑ turbulence model** is used for 3-D as well as 2-D simulations of airfoil segment in COMSOL fluid flow module to predict the flow regime. The k-ԑ model is one of the most common turbulence models. It is a two equation model - that means, it includes two extra transport equations to represent the turbulent properties of the flow. The followings

Equations used to solve N_S equations:

The continuity equation given as

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho V_x)}{\partial x} + \frac{\partial (\rho V_y)}{\partial y} = 0$$ ...................................................(3)

The momentum equation in x direction given as

$$\frac{\partial (\rho V_x)}{\partial t} + \frac{\partial (\rho V_x V_x)}{\partial x} + \frac{\partial (\rho v_y V_x)}{\partial y} = - \frac{\partial P}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{yx}}{\partial x}$$ ..........................................(4)

Similarly, the momentum equation in y directions can be expressed as:

$$\frac{\partial (\rho V_y)}{\partial t} + \frac{\partial (\rho V_x V_y)}{\partial x} + \frac{\partial (\rho v_y V_y)}{\partial y} = - \frac{\partial P}{\partial y} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{yx}}{\partial x}$$ ........................................(5)

V_x, V_y = component of the velocity vector in the x, and y direction.  ρ = density

μ_e = effective dynamic viscosity

g_x, g_y = component of acceleration due to gravity.

For the turbulent case the effective viscosity:

$$\mu_e = \mu + \mu_t$$ ..........................................................(6)

where μ: laminar viscosity or dynamic viscosity

μ_t: turbulent viscosity

The two equations (k-ԑ) turbulent model is used to evaluate the turbulent viscosity through the expression

$$\mu_t = C \mu \rho \frac{k^2}{\epsilon}$$ ..................................................(7)

where k: turbulent kinetic energy

ε: turbulent kinetic energy dissipation rate

Triangular meshing is used for the analysis. Each model is first simulated at coarse meshing. After every converged solution the meshing is refined and again the model is simulated. Fig (4 ) shows the meshing around the airfoil while fig.(5) shows the 3 –dimensional model inside the wind tunnel .
RESULTS AND DISCUSSION

In Figure 6 (a, b, c) shows the flow over the airfoil without riblets, it is possible to observe the flow field as velocity distribution, velocity profile and contours of the air flow past the airfoil. Here it is observable the velocity changes in the selected domain; in our case the most important is to observe this phenomenon near the surface of the model. However these pictures do not show clearly the separation and reattachment points, while figure d shows the pressure distribution over the airfoil.

Figures for 7, 8, & 9 represent the pressure and velocity distribution for the airfoil with riblets (h=1mm, S=2mm, and W=2mm.) at different angles of attack ($\alpha = 5^\circ$, $10^\circ$, and $15^\circ$), from the pressure figure it is clear that adding the riblets create a high pressure differences between the upper and lower surface which enhanced the lift coefficient.

The experimental effect of the Riblets on airfoil pressure distribution at various angles of attack is shown in Fig. 10, 11, and 12. The Riblets (h=0.5 mm and h=1mm ,S=2 mm, and W =2 mm, $\alpha =10^\circ$ ) shows increases the pressure difference between the upper and lower surfaces, as shown by the area between the two curves particularly in the vicinity of the trailing edge. This leads to increased lift and reduction in the drag. The area between the two curves for h=1 mm is higher than that for h-0.5mm.

Fig.13 shows the experimental results for the airfoil with different riblets height. In general, the lift coefficient increases as the riblets height increases for a given angle of attack. As an example, for $\alpha=15^\circ$ increasing the riblets height to 0.5 mm cause to the lift to increase by 18% while for height of 1 mm increase by 27%, which shows a positive results for using the riblets on airfoils.

Fig. 14 shows comparisons between experimental and computational work for the lift coefficient. While there is general agreement with the experimental data, there are differences between the two results. The computed flow field has more separation than the experiment for ($\alpha=17^\circ$) which may account for the drag getting higher at this angle.
CONCLUSIONS

Addition of Reblits has proven to be effective in altering various aspects of the flow structure. With such significant flow structure the resultant lift and drag forces are also improved. Primarily the Reblits with h=1mm are proved to be most suitable as proved by the results of this study. Based on these results adding Reblits over airfoils which will sense boundary layer separation and results in least drag and high lift configuration.

The concept is extremely beneficial in making an aircraft more manoeuvrable 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.

REFERENCES

3. Caram JM, Ahmed A. Effects of riblets on turbulence in the wake of an airfoil. AIAA
7. flight research. American Institute of Aeronautics and Astronautics, Washington,
c/ Velocity Contour
d/ Pressure distribution
Fig. (6) Numerical results for zero angle of attack without riblets

Fig. (7) pressure and velocity distribution with Riblets (h=1mm) & $\alpha = 5$

Fig. (8) pressure and velocity distribution with Riblets (h=1mm) & $\alpha = 10$

Fig. (9) pressure and velocity distribution with Riblets (h=1mm) & $\alpha = 15$
Fig. 10) Distribution of pressure coefficient with no riblets.

Fig. 11) Distribution of pressure coefficient with Riblets (h=0.5 mm).

Fig. 12) Distribution of pressure coefficient with Riblets (h=1 mm).

Fig. 13) Coefficient of lift for different angle of attack.

Fig. 14) Comparison between Experimental and Theoretical work.