ON THE DRAG REDUCTION OF THE FLOW OVER AN AIRFOIL

ICAS-94-4.1.2

By

Dr. T. M. Maksoud and A. M. Al-Shihry

Department Of Mechanical And Manufacturing Engineering

University Of Glamorgan, UK

Abstract

Preliminary experiments have shown a potential for improvements in skin friction reduction. This was made by introducing a special form grooves on top of the wing section in the span wise direction. These grooves provide a very small vortices underneath the boundary layer and consequently reduce the skin friction through creation of an 'air bearing' layer. It is also believed that it helps in energizing the flow. By simulating the flow pattern and the three dimensional turbulent field numerically, a prediction of the effect of different forms on the drag can be made. It is also instigated that a reverse air suction will be used to further enforce the existence of the air vortices. A set of experimentation are in progress to further develop the suction technique. So far the new development also has shown a potential for the delaying of the onset of stall region. This would have a scope for improving the maneuverability of the aircraft.

Nomenclature

C_d = Drag coefficient

C_f = local skin-friction coefficient

C_l = lift coefficient

C_q = suction coefficient $= -\frac{|V_w|}{U_0}$

D_r = percent of drag reduced

U = mean velocity component

U_0 = velocity outside the boundary layer

S = distance between the grooves

x = stream wise distance from the leading edge

y = normal distance from the wall.

α = angle of attack

δ = boundary-layer thickness

δ_θ = momentum thickness

V_ω = normal velocity of fluid injected or withdrawn through the wall

Introduction

Over the last two decades, extensive researches were carried out into the reduction of drag over the hydrodynamic and aerodynamic devices by modifying the geometry of the surface. Roughening and shaping are among the simplest passive methods to ensure flow attachment beyond a critical conditions and, thus, improved performance[1]. Longitudinal riblets on the surface were investigated and recognized as one simple way of achieving a reduction, however small, in the friction drag. The ideal geometrical characteristics and operating conditions necessary to obtain maximum performance are now reasonably assessed, due to a large amount of experimental work carried out on turbulent boundary layers and channel flows over ribbed surfaces[2]. On the other hand, the theoretical answer to the question 'How riblets do actually work in reducing the friction drag?' constitutes the main goal in many recent researches [2,3,4].

Among the ways of explanation of the mechanism of the drag reduction near grooved surfaces is the one which suggested that the corrugations interfere with the secondary cross-flow associated with the longitudinal vortices which randomly appear in the turbulent flow, and some how manage to dampen these vortices and therefore the level of the turbulence itself; the consequent reduction in the rate of turbulent diffusion makes for a lower eddy viscosity and the reduction of drag[3,4]. The *rms* turbulence intensity near the surface is reduced, but burst frequency is not; while the wall streaks tend to align themselves over the riblets[5]. However, the studies of using the rough surfaces in reducing the drag were begun long time ago. Mulhearn[6] used a square grooves of 3.2mm and spaced at 12.7mm. He reported that the Reynolds stresses in the turbulent flow near the wall were reduced; but the using of rough surfaces did not prove a good results in reducing drag. The difficulty of simulating the turbulent flows in complex geometry's is the reason behind the lake in full understanding of the nature of flow between and inside riblets; and because of that most of research are related to the global measurements of the effect of riblets in boundary layer instead of analyzing the flow field in the vicinity of and inside the grooves. Recently; Baron and

Quadrio[2] reported that there is no general agreement between researches in understanding the theoretical reason for the drag reduction obtained from using riblets.

One can divide the researches carried out on this type of longitudinal riblets to four types; (*a*) riblets on a flat plate that was introduced by Walsh[7]. Many extensive research works were made in optimizing the geometry of the riblets and measuring their effects in reducing the drag; see for example Walsh[7] and Walsh[8], and Mulhearn[5]. (*b*) Riblets over an axisymmetric and curved bodies. Konovalov et.al[9] tested the riblets over a 2.626 m long axisymmetric body and reported a maximum drag reduction of 8%. Lashkov et. al.[10] applied the riblets over a curved surface and reported some improvement in reducing the turbulent friction. (*c*) Riblets are tested as well with an internal flows and gave a drag reduction up to 6%[2]. (*d*) Riblets used over an airfoil[11], and used over a wing section in real flight (see Walsh[8]) where the drag was found to be reduced by an amount up to 6%. A triangle-shaped riblets suggested by Walsh[7] were found to give the best performance and consequently manufactured on a sheet that can be stuck on the top surface and recently was used by many researches.

In all of these research works, the riblets used were longitudinal, i.e., along with the flow direction. On the other hand, the cross-flow riblets were studied in fewer numbers of researches.

In the present work, an investigation of the effect of the cross flow riblets over an airfoil is studied. The way in which the longitudinal riblets affect the flow and hence reduce the drag is thought to be different when using span wise riblets. Instead of 'destroying' the rolling stream wise vortices and cutting the link between them and the longitudinal ones; this technique is thought to make use of those vortices, but in this time these vortices will be 'kept' inside the span wise grooves at the surface. The geometry of the grooves should provide the flow with an ability to create and maintain the vortices inside them. The geometry of the grooves is chosen after a preliminary results of numerical studies on many shapes over the flat plate to obtain a net drag reduction.

Since the air suction through the top surface is thought to have special effect in this technique; the effect of air suction used along with the stream wise grooves is studied as well.

Theoretical Study

The simulation of 2D turbulent flow over a grooved flat plate was made using finite difference computer code to solve the Navier Stokes equations. The turbulence model used was κ-ε model since the flow is not assumed to have any high swirling and there is no high streamline curvature. To have a good representation of the flow field inside the transverse grooves, a body fitted coordinates were used where the grid mesh is condensed near the wall and inside the grooves. This enables one to show the flow pattern inside the grooves in which are thought to have a small rolling vortices. These vortices are thought to give an effect similar to that of roll bearing. This is why this technique is thought to produce a so called 'Air bearing' underneath the viscous sub layer.

In drag reducing flows with this kind of techniques, the accurate measurements of the coefficient of local skin friction, C_f, is difficult. The measurements of $dU/dy|_{y=0}$ is highly desirable[12] .Therefore the profile $dU/dy|_{y=0}$ is plotted for smooth and grooved surfaces, where the increase on $dU/dy|_{y=0}$ indicates a reduction of the height of the viscous layer and, consequently, a reduction on the total skin friction. The problem arises here is that at what point should the value of U is to be taken to show $dU/dy|_{y=0}$ since the height of the grooves is relatively large compared to the boundary layer thickness. In the present work, however, the values of U were taken at a line parallel to the wall over the groove peaks by a distance of $\Delta y=O(0.1\delta)$.

Since it is not a part of the scope of the present work to study the effect of transverse grooves in a separated flows- i.e. backward flows; the flow used in both the numerical and experimental studies was of Reynolds numbers based on cord length between $1\text{x}10^5$ to $3\text{x}10^5$. This range of air flow -when applied to an airfoil- will be expected to have no separation or very small separation bubble followed by reattachment[1]. This insures that the desired effect of the grooves over the surface will exist.

Experimental work

The measurements were carried out in a high speed wind tunnel of a 0.3X0.3m test section. The free stream velocity in this study is ranged from 25 to 36 m/s. Three 2D (NACA0012) with symmetric cross section airfoil models were used in this study. They were manufactured from hardwood and have a good surface finish. The first one was the smooth one, i.e. no alteration was made to its geometry; while the second one has a grooved upper surface; and the third one was grooved with a suction holes distributed at the top surface of the airfoil. The Reynolds numbers resulted from these data based on cord length were to stay within the range of $1\sim3.0\text{x}10^5$which complies with what already discussed in the previous section.

An electronic drag balance system was designed using a very sensitive, well calibrated load cells. The system can give -directly- the value of the drag force and lift force existing on the airfoil at wide range of angle of attack. The tests were made over a wide range of angle of attack.

Suction Modeling

The main benefit which can be obtained from using suction technique is either to control the flow separation, i.e. to delay the separation far from the leading edge, or to inhibit the growth of the boundary layer so that the critical Reynolds numbers based on thickness may never be reached and thus maintaining laminar flow. On the other hand the inhibiting of the growth of boundary layer yields to reduction in pressure (form) drag and therefore the overall drag of the body. For a flat plate with zero pressure gradient, and a uniform suction through the wall, the skin friction coefficient can be written as follows[1]

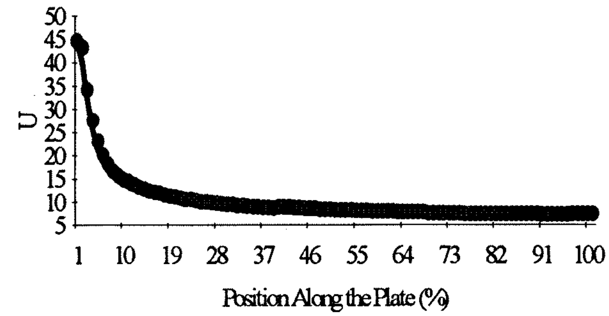
$$\frac{C_f}{2} = \frac{d\delta_\theta}{dx} - C_q \qquad (1)$$

In this equation, although withdrawing the fluid through the wall leads to a decrease in the rate of growth of the momentum thickness Cf increases directly with the suction coefficient Cq. This emphasizes the importance of choosing the suction flow rate to balance between the increase on the second term with the decrease in the first one.

The suction used in the literature was used from only one point of view; i.e. to have a 'scalar' net mass withdrawn from the wall. In the present work, an additional goal is tried to be obtained. This goal is to get benefit from the 'direction' of the fluid sucked through the wall in such a way to accelerate the vortices inside the grooves itself. This goal may be accomplished by carefully choosing the position of the suction holes or slots inside the grooves. This is thought to energies the vortices created inside the grooves provided that the suction flow rate is not large to the extent that all the flow enter the groove is sucked and hence no vortices are generated.

Discussion of Results

As already mentioned in a pervious section, the velocity profile was used as a measure of the change in the drag over the surface. The comparison of the resulted mean velocity profiles obtained from computational experiments over a smooth and grooved airfoil are shown in Fig[1]. It shows evidence of a small increase of U/U_0 over the grooved surface. This shift on the velocity profile is a peculiar phenomenon for drag-reducing flows as clarified by Baron and Quadrio[2].



(• *half circle*, □ *square*, Δ *tringle*, — *smooth*)

Figure 1 - Skin Friction over a flat plate of different groove Shapes

In Fig.[1] the velocity profiles are plotted for three different surface-grooves as well as for the smooth one. In these cases the shapes of the grooves were changed while the sizes are kept the same. Comparing the half-circle groove with others show that it gives better performance. This may be referred to the fact that the rolling vortices inside the half-circle groove keeps more of its momentum during the circulation than in the case of rectangular and triangular ones were the separation bubbles are formed at the corners and may cause loss of momentum. However, the improvement in the velocity profiles comparing with the smooth one may support the idea of creating an 'air bearings' inside the grooves.

In an attempt to enhance the conservation of the momentum of the fluid entering the groove, it is suggested that all the separation bubbles inside the groove should be either eliminated or kept at small, in size, as possible. This includes the separation bubbles which may be formed at the groove-inlet tip.

The idea of creating an air bearing underneath the viscous layer of the fluid can be further enhanced using the suction technique as discussed in the last section.

It is worth mentioning that the choice of the spacing between the grooves S plays an important role in determining the amount of benefit that can be gained from this technique. Theoretically, the ideal geometry should have $S \rightarrow 0$ to ensure that the air 'fly's' over the vortices (air bearings) inside the grooves without touching the surface of the body; but practically this may not be possible. The reason is that the existing of a spacing s with finite value indicates the angle at which the air enters the next groove. This is a very important factor in determination of the characteristics of the formed vortices.

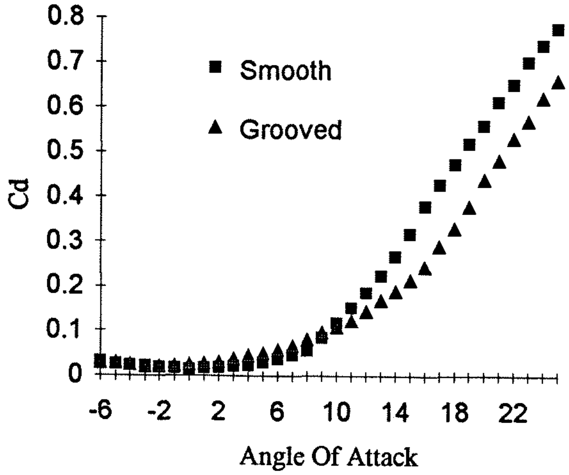


Figure 2 - Drag Coefficient over an Airfoil

As this is a part of an ongoing research on its subject, at the current stage of work only one value of suction flow rate was used and only one geometry of grooves were used throughout the experiments presented here. The suction experiments were made in many arrangements (1) the suction was applied to all of the upper surface of the grooved airfoil. (2) The suction was applied to a line of grooves (while keeping the rest of the holes closed). The suction holes was tested without suction (i.e. suction the trailing edge of the airfoil. (3) The grooved airfoil with location of this line was changed from the front edge to flow rate =0). The main idea behind the selection of these many arrangements was to come up with a comparable results.

In Fig.[2] the drag coefficient is found to be smaller for the grooved airfoil than that for the smooth one. The reduction on the drag starts after $\alpha = 9°$ to reach an amount of D_r= 18%; where :

$$D_r = \frac{C\,d\big|_{smooth} - C\,d\big|_{grooved}}{C\,d\big|_{smooth\ at\ stall\ position}} \times 100\% \qquad (2)$$

while in the range of $\alpha<9°$ the drag is slightly increased to about $Dr = 3\%$. As an attempt to explain the increase of the drag below 9°; the existence of the grooves at the leading edge of the airfoil may disturb the flow at this critical region causing it to be turbulent at a very early stage. From the figures it is clear that at higher angle of attack the drag decreases; this may be considered as an indication of less distortion caused by the front grooves. The reason for that may be at higher angle of attack the stagnation point goes down the leading edge , and hence gives the flow enough opportunity to *'relax'* on the top surface and then to create the vortices starting from the front grooves. However, the phenomenon of increasing the drag at the range of $\alpha<9°$ and the effect of the onset of the front grooves should be more understood if the grooves are started from a certain distance behind the leading edge. This subject is taken into consideration in an ongoing work. On the other hand, the curves in Fig.[3] show a slight decrease in the lift of the grooved airfoil for the range of $\alpha<12°$ by about 3%; where the lift is recovered and increased to about 13% after that region.

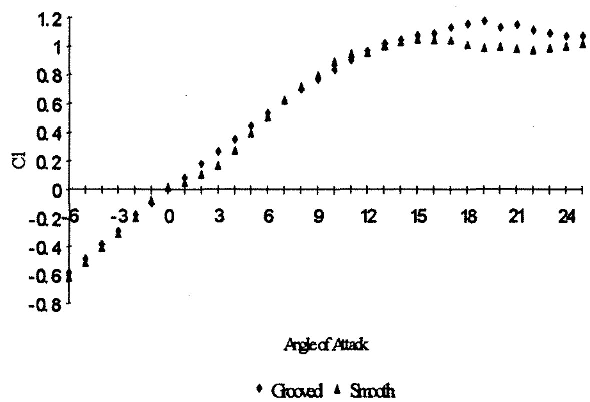


Figure 3 - Lift Coefficient over an Airfoil

The problem of decreasing the lift may be caused by the distortion of the flow at early stage as already discussed. The lift is hopefully be enhanced when the drag-increasing problem is solved. The lift curve shows a very important advantage for this technique; this advantage is that the stall region is extended by about 5°. This shows the usefulness of using transverse grooves not only to enhance the lift and to reduce drag but also to increase the maneuverability feature of the wing.

Although the application of the suction at the top surface as discussed above does not give a considerable net drag reduction, the limiting of the suction holes at the aft side of the airfoil is found to give less drag than spreading the holes at wide area of the top surface.

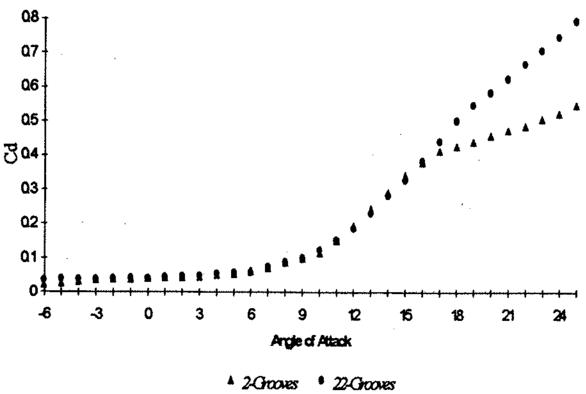


Figure 4 - Drag coefficient over an airfoil for different suction arrangements

However, due to the time limitation, the *directing* of the suction through a certain angle to enhance the momentum of vortices inside the grooves is not fully completed, but the theoretical results give promises in the positive effect of the application of the 'directed' suction in drag reduction. A further tuning of the process is currently being carried out.

Conclusion

The use of span wise riblets over an airfoil shows an evidence of reducing the drag and increasing the lift after a certain angle of attack. The stalling point of the airfoil is extended beyond that of the smooth airfoil by more than 5°. This technique used along with a directed suction arrangements gives a good potential for enhancing the performance of the airfoils and, consequently, reduce the energy loss accomplished when using them in aerodynamic devices.

Acknowledgment

The authors would like to acknowledge University of Glamorgan for the support provided by them in this research program.

References

[1] M. Gad-El-Hak, "Control of Low- Speed Airfoil Aerodynamics, *AIAA Journal* Vol. 28,1990, No. 9,p 1537-52.

[2] A. Baron and M. Quadrio; "Some Preliminary Results on the Influence of Riblets on the Structure of a Turbulent Boundary Layer"; *Int. Journal Heat and Fluid Flow*, Vol. 14, No. 3, Sep 1993.

[3] P. Prluchini, and F. Manzo, and A. Pozzi; "Resistance of a grooved Surface to Parallel Flow and Cross-Flow"; *Journal of Fluid Mech.* Vol. 228, 1990, pp. 87-109.

[4] D. W. Bechert, and G. Hopee; "Turbulent Drag Reduction by Non planar Surfaces - A Survey on the Research at TU/DLR Berlin"; Structure of Turbulence and Drag Reduction, IUTAM Symp. Zurich/Switzerland 1989.

[5] D. M. Bushnell, "Turbulent Drag Reduction for External Flows"; AIAA paper 83-0227, 1983.

[6] P. J. Mulhearn; "Turbulent Flow Over a Periodic Rough Surface"; *Phys. Fluids*, Vol. 21, No. 1, July 1987.

[7] Walsh, M. J. and Weinstein, M., "Drag and Heat Transfer on Surfaces with Small Longitudinal Fins," AIAA paper 78-1161, 1987.

[8] Walsh, M. J., "Riblets"; *Progress in a Astronautics and Aeronautics*, Vol. 123, 1990.

[9] S. F. Konovalov, YU. A. Lashkov, V. V. Mikhailov I. V. Fadeev, and G. K. Shapovalov; "Effect of Longitudinal Riblets on Axisymmetric Body Drag"; Moscow, Translated from Izvestiya Rossiiskoi Akademii Nauk; Mekhanika Zhidkosti i Gaza, No. 1; p. 177-179, January-February, 1992.

[10] YU. A. Lashkov, I. N. Sololova, and Shumikina; "Jet Flow Over Ribbed Curved Surfaces"; Translated from Izvestiya Rossiiskoi Akademii Nauk. Mekhanika Zhidkosti i Gaza, No. 1; p. 174-178, March-April, 1992.

[11] J. M. Caram, and A.. Ahmed; "Development of the Wake of an Airfoil with Riblets."; *AIAA Journal.*, Vol. 30, p2817-18, December' 92.

[12] P. R. Bandyopadhyay; "Review-Mean Flow in Turbulent Boundary Layers Disturbed to Alter Skin Friction"; *Journal of Fluid Engineering*, Vol. 108, 1986, pp.. 127-140.