CFD Investigation of Fluidic Momentum Injection as Alternative to Mechanical High Lift Devices for Boundary Layer Control

Kyle A. Smith
Kungliga Tekniska Högskolan, Stockholm, Sweden

The purpose of this computational fluid dynamics investigation was to consider a method for boundary layer control using air injection on a modern transonic airfoil similar to those found on most commercial airliners. Common industry software titles such as Pointwise for mesh generation, and ANSYS Fluent for the CFD computation were implemented. Coordinates for an experimental transonic airfoil which had been used previously in wind tunnel testing were acquired and used in this analysis. A full alpha sweep, from 0 degrees to stall, was performed on the standard three-element configuration at wind tunnel testing conditions. ISA conditions at sea level was assumed for all simulations. These simulations mirrored previously published wind tunnel results. A series simulations were then performed on the standard configuration with retracted leading edge slat and four different injection points. It was found that although the flight performance could be matched at lower angles of attack, the $C_{L_{\text{max}}}$ and lift at the stall angle of the standard configuration could not be achieved.
Nomenclature

\( C_D \) Coefficient of drag  
\( C_L \) Coefficient of lift  
\( C_{L_{\text{max}}} \) Maximum lifting coefficient  
\( C_m \) Pitching moment coefficient  
\( CG_x \) Center of gravity about the \( x \) axis, m  
\( CG_y \) Center of gravity about the \( y \) axis, m  
\( FMG \) Full multi-grid  
\( Re \) Reynolds Number  
\( RN_c \) Critical Reynolds number  
\( SST \) Shear Stress Transport  
\( TF \) Turbulence factor  
\( I \) Turbulence intensity  
\( TUI \) Text user interface  
\( U_\infty \) Freestream velocity, ms\(^{-1}\)  
\( c \) Chord length, m  
\( d_{\text{gap}} \) Gap length, m  
\( k \) Turbulent kinetic energy  
\( kl \) Laminar kinetic energy  
\( s_{\text{max}} \) Maximum cell wall length, m  
\( s_{\text{min}} \) Minimum cell wall length, m  
\( \bar{s} \) Average cell wall length, m  
\( t \) Maximum thickness, m  
\( v \) Velocity, ms\(^{-1}\)  
\( v_i \) Injection velocity, ms\(^{-1}\)  
\( \alpha \) Angle of attack, Degree  
\( \alpha_{\text{stall}} \) Angle of attack at which stall occurs, Degree  
\( \delta \) Deflection angle, Degree  
\( \delta s \) Boundary decay rate  
\( \epsilon \) Turbulent dissipation rate  
\( \mu \) Viscosity, Pas\(^{-1}\)  
\( \mu_t \) Turbulent viscosity, Pas\(^{-1}\)  
\( \omega \) Turbulent specific dissipation rate \((\text{SST } k-\omega)\)  
\( \omega \) Inverse turbulent time scale \((\text{Transition } k-kl-\omega)\)

Notes

1. Standard configuration refers to the section configuration tested in wind tunnel experiments with three elements. A main body at relatively \( 0^\circ \), a slat at \( 25^\circ \), and single flap deployed at \(-20^\circ\).\(^1\)

2. Adjusted configuration refers to the same wing section with retracted slat, deployed flap at \( 20^\circ \), and an air injection slot near the leading edge.

3. All simulations were performed using a wind oriented, Cartesian coordinate system.

4. Drag coefficients were given for a purely positive \( X \) direction, lift for a purely positive \( Y \) direction. Pitching moment was calculated about the centroid of the main body element, and was positive for a moment which induced a pitch nose-down.

\( \)
I. Introduction

Advancements in technology have led to more complex wing structures which include wing sections with multiple components. These added components on the leading and trailing edges aim at manipulating the boundary layer in order to increase a wing configuration’s performance through increased $C_{L_{\text{max}}}$ and increased $\alpha_{\text{stall}}$. Common configurations on commercial aircraft include a single leading edge slat with either single or multiple trailing edge flaps as seen in Figure 1b. During the research and development stage of aircraft design CFD and wind tunnel tests are performed in order to determine the optimal $d_{\text{gap}}$ and $\delta$ of wing section components to produce the highest coefficient of lift. The gap between the components is intended to allow cross flow between the high pressure flow on the bottom side of the wing, and the low pressure flow on the upper side as seen in Figure 1a.

(a) Boundary layer visualization of high lift configuration.2

(b) Modern high lift wing configuration with slats and flaps deployed.3

Figure 1: High lift wing section configurations.

Fluidic momentum injection to energize the boundary layer, also known simply as boundary layer blowing, is not a new concept. In fact, it has been experimented with in military applications dating back to the 1950’s, and commercially in the 1980’s. The American-Pioneered concept was known as a Boundary Layer Control System (BLCS), although the first systems employed a leading edge suction technique using porous materials in place of blowing.4 The concept was initially intended for fighter aircraft due to the aerodynamic properties of their wings. Fighter aircraft use thin wing profiles with small diameter leading edges for higher speed flight with minimized drag. However, during landing this poses a problem for most aircraft due to a leading edge separation bubble and high wing loads. The small diameter of the leading edge leads to a stall condition at small angles of attack, thus requiring greater thrust and airspeed during approach and landing. This is significant because it hinders the deployment opportunities of such aircraft due to runway length restrictions or load restrictions of arrested landings. This early system employed the Coandă effect to prevent stall across the wing when the flaps and ailerons were deployed for landing; a fluidic principle described by Tritton.4,5 This was the earliest example of boundary layer blowing, but differs from the concept tested in this investigation because air was only injected at the trailing edge, and the designs did not include leading edge slats. This investigation focuses on the viability of replacing modern mechanical slats on the leading edge with an air injection point. Modern day blown flap systems, such as that found on the Lockheed C-17 utilize engine exhaust in or-
In order to decrease wing loading while increasing flap effectiveness by employing the Coandă effect.\textsuperscript{6}

\section{Theory}

Although the wing section tested in this investigation only had a single trailing edge flap, the operational concept is the same as in Figure 1a. There are many principles of flight physics being applied simultaneously in order to increase lift and stall angle. The first of these is simple geometric manipulation. Extending the slat and flaps as seen in Figure 1 increases the positive camber of the wing section accomplishing two tasks concurrently to increase the lift coefficient. A wing section with a more positive camber will produce a greater lift.\textsuperscript{7} The increase in positive camber increases the distance by which the air on the upper surface must travel from leading to trailing edge, thus accelerating it and decreasing the pressure as is evident from the fundamentals of fluid mechanics. Furthermore, the positive camber will change the trailing edge down-wash angle. This angle change will create a downwash, which will alter the pitching moment and increase lift as is made evident from a free-body diagram balancing the forces.

\subsection{A. Boundary Layer Physics}

The boundary layer region is a near-wall region of flow where the free-stream flow interacts with the wall of the body, in this case the wing. The air nearest the wall is assumed to be at a zero velocity, and tangential velocity increases through the boundary layer until it matches free-stream velocity as seen at reference point 1 in Figure 2. When the flow meets an adverse pressure gradient, which can be caused by density changes as the flow is accelerated around the curvature of the wing profile, the tangent of the velocity profile becomes normal to the wing surface. At this point, seen at reference point 3 in Figure 2, the flow is said to have become separated. In the same image, the line denoted by “a” is the profile of the separation bubble. This bubble may extend to the trailing edge of the body, or may reattach into a turbulent boundary layer.

The impact of boundary layers on this investigation involves wake turbulence. When the boundary layers of the upper and lower surfaces of a body meet at the trailing edge they create a shear layer and induce turbulence in the flow. The turbulence intensity rises

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{boundary_layer.png}
\caption{Velocity profiles of boundary layers as it passes over a curved surface.\textsuperscript{8}}
\end{figure}
in the local region of the flow field and impacts boundary layers on subsequent bodies it meets downstream. This increased turbulence will induce an earlier and more rapid boundary layer transition. This is of particular importance to this investigation because turbulent boundary layers, although harmful to the drag performance of the profile, are advantageous in keeping flow attached longer in flows with local adverse pressure gradients.

B. Turbulence

The governing dynamics of the slat configuration which leads to higher lift coefficients is centered around boundary layer and flow channel mixing. Thus, in addition to capturing the boundary layer with a properly constructed mesh, turbulence modeling was priority in this investigation. Turbulence is a three-dimensional phenomena and should be modeled as such in order to achieve the most accurate results. For the most realistic results a direct numerical simulation is required, but the computational power exceeds what computer platforms can provide at the current time. Knowing this, the Navier-Stokes equations have been altered by Fluent to provide a time-averaged solution of the flow field. This process is known as Reynolds-Averaged Navier-Stokes (RANS). This effectively eliminates all turbulent structures such as vortecies from the flow field.

Specifics for coefficient calculations will be discussed in a subsequent subsection of method, however this section will cover generalities of turbulence modeling in Fluent via the ANSYS manual. Two models utilized in this investigation were the 2-equation Realizable $k-\epsilon$ model, and the more robust 5-equation Reynolds Stress Model (RSM).

III. Method

A. General Mesh and Test Parameters

The test airfoil and validation airfoil sections both had a chord length of one meter. In order to obtain good computational convergence a rule of thumb for pipeline measurement was used where measurements should be 5 diameter lengths from any in-line device which could disturb flow. The measurement benchmark for the simulations was convergence measurement of mass continuity at the inlet and outlet boundaries. In this case the chord length was the determinant dimension. Therefore, it was decided to use a 10 meter by 10 meter square test section with the airfoil centered within it. This would assure proper dissipation range for all flow non-uniformities and allow for the necessary convergence. As per ANSYS Fluent documentation the target convergence for directional
velocities, energy, and mass flow was $1 \cdot 10^{-6}$. However, values of $3 \cdot 10^{-5}$ were more realistic for some turbulence models, and $1 \cdot 10^{-6}$ was achievable for others.

The wing section was considered to coincide with an $\alpha$ of zero when the main wing component had a chord line which was parallel to the free stream flow direction since no criteria for $\alpha$ was designated in the wind tunnel results. This will be discussed more in later sections since this may have led to some discrepancies between the wind tunnel and CFD results. The standard Fluent coordinate system was employed such that lift was calculated in a pure positive Y direction, drag in a positive X direction, and pitching moment was associated with a nose-down moment about the center of mass of the center wing section.

B. Adjusted Configurations

Due to time constraints only a few altered configurations could be tested. All configurations injected air near the leading edge from a slot which had a width chosen to be equivalent to 2% of the chord. Since air would be assumed to travel from the engine compressors to the injection point via a piping system fully developed turbulent pipe flow properties were assumed for the injection flow.

The configurations seen in Figure 4 and Figure 5 were decided based on either geometric reference points, or previously constructed geometries which showed positive results as described by their captions.
C. Grid Creation

Initial attempts were made to create a hybrid structured, unstructured grid with ICEM. However, due to a lack of knowledge of the software and the complex nature of creating a viable 2D grid around a multi-element airfoil prevented the creation of such a grid. The author then turned to a newer, more intuitive software called Pointwise. Again, initial attempts were made to create a hybrid grid. Although successful in Pointwise, ultimately the grids would not import correctly into Fluent. The errors produced by Fluent prevented any simulations from being attempted.

As such, a fully unstructured grid as seen in Figure 6 was attempted and successfully simulated in Fluent. However, the consequences of using a grid of this type were unknown in the early stages of testing. A primary concern was the ability of an unstructured grid to fully resolve the boundary layer. Advancements in technology have allowed for the construction of much more complex geometries, and have led to research in alternative methods for grid generation in order to be able to reliably construct a grid in a reasonable amount of time. Recent research has shown that the employment of a fully unstructured grid generates very little error in the numerical data for external aerodynamic simulations in a majority of cases.\(^\text{12}\)

As mentioned before, boundary layer resolution was a particular concern when using a fully unstructured grid, especially flow downstream of the trailing edge of a body. As seen in Figure 7 the region between the main body and slat was particularly fine to ensure the resolution of the mixing region near the top leading edge of the main element. To ensure better quality results the node density near and on trailing edges was increased. It is difficult to judge scale in mesh images. For reference, the blunt edge on the trailing edge of the slat is 1.1mm in length. Additionally, the max distance between nodes along element walls was limited to a maximum of 10\(\mu\)m. This distribution was governed by a geometric formula within Pointwise. Post generation of the unstructured flow field, prior
to export to Fluent, quality metrics were checked overall as well as in specific areas of concern. Specifically for overly skewed or high aspect cells which may cause numerical solution issues.

D. Grid Validation

The geometry for this particular simulation was relatively very simple, however investigatory depth was limited by a lack of computational power. Therefore, given time and computational constraints it was decided that a fully unstructured grid constructed with the correct parameters in order to capture boundary layer effects would be a feasible and reliable method for this investigation. In order to validate the mesh construction method a NACA airfoil with well documented aerodynamic properties was used. NASA archive produces well documented performance values for the NACA 0009 airfoil which was selected for mesh validation purposes. Three different fully unstructured meshes were created and simulated with the NACA airfoil at a 0° incidence, and same free stream flow field conditions as shown in the NACA wind tunnel experiments.\textsuperscript{13} Mesh properties were as listed below in Table 1

<table>
<thead>
<tr>
<th></th>
<th>I</th>
<th>II</th>
<th>III</th>
</tr>
</thead>
<tbody>
<tr>
<td>$s_{\text{min}}$</td>
<td>$1 \cdot 10^{-5}$</td>
<td>$1 \cdot 10^{-5}$</td>
<td>$1 \cdot 10^{-4}$</td>
</tr>
<tr>
<td>$s_{\text{max}}$</td>
<td>0.2</td>
<td>0.2</td>
<td>0.2</td>
</tr>
<tr>
<td>$\delta s$</td>
<td>0.95</td>
<td>0.99</td>
<td>0.95</td>
</tr>
</tbody>
</table>

It was found that cases I and II produced the closest values of $C_L$, $C_D$, and $C_m$ relative
to the wind tunnel results. However, due to the decreased decay rate $\delta_s$ of case II, the node count doubled from 31,560 nodes. Given that the numerical results varied less than 0.1% between cases I and II, it was suspected that the values from case I should be used for multi-element airfoil mesh generation in order to reduce computational time. In order to confirm this, a test case was performed using the values from case I and II on the test airfoil section at $\alpha=0^\circ$ with slat and flap deployed. This generated meshes with 381k and 770k nodes respectively. Again, value differences were unremarkable, and so mesh generation parameters from case I were employed for the remainder of the meshes on the test airfoil configuration.

E. Grid Initialization

Given the complex nature of the flow field with mixing boundary layers and pressure zones, convergence became a problem at higher $\alpha$ angles, causing divergence within the first 100 iterations. This issue was addressed using the full multi-grid initialization feature of Fluent using through the text user interface. The FMG feature uses a series of grids which are gradually coarser from the original fine mesh.

Through a series of iterations on the coarse meshes as seen above in Figure 8, the original mesh is initialized to the best possible state. This process is computationally efficient, and in addition to providing better convergence results, allows for a quicker convergence as well.

![Figure 8: FMG initialization process as per ANSYS](image)

F. Turbulence Modeling

Due to the significant role turbulence played in boundary layer degradation in this investigation it was critical to choose a turbulence model which best represented wind tunnel data while also allowing for efficient computational time. Three models were investigated including Realizable k-$\epsilon$, and two variations of the k-$\omega$ model, SST and Transition k-$kl$. After reviewing the ANSYS Fluent theory guide these three models were chosen for specific reasons, although prior to testing it was unknown to the author which would produce the best results.

Although the ANSYS manual advised that most coefficients were suitable for a majority of the cases, however calculation of the turbulence parameters for inlet boundary conditions was essential prior to simulation. $I$ was assumed to be lowest for realistic departure situations, where the wing was the moving body relative the surrounding air. For this case the $I$ was an assumed value of 0.05%. Adjustments to this value will be
discussed in the section regarding the $k$-$kl$-$\omega$ model since this case was most sensitive to turbulence intensity for boundary layer transition calculations.

Turbulence parameters for external flows require estimation within given values from observed values. The procedure from the ANSYS Fluent was employed to derive values for $I$, $k$, $\epsilon$, and $\omega$. Based on information given by the ANSYS manual, the turbulent viscosity ratio $\mu_t/\mu$ was estimated to be 1. Additionally, $C_\mu$ is an empirical value estimated to be 0.09 in Fluent.\textsuperscript{10} Using the previously mentioned value for $I$, and the turbulent viscosity ratio the following sequence was used to determine the remaining values.\textsuperscript{10}

\begin{align*}
  k &= \frac{3}{2} (U_\infty I)^2 \\
  \epsilon &= \rho C_\mu \frac{k^2}{\mu} \left( \frac{\mu_t}{\mu} \right)^{-1} \\
  \omega &= \rho \frac{k}{\mu} \left( \frac{\mu_t}{\mu} \right)^{-1}
\end{align*}

1. **Realizable $k$-$\epsilon$ Model**

This model was chosen because coupled with the enhanced wall treatment option it offered good boundary layer resolution. However, it was known that results using this model may be unreliable. Although the boundary layer treatment was good, it is poor for use in external aerodynamic simulations because it lacks the ability to handle strong adverse pressure gradients.\textsuperscript{10} It follows that results near stall, and at high $\alpha$ angles would be questionable since regions of strong adverse pressure gradients surely exist.

2. **SST-$k$-$\omega$ Model**

The SST version of the $k$-$\omega$ model in Fluent has been altered to include elements from both the $\epsilon$ and $\omega$ families to help stabilize the results from free stream aerodynamic simulations.\textsuperscript{10} In addition, the SST version of the model uses the “enhanced wall treatment” option by default making it more ideal for external aerodynamic simulations. However, the resulting variation was nominal relative the Realizable $k$-$\epsilon$ simulations. Specifics will be provided later in results and discussion.

3. **Transition $k$-$kl$-$\omega$ Model**

This model offers the benefits of the previously discussed SST $k$-$\omega$ model with the added benefit of boundary layer transition capability. However, this model is highly sensitive to mesh defects, requires refined boundary layer meshing resolution, and accurate turbulence coefficient calculations for boundary conditions. This naturally introduced a bit of error in the simulations. External flows can be approximated as low as 0.05% for turbulent intensity.\textsuperscript{14} However, this quality of flow is unlikely to occur in a wind tunnel, and so the simulation would need to be performed twice with different $I$ values. Simulated first under wind tunnel conditions with to validate the reliability of the CFD results, and once in external flow conditions to investigate the viability of the boundary layer blowing configuration.

Turbulence factor is a common scaling factor given in wind tunnel testing to quantify the turbulence intensity of the flow field and can be calculated as seen below in (4).

\[ TF = \frac{385000}{RN_c} \]
The turbulence factor method was based on a NACA publication from the late 1920’s, and was further developed in a technical publication by Barlow et al. This method is still relevant today, being applied in modern wind tunnels.

The comparative wind tunnel tests were performed in a wind tunnel with $TF = 1.068$. From this number we can conclude that the turbulence intensity of the wind tunnel is approximately 0.8%.

IV. Results

A. Turbulence Model Comparison

The simulation of the standard configuration showed good correlation with the wind tunnel testing results. The Transition $k-kl-\omega$ model showed the best correlation up to stall with a value error of -2.6% on average. The base $k-\omega$ and $k-\epsilon$ models showed a value deviation of 5% to 6%, with the $\epsilon$ regime diverging at higher $\alpha$ values due to its adverse pressure gradient sensitivity. Figure 9 clearly shows all models failed to correctly predict stall. As previously mentioned this was one of the motivations for testing free stream turbulence alterations. Each turbulence model was simulated with an intensity $I$ value of 0.05% and 0.8%. The numerical deviation was negligible with turbulent shift and all six regimes failed to correctly predict the stall point.

![Turbulence Model Comparison with Experimental Results](image_url)

Figure 9: Comparison of NHPL wind tunnel results with the tested CFD turbulence models on the standard configuration.
B. Standard and Adjusted Configuration Comparison

Using the wind tunnel data as a benchmark, snapshots of pressure and velocity distributions were taken of the standard and altered configurations at the $\alpha$ correlating to the maximum lift coefficient prior to stall.

![Standard configuration pressure distribution at $\alpha=22^\circ$.](image1)

![Standard configuration velocity distribution at $\alpha=22^\circ$.](image2)

Figure 10: Standard configuration distributions using $k-kl-\omega$ model at wind tunnel stall point.

The visuals in Figure 10 are not only meant for comparative purposes, but to help the reader visualize the functionality of the multi-element configuration as previously discussed. The angle deviation of the slat from the main element, specifically 25° for this case, prevents stall across the upper surface of the slat due to the change in wall angle relative free stream flow direction. Furthermore, the gap between the slat and main body plays a crucial role in preventing stall. The stagnation of flow trailing the slat body can be seen clearly, causing a pressure differential between upper and lower bounds of the gap, allowing air to flow through the gap and over the top of the main body. This energizes the flow, delaying boundary layer separation across the upper surface of the main body. The gap between the main body and flap operates in an identical fashion for the same purpose.

None of the four altered configurations achieved comparable lift coefficient at the same angle of inclination as the standard configuration. All the configurations exhibited a fully stalled flow regime across the upper surface at this angle as seen in Figure 11.

![Figure 11: T1 configuration at $\alpha=22^\circ$ in fully stalled regime.](image3)

With the exception of the configuration seen in Figure 12, the stall occurred at a relatively small percentage of the chord on the upper surface. The detachment point seen in Figure 13a was typical of all altered configurations. Shown in Figure 13 are configurations B1 and B2 with an injection velocity $v_i$ equivalent to 100 meters per second. A wide range of injection velocities were tested between 60 and 120 meters per
second. The result of a change in $v_i$ was minimal, only shifting the stagnation point on the leading edge, but never delaying stall.

![Figure 12: T2 configuration implementing Transition $k$-$kl$-$\omega$ model at wind tunnel stall point with $v_i=100\text{ms}^{-1}$](image)

The T2 configuration attempted to inject momentum at an early chord wise position near the observed stall point in order to inhibit this stall. As can be seen in Figure 12b the separation does occur at a later chord-wise position, however the flow still separates. This maybe a combination of injection angle, injection velocity, and geometry at the wall injection port inlet, although reattachment could not be achieved with injection velocity variation.

![Figure 13: Distributions of B configurations using Transition $k$-$kl$-$\omega$ model at wind tunnel stall point with $v_i=100\text{ms}^{-1}$](image)

Table 2 shows some comparative lift results. Note that the percentage values were calculated using standard statistical percent error given by (5). The lift coefficient values from both Table 2 and Figure 9 for experimental results were estimated from graphical data presented by Moir et al. because no numerical table values were published. The plot was imported to photoshop and aligned with the software coordinate system. A grid with associated scale markers was then placed over the plot to determine the $C_L$ values.
to the nearest hundredth.

\[
\%Error = \frac{|Experimental - Theoretical|}{Theoretical} \times 100
\] (5)

Table 2: \( C_L \) comparison across all configurations and models.

<table>
<thead>
<tr>
<th>CFG.</th>
<th>Model</th>
<th>( \alpha ) (°)</th>
<th>( C_L )</th>
<th>%</th>
</tr>
</thead>
<tbody>
<tr>
<td>Std.</td>
<td>Experimental</td>
<td>22</td>
<td>4.18</td>
<td>-</td>
</tr>
<tr>
<td>Std.</td>
<td>( k-\epsilon )</td>
<td>22</td>
<td>3.98</td>
<td>4.92</td>
</tr>
<tr>
<td>Std.</td>
<td>SST ( k-\omega )</td>
<td>22</td>
<td>4.08</td>
<td>2.51</td>
</tr>
<tr>
<td>Std.</td>
<td>Transition ( k-kl-\omega )</td>
<td>22</td>
<td>4.22</td>
<td>0.960</td>
</tr>
<tr>
<td>T1</td>
<td>Transition ( k-kl-\omega )</td>
<td>22</td>
<td>1.84</td>
<td></td>
</tr>
<tr>
<td>T2</td>
<td>Transition ( k-kl-\omega )</td>
<td>22</td>
<td>1.74</td>
<td></td>
</tr>
<tr>
<td>B1</td>
<td>Transition ( k-kl-\omega )</td>
<td>22</td>
<td>1.41</td>
<td></td>
</tr>
<tr>
<td>B2</td>
<td>Transition ( k-kl-\omega )</td>
<td>22</td>
<td>1.77</td>
<td></td>
</tr>
</tbody>
</table>

Again computational and time limitations played a role in the number of simulations allowed with the altered configurations. With the need for a variation in injection speed to be investigated as well as the injection geometry, the ability to simulate different turbulence models was sacrificed. Naturally, the model with the closest relation to wind tunnel data was chosen for simulating all altered configurations.

V. Discussion

There were several issues with this alternative method for use as a high lift device which came to light during both the research and simulation phases. This investigation hoped to show potential for a retrofit style alteration of existing aircraft in order to improve performance and efficiency by eliminating mass from the leading edge of the wing. This preliminary investigation shows that this method is most likely not viable without major geometry alterations to the wing section. The leading edge radius, maximum camber and thickness chord-wise locations would all most likely need refinement for a system to work properly. Furthermore, changing the mass concentration along the leading edge would modify many other performance features such as the aeroelastic stability of the wing due to a change in local rigidity and shear center. Changing aeroelastic properties would require recertification of the aircraft, a costly process. With this in mind, the engineers should design the wing for such a system from the beginning, making a retrofit kit not feasible.

Even with the costly design change investments, the configurations tested in this investigation failed to produce even 50% of the lift generated by the common multi-component design. This lack of lift at departure would drastically lower the performance of the aircraft, requiring longer times to reach cruise altitude. The lower lift coefficient would require a higher dynamic pressure in order to produce the same lift to counter the weight of the aircraft, and therefore require longer runways to reach a higher lift-off velocity. Simply put, without matching existing performance profiles, no alteration is possible because it would cripple the aircraft’s flight envelope, severely limiting its operational locations around the globe.
Moreover, one-engine-inoperable (OEI) departure is crucial to aircraft certification. Certification requirements state that an aircraft must be able to perform a departure within normal weight parameters without the use of all engines. In the case of such an event, the lift-off distance and velocities are adjusted in order to allow for a safe departure. With a mechanically controlled flow regime around the wing, lifting capabilities are still possible, and only adjustments need to be made for the reduced thrust and increased yaw. In the case of the blowing boundary layer control system the lift on one wing would be severely compromised since the injection air would theoretically be sourced as bleed air from each engine. This would most likely result in an inability of the aircraft to achieve flight. Thus, certification standards would not be met.

There were also simulation issues which introduce more uncertainty regarding this method. Although the Transition $k$-$kl$-$\omega$ model exhibited excellent agreement with wind tunnel data, less than 1% deviation, all simulations failed to correctly predict stall. All three overestimated the stall angle, and in some cases by as much as six degrees. One publication suggests that using time-averaged models within CFD simulations will lead to an incorrect estimation of stall. This publication states that the time-averaged approach “under-predicts” flow separation around the wing and therefore will provide an “optimistic” value for stall angle. Although the team used CFX in their investigation, the turbulence models analyzed therein were the same as used in this investigation and therefore their findings establish probable cause for the overestimation of stall in this case.

VI. Conclusion

The primary aim of this investigation was to test the feasibility of a boundary layer blowing system as a substitute for mechanical slats on the leading wing edge for use during departure to allow for increased lift capability and delayed stall performance. It has been clearly shown that the configurations tested in this investigation were not viable solutions. None of the four alternate configurations produced half of the lift as the standard three-element configuration. There maybe a specific configuration of the injection point which would allow for potential implementation on existing aircraft as a retrofit.

Yet, it has also been established that the boundary layer blowing system presents many other issues. Primarily, a safety concern for aircraft performance under an OEI departure. The aircraft would most likely not possess the necessary lift potential to depart within runway limitations. Furthermore, the alteration of a wing in such a manner would present considerable need for redesign with respect to aeroelastic and other structural properties. Without addressing these two issues the aircraft will not meet certification requirements under FAA and EASA regulations.

An unintended side effect of this investigation was validation of meshing methodology. General practice within the computational fluid dynamics community has been to use a hybrid mesh when simulating wing sections in a two-dimensional regime. A structured mesh immediately surrounding the wing wall provides ample resolution of the boundary layer, while an unstructured mesh allows for a computationally efficient far-field. In this investigation an unconventional meshing method employing a fully unstructured grid proved that reliable results can be achieved in accordance with wind tunnel results. This allows for simulations where computational power and time is at a premium. Again, one must consider that CFD models currently underestimate separation and hence overestimate stall. Although this consequence will apply to both the conventional hybrid and
unconventional fully unstructured grid, and a hybrid construction may still be required in some cases so the user is urged to consider the parameters of the investigation before deciding on a meshing structure.
## Appendix

Table 3: Standard Configuration with Realizable $k$-$\epsilon$ Turbulence Model Applied.

<table>
<thead>
<tr>
<th>$\alpha$ (°)</th>
<th>$C_m$</th>
<th>$C_L$</th>
<th>$C_D$</th>
<th>Mass Continuity Residual</th>
<th>Iterations</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>1.76837</td>
<td>1.4782</td>
<td>3.1557·10^{-2}</td>
<td>1.6135·10^{-5}</td>
<td>1000</td>
</tr>
<tr>
<td>1</td>
<td>1.533</td>
<td>1.6225</td>
<td>3.7695·10^{-2}</td>
<td>1.417·10^{-5}</td>
<td>6000</td>
</tr>
<tr>
<td>2</td>
<td>1.2882</td>
<td>1.7579</td>
<td>3.9883·10^{-2}</td>
<td>2.6597·10^{-5}</td>
<td>5000</td>
</tr>
<tr>
<td>3</td>
<td>1.0376</td>
<td>1.8909</td>
<td>4.2792·10^{-2}</td>
<td>2.8121·10^{-5}</td>
<td>2500</td>
</tr>
<tr>
<td>4</td>
<td>7.8595·10^{-1}</td>
<td>2.0217</td>
<td>4.5897·10^{-2}</td>
<td>2.3385·10^{-5}</td>
<td>5000</td>
</tr>
<tr>
<td>5</td>
<td>5.41364·10^{-1}</td>
<td>2.15165</td>
<td>5.00194·10^{-2}</td>
<td>1.9253·10^{-5}</td>
<td>5000</td>
</tr>
<tr>
<td>6</td>
<td>2.8676·10^{-1}</td>
<td>2.2845</td>
<td>5.3609·10^{-2}</td>
<td>2.0534·10^{-5}</td>
<td>5000</td>
</tr>
<tr>
<td>7</td>
<td>3.1224·10^{-2}</td>
<td>2.4126</td>
<td>5.7428·10^{-2}</td>
<td>1.8692·10^{-5}</td>
<td>5000</td>
</tr>
<tr>
<td>8</td>
<td>-2.18389·10^{-1}</td>
<td>2.54133</td>
<td>6.17895·10^{-2}</td>
<td>2.0158·10^{-5}</td>
<td>3117</td>
</tr>
<tr>
<td>9</td>
<td>-4.7284·10^{-1}</td>
<td>2.6623</td>
<td>6.6041·10^{-2}</td>
<td>1.6947·10^{-5}</td>
<td>2617</td>
</tr>
<tr>
<td>10</td>
<td>-7.2206·10^{-1}</td>
<td>2.7846</td>
<td>7.1277·10^{-2}</td>
<td>1.5166·10^{-5}</td>
<td>2500</td>
</tr>
<tr>
<td>11</td>
<td>-9.6701·10^{-1}</td>
<td>2.8973</td>
<td>7.6298·10^{-2}</td>
<td>1.4464·10^{-5}</td>
<td>2808</td>
</tr>
<tr>
<td>12</td>
<td>-1.2119</td>
<td>3.0228</td>
<td>8.1866·10^{-2}</td>
<td>1.3364·10^{-5}</td>
<td>2500</td>
</tr>
<tr>
<td>13</td>
<td>-1.4538</td>
<td>3.1487</td>
<td>8.7892·10^{-2}</td>
<td>1.4431·10^{-5}</td>
<td>2500</td>
</tr>
<tr>
<td>14</td>
<td>-1.6981</td>
<td>3.2529</td>
<td>9.3486·10^{-2}</td>
<td>1.64786·10^{-5}</td>
<td>2584</td>
</tr>
<tr>
<td>15</td>
<td>-1.9366</td>
<td>3.3546</td>
<td>1.0018·10^{-1}</td>
<td>1.1599·10^{-5}</td>
<td>2500</td>
</tr>
<tr>
<td>16</td>
<td>-2.1841</td>
<td>3.4635</td>
<td>1.0618·10^{-1}</td>
<td>1.0926·10^{-5}</td>
<td>2500</td>
</tr>
<tr>
<td>17</td>
<td>-2.3921</td>
<td>3.5568</td>
<td>1.1350·10^{-1}</td>
<td>1.2979·10^{-5}</td>
<td>2500</td>
</tr>
<tr>
<td>18</td>
<td>-2.6178</td>
<td>3.6562</td>
<td>1.2079·10^{-1}</td>
<td>1.1982·10^{-5}</td>
<td>3000</td>
</tr>
<tr>
<td>19</td>
<td>-2.8414</td>
<td>3.7439</td>
<td>1.2870·10^{-1}</td>
<td>1.2903·10^{-5}</td>
<td>2500</td>
</tr>
<tr>
<td>20</td>
<td>-3.0648</td>
<td>3.8257</td>
<td>1.3571·10^{-1}</td>
<td>9.9970·10^{-6}</td>
<td>2801</td>
</tr>
<tr>
<td>21</td>
<td>-3.2624</td>
<td>3.9083</td>
<td>1.4429·10^{-1}</td>
<td>1.2863·10^{-5}</td>
<td>3000</td>
</tr>
<tr>
<td>22</td>
<td>-3.4560</td>
<td>3.9841</td>
<td>1.5231·10^{-1}</td>
<td>1.3261·10^{-5}</td>
<td>3000</td>
</tr>
<tr>
<td>23</td>
<td>-3.6480</td>
<td>4.0408</td>
<td>1.6053·10^{-1}</td>
<td>9.9884·10^{-6}</td>
<td>3355</td>
</tr>
<tr>
<td>24</td>
<td>-3.8266</td>
<td>4.1025</td>
<td>1.6915·10^{-1}</td>
<td>1.2024·10^{-5}</td>
<td>4500</td>
</tr>
<tr>
<td>25</td>
<td>-3.9739</td>
<td>4.1317</td>
<td>1.7929·10^{-1}</td>
<td>3.1093·10^{-5}</td>
<td>3000</td>
</tr>
<tr>
<td>26</td>
<td>-4.1453</td>
<td>4.1702</td>
<td>1.8838·10^{-1}</td>
<td>9.9810·10^{-6}</td>
<td>2045</td>
</tr>
<tr>
<td>28</td>
<td>-4.2821</td>
<td>4.2342</td>
<td>1.9815·10^{-1}</td>
<td>8.4972·10^{-6}</td>
<td>3000</td>
</tr>
<tr>
<td>28</td>
<td>-4.3828</td>
<td>4.2369</td>
<td>2.0993·10^{-1}</td>
<td>9.2474·10^{-6}</td>
<td>5822</td>
</tr>
<tr>
<td>29</td>
<td>-4.4421</td>
<td>4.2050</td>
<td>2.2040·10^{-1}</td>
<td>9.9903·10^{-6}</td>
<td>5874</td>
</tr>
</tbody>
</table>
Table 4: Standard Configuration with SST $k$-$\omega$ Turbulence Model Applied.

<table>
<thead>
<tr>
<th>$\alpha$ ($^\circ$)</th>
<th>$C_m$</th>
<th>$C_L$</th>
<th>$C_D$</th>
<th>Mass Continuity Residual</th>
<th>Iterations</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>1.7572</td>
<td>1.4858</td>
<td>3.4084-10^{-2}</td>
<td>4.6986 -10^{-6}</td>
<td>3781</td>
</tr>
<tr>
<td>5</td>
<td>5.2932·10^{-1}</td>
<td>2.1527</td>
<td>4.7938-10^{-2}</td>
<td>9.9746·10^{-6}</td>
<td>1888</td>
</tr>
<tr>
<td>10</td>
<td>-7.3472·10^{-1}</td>
<td>2.7925</td>
<td>6.7564-10^{-2}</td>
<td>9.9923·10^{-6}</td>
<td>1151</td>
</tr>
<tr>
<td>15</td>
<td>-1.9486</td>
<td>3.3794</td>
<td>9.4635-10^{-2}</td>
<td>9.9759·10^{-6}</td>
<td>1379</td>
</tr>
<tr>
<td>15*</td>
<td>-1.9554</td>
<td>3.3687</td>
<td>9.4295-10^{-2}</td>
<td>9.9823·10^{-6}</td>
<td>1937</td>
</tr>
<tr>
<td>20</td>
<td>-3.0812</td>
<td>3.8928</td>
<td>1.2842-10^{-1}</td>
<td>9.9888·10^{-6}</td>
<td>1484</td>
</tr>
<tr>
<td>21</td>
<td>-3.2823</td>
<td>3.9878</td>
<td>1.3661-10^{-1}</td>
<td>9.9609·10^{-6}</td>
<td>1571</td>
</tr>
<tr>
<td>22</td>
<td>-3.4747</td>
<td>4.0776</td>
<td>1.4461-10^{-1}</td>
<td>9.9987·10^{-6}</td>
<td>2888</td>
</tr>
<tr>
<td>23</td>
<td>-3.6716</td>
<td>4.1502</td>
<td>1.5248-10^{-1}</td>
<td>9.9843·10^{-6}</td>
<td>3147</td>
</tr>
</tbody>
</table>

Table 5: Standard Configuration with Transition $k$-$kl$-$\omega$ Turbulence Model Applied.

<table>
<thead>
<tr>
<th>$\alpha$ ($^\circ$)</th>
<th>$C_m$</th>
<th>$C_L$</th>
<th>$C_D$</th>
<th>Mass Continuity Residual</th>
<th>Iterations</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>5.5689·10^{-1}</td>
<td>2.2037</td>
<td>4.8094-10^{-2}</td>
<td>6.0817·10^{-5}</td>
<td>5458</td>
</tr>
<tr>
<td>10</td>
<td>-6.9473·10^{-1}</td>
<td>2.8657</td>
<td>6.6805-10^{-2}</td>
<td>1.9869·10^{-5}</td>
<td>4000</td>
</tr>
<tr>
<td>15</td>
<td>-1.9124</td>
<td>3.4697</td>
<td>9.3132·10^{-2}</td>
<td>2.8178·10^{-6}</td>
<td>2506</td>
</tr>
<tr>
<td>20</td>
<td>-3.0602</td>
<td>4.0143</td>
<td>1.241·10^{-1}</td>
<td>1.0065·10^{-6}</td>
<td>3295</td>
</tr>
<tr>
<td>21</td>
<td>-3.2708</td>
<td>4.1179</td>
<td>1.3235·10^{-1}</td>
<td>1.0012·10^{-6}</td>
<td>3147</td>
</tr>
<tr>
<td>22</td>
<td>-3.4712</td>
<td>4.2205</td>
<td>1.3934·10^{-1}</td>
<td>1.0022·10^{-6}</td>
<td>3221</td>
</tr>
<tr>
<td>23</td>
<td>-3.6808</td>
<td>4.2944</td>
<td>1.4632·10^{-1}</td>
<td>1.0078·10^{-6}</td>
<td>3135</td>
</tr>
<tr>
<td>24</td>
<td>-3.8735</td>
<td>4.3845</td>
<td>1.5479·10^{-1}</td>
<td>1.0011·10^{-6}</td>
<td>2542</td>
</tr>
<tr>
<td>25</td>
<td>-4.0534</td>
<td>4.4687</td>
<td>1.6164·10^{-1}</td>
<td>1.0043·10^{-6}</td>
<td>3244</td>
</tr>
</tbody>
</table>

Acknowledgments

I would like to thank my project adviser Arne Karlsson (Kungliga Tekniska Högskolan, Department Aeronautics and Vehicle Engineering) for his guidance during this investigation.
References