A Numerical Proof of Concept for Thermal Flow Control

Valeriu Dragan
Computational Fluid Dynamics Department
National Research and Development Institute for Gas Turbines, COMOTI
Bucharest, Romania
drvaleriu@gmail.com, valeriu.dragan@comoti.ro

Abstract—In this paper computational fluid dynamics is used to provide a proof of concept for controlled flow separation using thermal wall interactions with the velocity boundary layer. A 3D case study is presented, using a transition modeling Shear Stress Transport turbulence model. The highly loaded single slot flap airfoil was chosen to be representative for a light aircraft and the flow conditions were modeled after a typical landing speed. In the baseline case, adiabatic walls were considered while in the separation control case, the top surface of the flaps was heated to 500 K. This heating led to flow separation on the flaps and a significant alteration of the flow pattern across all the elements of the wing. The findings indicate that this control method has potential, with implications in both aeronautical as well as sports and civil engineering applications.

Keywords—heat transfer; CFD simulation; flow control; high lift device

I. INTRODUCTION

Controlling flow attachment on aerodynamic surfaces is one of the main engineering subjects of recent years. From aircraft wings [1] or thrust vectoring [2] to both radial [3] or axial [4] flow turbomachinery and even synthetic jets in sports vehicles [5] and buildings [6], having the ability to control boundary layer separation opens new possibilities for design and operation. This paper deals with a theoretical concept for flow control which has been tested, using state of the art CFD methods, with advanced turbulence modeling. In [7], a multitude of traditional flow control methods is described and commented. Thin wall jets have been used for circulation control as well as supercirculation [8]. Additionally, more recent development of piezo-electric [9] actuators, unsteady passive actuation [10] or even plasma actuators [11] have made their way to high technology readiness levels, providing viable alternatives for aerodynamicists in various fields of engineering.

Conventionally, heating was used on aircraft wings to reduce the Reynolds number near the wall of the wing in order to diminish turbulent friction. Indeed the theoretical background this technique is rooted in provides an optimal point in which virtually all flow near the wall is essentially laminar. However, the power consumption associated with the heating as well as the advent of natural laminar airfoils [12] has made this all but useless for airline applications. Smaller aircraft and drones have been fitted with similar technologies—in the same traditional sense. Due to their small characteristic length, UAV propellers have been shown to be positively influenced by surface heating [13]. As far back as the 1980’s, the prospect of using heated airfoils for improved aerodynamic performances was explored [14]. The typical applications rely on the lowering of the Reynolds number through the local increase in fluid viscosity as the result of heating. In this way, the friction coefficient becomes a function of temperature.

However, in the proposed embodiment of this method, a spoiler-like behavior is sought without the need for geometry variations. Such a method could be used for small aerial vehicles for both rolling maneuvers and landing, without the requirement for complex and heavy moving parts, thus increasing reliability and reducing overall weight.

\[
\tau_u = \mu \left. \frac{\partial u}{\partial y} \right|_{y=0} \tag{1}
\]

\[
\mu = \mu_0 \left( \frac{T}{T_0} \right)^{3/2} \frac{T_u + S}{T + S} \tag{2}
\]

\[
Re = \frac{\rho \cdot V \cdot C}{\mu} \tag{3}
\]

A previous study confirmed that, for airfoils with low camber and viscous dominated drag, a careful distribution of wall temperature can lead to a serious reduction in the drag coefficient as well as a marginal increase in lift coefficient [15].

II. COMPUTATIONAL FLUID DYNAMICS CASE SETUP

The current paper uses state the of the art k-omega SST RANS model, with additional equations for compensating the laminar to turbulent transition and curvature correction to obtain a proof of concept case for the use of thermal actuation for boundary layer separation. A highly loaded airfoil was considered, with typical single-slot flaps and gurney type element on the trailing edge. In the baseline case adiabatic no-slip walls were used whereas in the thermal trimming case, the top surface of the flaps was set to a uniform surface temperature of 500 K. Surface roughness was considered null, bearing in mind that the rugosity itself will (in a real life application) promote the mixing of the turbulent boundary
layer and therefore increase the heat transfer. The geometry of the airfoil and flaps was constructed using an Eppler S1223 for the main airfoil and flaps with a 10 mm gap between the two and a combined chord of 300 mm. The setting angle of attack for the main airfoil is 0° and the flaps is set at 40°. This airfoil has been chosen due to its high loading and low drag [16].

Since this is a preliminary study, seeking an additional validation of the theoretical principle behind the separation control, the CFD study was conducted using RANS methods. Having said this, the case presents a 3D domain - albeit with periodic conditions - while the turbulence model was chosen in such a way that transition from laminar to turbulent - as well as relaminarisation could be theoretically captured. Furthermore, grid sensitivity tests were performed on three grids with increasing cell density (i.e. baseline, baseline x2³, baseline x2⁶). The current mesh was the result of the analysis of the trends of the three grids with respect to lift, drag and moment coefficient. Another relevant aspect is that the first cell (nearest to the wall) was kept constant for all four cases, insuring the same y⁺ for every case. The fluid considered was the classical Redlich-Kwong which has a built in wall temperature correlation. Figures 1 and 2 depict the blocking structure, near wall cell distribution and the y⁺ distribution for both presented cases.

Fig. 1. Blocking structure for the adiabatic and heated cases

Fig. 2. The y⁺ distribution for the adiabatic and heated cases

As stated, the turbulence model employed was the transition SST-RC k-omega [17], as implemented in Ansys Fluent v17. As seen, the near wall mesh meets the criteria y⁺ for k-omega models, in particular for the one used. Since the Reynolds number is expected to decrease with temperature, the cell size used in the adiabatic case was considered - and proven - to be sufficiently small to also cover the heat transfer case. The method used to calculate the first cell size was through the equation below [18] where h is the first cell height, ν is the kinematic viscosity and τw is the wall shear stress.

\[ h = \frac{\tau_w y^+}{\nu} \quad (4) \]

One of the key elements of the heat transfer simulation is the interaction between the velocity boundary layer with the thermal boundary layer. Multiple tests have been reported in the literature, concluding that the SST [19] and SA [20], which both model the boundary layer without wall functions, are best suited for the heat transfer [21].

In terms of boundary conditions, the solid surfaces were considered smooth walls, with no slip condition, the sides of the domain were considered periodical and the velocity inlet was set along the Ox axis at a magnitude of 50 m/s with an ISA atmosphere as a reference. The CFL condition was set to a value of one unit, in order to correctly capture the physical phenomena of heat convection through the boundary layer.

III. RESULTS AND DISCUSSIONS

As is the case with all flapped airfoils, the increase in the overall lift coefficient is owed to the change of pressure distribution around the main airfoil as a result of the secondary airfoil (flaps) disturbance in the flow. By shifting the leading edge stagnation point, the flaps makes to airfoil behave as if the angle of attack had been increased, but without the danger of destabilizing the flow on the top of the main foil. The shift of the LE stagnation point has two main components, the first of which being the deceleration on the underside of the foil, the second reason is more subtle and has to do with the interaction with the flow on top of the flaps itself. Since the flow on the top of the flaps acts similarly to a curved wall jet, the entrainment effects on the flow circulating on the top of the main foil contributes to the apparent increase in angle of attack. In the heated version however, this entrainment occurs to a
significantly lesser degree, meaning that its contribution to this apparent AoA increase will be negligible.

Figure 3 provides an interesting behavior (particular to this type of flow control) in which the flaps no longer manages to perturb the flow in the same manner. Notice that, although the underside flow pattern remains virtually the same, the topside flow is significantly different. It is this difference that leads to the differences in lift and drag seen in the figure below. Figures 4-6 shows bar charts of the results.

As shown increased drag is experienced by the stalled flaps due to the thermal trimming is in fact the result of a double influence. Equally the positive drag flaps is increased whereas the negative drag of the main airfoil is decreased. It must be said that the negative drag on the main foil arises due to the pseudo-incidence that the flaps induces. This negative drag is always going to be canceled out and surpassed by the added positive drag of the flaps, even if it is not stalled. Unfortunately, in this case, the negative drag component on the main airfoil is inextricably linked to the lift force on the respective section. Therefore, if we pursue the diminishing of the overall lift we will have to accept the diminishing of the negative drag. Hence, the only avenue for further optimization of this arrangement will have to rely on the flaps pressure drag and, to some extent, on the main airfoil skin friction drag - perhaps with the classical heating described in [7].

Further, it can be seen that both the main and flap airfoils lead to significant decreases in their respective lift force, to approximately 70% in the case of the main and 60% for the flaps. This is due primarily to the diminishing of the circulation on the top side of both the airfoil elements as downstream diffusion is significantly decreased hence impeding the acceleration of the upstream flow. The Gurney element also registers an influence, however it is marginal when viewed only on the element itself.

As shown increased drag is experienced by the stalled flaps due to the thermal trimming is in fact the result of a double influence. Equally the positive drag flaps is increased whereas the negative drag of the main airfoil is decreased. It must be said that the negative drag on the main foil arises due to the pseudo-incidence that the flaps induces. This negative drag is always going to be canceled out and surpassed by the added positive drag of the flaps, even if it is not stalled. Unfortunately, in this case, the negative drag component on the main airfoil is inextricably linked to the lift force on the respective section. Therefore, if we pursue the diminishing of the overall lift we will have to accept the diminishing of the negative drag. Hence, the only avenue for further optimization of this arrangement will have to rely on the flaps pressure drag and, to some extent, on the main airfoil skin friction drag - perhaps with the classical heating described in [7].

Further, it can be seen that both the main and flap airfoils lead to significant decreases in their respective lift force, to approximately 70% in the case of the main and 60% for the flaps. This is due primarily to the diminishing of the circulation on the top side of both the airfoil elements as downstream diffusion is significantly decreased hence impeding the acceleration of the upstream flow. The Gurney element also registers an influence, however it is marginal when viewed only on the element itself.

The paper deals with the prospect of using skin thermal influences on the velocity boundary layer in order to induce separation, essentially stalling an airfoil which (under normal circumstances) would have no stall tendencies. In our case, a conventional highly loaded airfoil with single slotted flaps was used as a benchmark. The topside of the flaps, when heated, induced boundary layer separation on the flaps which - in turn - led to the change of the overall characteristics of the entire assembly. Since the Mach number was low, comparable to...
typical landing speeds of civilian aircraft [22], the influence upstream of the flaps is quite visible on the main foil. A breakdown of aerodynamic forces on the components of the assembly revealed that the lift and drag contribution of the flaps on the overall force is only part of the trimming. By looking at the flow pattern around the main airfoil, we can observe that the velocity distribution corresponds to a state where the flaps would be extended at a lower angle. Although the lift has been successfully reduced by using the proposed method, the drag penalty appears to be (at least in part) linked to it. Hence, the main direction in which the drag can be reduced would be to use the minimization of the pressure drag on top of the flaps though the use of conventional curved wall jets [23]. Although both theoretical [24] and empirical efforts [25] have been made to model the boundary layer velocity distribution on such flows, the addition of a wall temperature as a factor will most certainly require further study and modeling.

REFERENCES

www.etasr.com Dragans: A Numerical Proof of Concept for Thermal Flow Control