DOWNLOAD PDF STUDY OF THE UNSTEADY FLOW FEATURES ON A STALLED WING

Chapter 1 : Computational Study of Unsteady Flows around Dragonfly and - blog.quintoapp.com

The occurrence of large scale structures in the post stall flow over a rectangular wing at high angles of attack was investigated in a small-scale subsonic wind tunnel.

Deep stall condition â€” T-tail in "shadow" of wing The deep stall affects aircraft with a T-tail configuration. A deep stall or super-stall is a dangerous type of stall that affects certain aircraft designs, [31] notably jet aircraft with a T-tail configuration and rear-mounted engines. In these designs, the turbulent wake of a stalled main wing, nacelle-pylon wakes and the wake from the fuselage [32] "blanket" the horizontal stabilizer, rendering the elevators ineffective and preventing the aircraft from recovering from the stall. Taylor [33] states T-tail propeller aircraft, unlike jet aircraft, do not usually require a stall recovery system during stall flight testing due to increased airflow over the wing root from the prop wash. Nor do they have rear mounted nacelles which can contribute substantially to the problem. He also gives a definition that relates deep stall to a locked-in condition where recovery is impossible. Typical values both for the range of deep stall, as defined above, and the locked-in trim point are given for the Douglas DC-9 Series 10 by Schaufele. The final design had no locked in trim point so recovery from the deep stall region was possible, as required to meet certification rules. Taylor and Ray [39] show how the aircraft attitude in the deep stall is relatively flat, even less than during the normal stall, with very high negative flight path angles. Effects similar to deep stall had been known to occur on some aircraft designs before the term was coined. A prototype Gloster Javelin serial WD was lost in a crash on 11 June , to a "locked in" stall [40] However, Waterton [41] states that the trimming tailplane was found to be the wrong way for recovery. Low speed handling tests were being done to assess a new wing. The brake parachute had not been streamed as it may have hindered rear crew escape. Stick shakers are now a standard part of commercial airliners. Nevertheless, the problem continues to cause accidents; on 3 June , a Hawker Siddeley Trident G-ARPY , was lost to deep stall ; [45] deep stall is suspected to be cause of another Trident the British European Airways Flight G-ARPI crash â€” known as the "Staines Disaster" â€” on 18 June when the crew failed to notice the conditions and had disabled the stall recovery system. It recovered from the deep stall after deploying the anti-spin parachute but crashed after being unable to jettison the chute or relight the engines. One of the test pilots was unable to escape from the aircraft in time and was killed. Two Velocity aircraft crashed due to locked-in deep stalls. Please help improve this section by adding citations to reliable sources. Unsourced material may be challenged and removed. November Learn how and when to remove this template message Aircraft with a swept wing suffer from a particular form of stalling behaviour at low speed. At high speed the airflow over the wing tends to progress directly along the chord, but as the speed is reduced a sideways component due to the angle of the leading edge has time to build up. Airflow at the root is affected only by the angle of the wing, but at a point further along the span, the airflow is affected both by the angle as well as any sideways component of the airflow from the air closer to the root. This results in a pattern of airflow that is progressively "sideways" as one moves toward the wingtip. As it is only the airflow along the chord that contributes to lift, this means that the wing begins to develop less lift at the tip than the root. In extreme cases, this can lead to the wingtip entering stall long before the wing as a whole. In this case the average lift of the wing as a whole moves forward; the inboard sections are continuing to generate lift and are generally in front of the center of gravity C of G , while the tips are no longer contributing and are behind the C of G. This produces a strong nose-up pitch in the aircraft, which can lead to more of the wing stalling, the lift moving further forward, and so forth. This chain reaction is considered very dangerous and was known as the pitch-up. Tip stall can be prevented in a number of ways, at least one of which is found on almost all modern aircraft. An early solution was the addition of wing fences to re-direct sideways moving air back towards the rear of the wing. A similar solution is the dog-tooth notch seen on some aircraft, like the Avro Arrow. A more common modern solution is to use some degree of washout. Warning and safety devices[edit] Fixed-wing aircraft can be equipped with devices to prevent or postpone a stall or to make it less or in some cases more severe, or to make recovery easier. An aerodynamic twist can be introduced to the wing with the leading edge near the wing tip twisted downward. This is called washout and causes the wing root to stall before the wing tip. This makes the stall gentle and progressive. Since the stall is delayed at the wing tips, where the ailerons are, roll control is maintained when the stall begins. A stall strip is a small sharp-edged device that, when attached to the leading edge of a wing, encourages the stall to start there in preference to any other location on the wing. If attached close to the wing root, it makes the stall gentle and progressive; if attached near the wing tip, it encourages the aircraft to drop a wing when stalling. A stall fence is a flat plate in the direction of the chord to stop separated flow progressing out along the wing [59] Vortex generators , tiny strips of metal or plastic placed on top of the wing near the leading edge that protrude past the boundary layer into the free stream. As the name implies, they energize the boundary layer by mixing free stream airflow with boundary layer flow thereby creating vortices, this increases the momentum in the boundary layer. By increasing the momentum of the boundary layer, airflow separation and the resulting stall may be delayed. An anti-stall strake is a leading edge extension that generates a vortex on the wing upper surface to postpone the stall. A stick pusher is a mechanical device that prevents the pilot from stalling an aircraft. It pushes the elevator control forward as the stall is approached, causing a reduction in the angle of attack. In generic terms, a stick pusher is known as a stall identification device or stall identification system. A stall warning is an electronic or mechanical device that sounds an audible warning as the stall speed is approached. The majority of aircraft contain some form of this device that warns the pilot of an impending stall. The simplest such device is a stall warning horn, which consists of either a pressure sensor or a movable metal tab that actuates a switch , and produces an audible warning in response. An AOA indicator provides a visual display of the amount of available lift throughout its slow speed envelope regardless of the many variables that act upon an aircraft. This indicator is immediately responsive to changes in speed, angle of attack, and wind conditions, and automatically compensates for aircraft weight, altitude, and temperature. An angle of attack limiter or an "alpha" limiter is a flight computer that automatically prevents pilot input from causing the plane to rise over the stall angle. Some alpha limiters can be disabled by the pilot. Stall warning systems often involve inputs from a broad range of sensors and systems to include a dedicated angle of attack sensor. Blockage, damage, or inoperation of stall and angle of attack AOA probes can lead to unreliability of the stall warning, and cause the stick pusher, overspeed warning, autopilot, and yaw damper to malfunction. Therefore, when the aircraft pitch increases abnormally, the canard will usually stall first, causing the nose to drop and so preventing the wing from reaching its critical AOA. Thus, the risk of main wing stalling is greatly reduced. However, if the main wing stalls, recovery becomes difficult, as the canard is more deeply stalled and angle of attack increases rapidly. In this case, the wing can be flown at higher lift coefficient closer to stall to produce more overall lift. Flight beyond the stall[edit] As a wing stalls, aileron effectiveness is reduced, making the plane hard to control and increasing the risk of a spin starting. Post stall , steady flight beyond the stalling angle where the coefficient of lift is largest requires engine thrust to replace lift as well as alternative controls to replace the loss of effectiveness of the ailerons. For high-powered aircraft, the loss of lift and increase in drag beyond the stall angle is less of a problem than maintaining control. Some aircraft may be subject to post-stall gyration e. Control beyond-stall can be provided by reaction control systems e. NFA , vectored thrust, as well as a rolling stabilator or taileron. The enhanced manoeuvering capability by flights at very high angles of attack can provide a tactical advantage for military fighters such as the F Raptor. Spoiler aeronautics Except for flight training, airplane testing, and aerobatics , a stall is usually an undesirable event. Unlike powered airplanes, which can control descent by increasing or decreasing thrust, gliders have to increase drag to increase the rate of descent. In high-performance gliders, spoiler deployment is extensively used to control the approach to landing. Spoilers can also be thought of as "lift reducers" because they reduce the lift of the wing in which the spoiler resides. For example, an uncommanded roll to the left could be reversed by raising the right wing spoiler or only a few of the spoilers present in large airliner wings. This has the advantage of avoiding the need to increase lift in the wing that is dropping which may bring that wing closer to stalling. History[edit] Otto Lilienthal died while flying in as the result of a stall. Wilbur Wright encountered stalls for the first time in , while flying his second glider. This made recoveries from stalls easier and more gentle. In developing the resulting " autogyro " aircraft, he solved many engineering problems which made the helicopter possible.

Chapter 2 : Ansell Wins Young Investigator Award to Study Flow in Dynamic Stall | Aerospace at Illinois

Note: Citations are based on reference standards. However, formatting rules can vary widely between applications and fields of interest or study. The specific requirements or preferences of your reviewing publisher, classroom teacher, institution or organization should be applied.

The simulations are performed with a very wide computational domain 10 chord length to minimize the influence of spanwise periodic boundary con- ditions. For the URANS simulations, four different spanwise mesh resolutions are tested to determine the minimum resolution required to capture the formation of stall cells. The DDES results show much more complex flow patterns over the airfoil at these high angles of attack, although the spectral analysis of wall shear stress suggests the existence of flow structures having a similar spanwise length scale to the stall cells. Introduction Stall cells are three-dimensional separation patterns that can be observed on the suction side of air- foils near stall, initiated by the detachment of flow from the trailing edge. Their existence has been Preprint submitted to Elsevier September 30, documented over the past half century [1, 2]. A thorough review of these early studies has been provided in [4]. The geometrical features of these flow patterns are known to be affected by the aspect ratio of the wing [5], Reynolds number [3], shape of the airfoil and the angle of attack [6]. The presence of stall cells could be related to the lift hysteresis, since their formation lowers the aerodynamic unsteady forces [1, 6]. Moreover, the range of angles of attack in which the stall cells are observed is usually narrow. The topology of the stall cells has been described further in [7] as a couple of counter rotating vortices on the airfoil suction side developing in the streamwise direction, while the dynamic behavior of these patterns has been described in [8] depending on the Reynolds number, angle of attack and free stream turbulence variations. The formation of stall cells has not been fully explained from a theoretical point of view; however, several possible physical mechanisms have been proposed to date. Weihs and Katz [9] suggested the Crow instability [10] to be at the origin of the stall cells emergence, and thereby derived a relationship to estimate the spanwise dimension of the cells. In the work of Bragg et al. The effect of the Reynolds number has been investigated by Schewe [3], showing that laminar-turbulent transition could play a key role. Also, Rodriguez and Theofilis [12] found surface streamlines very similar to stall cells as a result of a global linear stability analysis of two-dimensional steady laminar separated flow superimposed by three-dimensional stationary disturbances. They have suggested that the formation of stall cells that can be observed in experiments at much higher Reynolds numbers is also due to a global instability, although their stability analysis was for a very low Reynolds number. More recently, the existence of multiple solutions to the Navier-Stokes equations has been demon- strated by Kamenetskiy et al. The flow patterns observed in some of these multiple solutions were similar to the stall cells, again suggesting that the formation of stall cells is quite sensitive to the stability of different flow patterns that could be observed in the post-stall range of angles of attack. This issue was discussed further by Spalart [14], who explained the spontaneous formation of stall cells using a periodic version of the lifting line theory. The uniform flow solution is not stable for a branch of the lift curve with a negative slope, resulting in a preferred pattern involving a finite number of cells. This is because the wing vortex interaction is self correcting with respect to spanwise perturbations only when the lift curve has a positive slope i. Along the same line of thinking, Gross et al. When a small separation zone appears at one spanwise location near the trailing edge, some localized flow can arise around the trailing edge from the pressure side to the suction side, leading to a local decrease in circulation due to the locally increased vertical velocity. Similarly, in a section between two zones of such a localized separation, there will be a local increase in circulation. Based on this argument, Gross et al. Gross and Fasel [19] employed a narrow computational domain with symmetric boundaries instead of periodic boundaries and observed a curved separation line, which the authors attributed to the presence of stall cells. Also, Manolesos et al. A summary of experimental studies about stall cells is provided in table 1, and that for computational studies is provided in table 2. The Reynolds number range in which stall cells are observed is typically around the order of one million [17], and since it is close to the operating range of wind turbine blades, this topic has recently been attracting the attention of many researchers in wind energy as well. An important issue from a practical point of view regarding the prediction of stall cells is its sensitivity to the mesh resolution in the spanwise direction [18]. This is a fundamentally important issue, for example, in the simulations of wind turbines with long blades, where a relatively coarse blade-spanwise resolution is often used in order to reduce computational costs. The formation of stall cells may affect the performance of both vertical axis turbines, where rotating blades tend to experience very high angles of attack, and horizontal axis turbines, where the progressive separation of flow from the root to the tip of the blades could be affected by the stall cells appearance. Furthermore, the recent developments of Unmanned Aerial Vehicles UAV have increased the interest in this phenomena observed in this range of Reynolds numbers. A key aspect of the present study is that we employ a very wide computational domain 10 chord length to minimize the influence of spanwise periodic boundary conditions. We test four different spanwise resolutions to determine the minimum spanwise resolution required to capture the formation of stall cells. We also test various angles of attack with a small increment every 0. Summary of experimental studies on stall cells. Summary of numerical studies on stall cells. Mesh and boundary conditions A series of preliminary 2D simulations has been carried out first to design a 2D mesh with 50, elements in total that yields mesh-independent 2D results. The mesh independence has been confirmed by doubling the number of nodes on each direction until the results in terms of lift and drag predictions did not change with further refinement. A summary of these 2D simulations is provided in table 3. A fully structured mesh is used in this study to minimize numerical diffusion and also to avoid any significant mesh-induced perturbation in the spanwise direction that could affect the formation of stall cells. The computational domain extends 30 chords in x and y directions from the leading edge of the airfoil, as can be seen in figure 1. A close view of the 2D mesh created is presented in figure 2. Once the trustworthy 2D mesh had been created, it has been extruded in the spanwise direction for 10 chord length to create a 3D mesh. Four different 3D meshes have been generated with 10, 20, 50 and nodes, respectively, along the spanwise direction. For the spanwise ends of the domain, periodic boundary conditions are imposed. For the inlet boundary shown by red in figure 1 fixed velocity components are prescribed depending on the angle of attack, whereas for the outlet boundary shown by blue in figure 1 zero pressure gradient is imposed. A summary of the 3D meshes used in this study is provided in table 4. Although not shown here for brevity, a further validation study was conducted with a narrower spanwise domain size of 5 chord length, which produced results similar to the wide domain 10 chord length cases. Front and back planes are periodic. The original version of DES had some problems, especially when used with an intermediate mesh resolution near the wall, i. Lift and drag curves. For the numerical schemes, a second order upwind scheme has been used for all variables in both URANS and DDES, with the exception of the convective terms in the DDES momentum equations, where a bounded central differencing scheme is adopted for its lower diffusivity. With regards to the time discretization a second order implicit scheme has been used, and the SIMPLE algorithm [30] is employed for the pressure velocity coupling. For each time step the normalized residuals on all equations have been ensured to fall below , whereas to evaluate the convergence of the unsteady solution, temporal variations of the lift and drag forces have been monitored. Each simulation has been run until a statistically stable solution was kept long enough at least for time steps, which correspond to about 4. Each simulation has been initialized with the entire flow field being equal to the inlet boundary conditions. Instantaneous skin friction lines together with contours of streamwise wall shear stress, computed from the URANS results. Results and discussion 3. URANS We first present the lift and drag curves for all simulations performed, which are shown in figure 3. Also plotted in this figure for comparison are the experimental data reported by Ladson [31]. Hence the spanwise mesh resolution does not affect the flow predicted at these angles of attack. As we increase further the angle of attack, a steep decrease in lift follows together with an abrupt increase in drag, i. This behavior has also been found in experimental measurements [31]; however, here the effects of spanwise mesh resolution on the numerical results are significant. It can be seen that the finer the spanwise resolution, the sharper the drop of the lift, although the two finest meshes N50 and N yield similar results which are also close to the DDES results to be discussed later in Section 3. For the drag curves, it is again possible to identify three distinctive angle-of-attack ranges or branches. As will be described below, the analysis of flow field around the airfoil can explain these three branches observed in the lift and drag curves, i. These flow structures consist of pairs of counter rotating streamwise vortices. As can be seen, the formation of 3D flow patterns coincides with the drop in lift and the rise in drag, in agreement with the findings of Broeren et al. The N20 mesh exhibits a strong sensitivity of the size of stall cells to the angle of attack, presumably due to the coarse spanwise mesh resolution not sufficient to capture the 3D flow physics properly. On the N10 mesh we did not observe any stall cells. This finding is in agreement with Gross et al. Note that the unit of the angle -5 -5 -4 -4 -3 Spanwise-averaged pressure profiles for three angles of attack corresponding to the pre- To have a deeper understanding of how stall cells affect the airfoil performance, the mean static pressure profiles around the airfoil averaged in the spanwise direction are presented in figure 5 left for three different angles of attack, namely On the pressure side of the airfoil, static pressure does not change with the angle of attack, whereas on the suction side the difference between the three cases is evident. Of particular interest is the difference between the The suction predicted on the airfoil upper surface is stronger when the flow is fully separated compared to the case with stall cells, considering the spanwise averaged value. This can be explained further in figure 5 right , which shows pressure profiles around the airfoil at This explains the very low lift coefficient predicted for the case with stall cells. To understand further the predicted performance of the airfoil with stall cells at What is clearly visible is that the mean flow pattern is closer to the separated pattern rather than the attached one, since the separated regions stall cells are much larger than the attached regions gaps between the stall cells. The structures of the stall cells can be seen more clearly in figure 7, which shows contours of instantaneous streamwise velocity and vorticity at several chordwise locations. This method Figure 7: URANS results for The results of this Fourier transform at several chordwise locations have been averaged to obtain the suction side average spectrum, which is presented in figure 8 for four different angles of attack. It is evident that the main peaks identified by the Fourier transform are located in the spatial frequency range of 0. At this particular angle of attack we have initially obtained a quasi-stable 2D solution without stall cells but after more than 20 seconds of simulation time the solution has started changing dramatically to another stable solution with stall cells, as can be seen from the time variations of lift and drag in figure 9. As discussed in [13] the appearance of these types of phenomena seems to be closely related to the stability of flow separated from a smooth body, especially near the stall angle of attack for the case of an airfoil. One possible explanation for the appearance of quasi-stable 2D solution in the current URANS simulation is due to the effect of turbulent viscosity stabilizing the mean flow during the initial stage of the simulation and extending the lifetime of 2D separated flow pattern which is only weakly unstable. Since both lift and drag coefficients obtained are significantly different between the quasi-stable 2D solution and the final 3D solution, a special care needs to be taken when judging the convergence of URANS solutions for this type of flow. As can be seen in figure 4, at Spanwise spectrum of the instantaneous streamwise wall shear stress averaged across the airfoil upper surface. Instantaneous skin friction lines together with contours of streamwise wall shear stress, computed from the DDES results. This is due to the difference in the length scale definition: It is worth noting that the drag predicted by URANS increases suddenly and significantly from the second branch to the third branch, reflecting the sudden change of flow pattern i. However, as shown in figure 10, it is difficult to visually recognize the structures of stall cells in the DDES solutions, since DDES resolves smaller-scale turbulent flow structures as well. Although the flow patterns visualized in figure 10 are rather complex, we can still recognize some locally attached flow regions near the leading edge, similarly to what we observed between stall cells in the URANS results.

Chapter 3 : Study of the Unsteady Flow Features on a Stalled Wing - CORE

Study of the Unsteady Flow Features on a Stalled Wing Steven A. Yon and Joseph Katz*" Department of Aerospace Engineering and Engineering Mechanics.*

American Institute of Aeronautics and Astronautics. Gao1, Hui Hu2, Z. Both 2D and 3D simulations were carried out by solving the unsteady Navier-Stokes equations to predict the behavior of the unsteady flow structures around the airfoils at different angles of attack AOAs. Extensive comparisons were made between the numerical results and wind-tunnel experimental results for the same configurations. It was found that the 2D and 3D simulations differ significantly at relatively high AOAs, and that the 3D computational results agree much better with the experimental data. It is believed that unsteady vortex-dominated flow at high angle of attack is strongly three-dimensional. As a result, the 2D simulations are not adequate in resolving the fundamental flow physics, and 3D simulations are necessary to correctly predict the flow behavior at such conditions. And for particular angles of attack AOAs , the flow will undergo a quick transition to turbulence and reattach to the airfoil surface, which is called a transitional separation bubble [1]. Low Reynolds number flows are often complicated since separation, transition and reattachment occur within a short distance and are said to be dominated by large scale vortex motions [2]. As a result, conventional airfoil designs for higher Reynolds number applications often have poorer performance at low Reynolds number regime [3]. Numerical studies of low Reynolds number flow started with 2D simulation with inviscid-viscous interaction methods [4, 5] and later full Navier-Stokes methods []. The challenge for low Reynolds number simulations is that the assumption that the whole flow field is turbulent is not valid for this flow regime. So normally certain transition prediction process needs to be used to turn on the turbulence model [10]. Another challenge is that standard turbulence models tend to work poorly in low Reynolds number flows. As a result, many special models are proposed and utilized [11, 12]. These studies focused on the time-averaged flow features and were able to obtain good agreements with experimental data up to the stalling AOA. However, most of them failed to work for higher AOAs. Lin and Pauley [2] conducted an unsteady simulation of a 2D airfoil at a low Reynolds number, which revealed that underlying the attached time-averaged flow, there was a successive vortex shedding behavior near the trailing edge. The simulation agreed with Hu et als PIV visualization [13]. In addition, an early incompressible Navier-Stokes N-S simulation was conducted in 2D with vorticity-streamfunction formulation of the N-S equations [14]. However, the post-stall trend is still not correctly predicted in either study. Recently, Tamai et al. In both studies, the unsteady features of the flow were revealed, and the detailed measurement data make a good benchmark for the validation of the present numerical studies. Most previous numerical studies are based on the 2D Navier-Stokes equations. But the question remains open whether the 2D simulation is sufficient to reveal the flow physics that is possibly dominated by large scale 3D structures, since a lot of these numerical simulations failed for high AOA cases. In the present study, both 2D and 3D numerical simulations were carried out and compared with experimental data. Through these comparisons, the validity of the 2D simulations at high AOAs is examined. It was found for high AOA flows, 3D simulations are necessary to correctly predict the flow in the stall regime. All the numerical simulations in this study were mainly performed with the MUSIC code, a 2nd order finite volume flow solver capable of handling arbitrary meshes [16]. Although it was claimed in the PIV tests [13, 15] that turbulence existed in the separation regions and played an indispensable role in the reattachment of laminar separation, the computations in the present study were done with the Navier-Stokes equation without involving any turbulence models. This could be viewed as an implicit large eddy simulation LES simulation, in which the numerical dissipation serves as the sub-grid model. In order to verify some of the numerical simulations, several configurations were also simulated with a high-order spectral difference flow solver [17], and the commercial code Fluent. The use of different solvers in tackling the same configurations was designed to reduce numerical uncertainties involved in the simulations, and served to verify these simulations to some degree. Numerical Uncertainty Elimination Given a perfect flow solver, there are still many uncertainties associated with the numerical simulations. The most well known one is the uncertainty with mesh resolution, which can be assessed with mesh refinement studies. Another important one for external flows is the size of the computational domain. For unsteady flow simulations, the size of the time step or CFL number, and the convergence tolerance used in the time step sub-iteration is also very critical. In order to build confidence in the numerical results, all uncertainties need to be minimized or eliminated if possible. In the present study, a systematic way was employed to minimize the above uncertainties. For example, in order to choose the most economical domain size, numerical simulations were conducted with different domain sizes with other factors being identical. The smaller domain is roughly half of the size of the larger ones. Similar approaches were used to determine the time step, and the convergence tolerance. After that, a set of meshes with different resolutions were generated to use in the grid refinement studies. Some more details are presented next. Boundary Conditions First, the effect of the size of the computational was studied with several mesh configurations, after an appropriate domain size was chosen. Two types of inlet boundary conditions were applied at the inlet fixing all 2 American Institute of Aeronautics and Astronautics inflow variables and characteristic boundary condition or far field , and the results showed that as long as Reynolds number was maintained, the type of boundary condition did not cause noticeable differences. Initial conditions The experimental study on the same configurations showed no hysteresis effects [13, 15], which numerically means the solution should not be dependent on the initial conditions, and thus initial condition here was just taken as freestream. Time integration A second order accurate implicit backward difference scheme was used in time integration. The CFL number used and the inner convergence criteria were also carefully tested to confirm that no significant error was caused by these parameters. If two solvers with two different numerical methods give similar solutions, there is a good chance that both solvers are correctly solving the flow problem. The case with 10 deg AOA was chosen as the base case, where all the numerical verification studies were conducted to eliminate possible numerical errors. A set of meshes with different resolutions coarse mesh: The computed surface Cp distributions were displayed in Fig. It appears the results on the medium mesh agree well with those on the fine mesh, indicating the medium mesh is adequate. Note that also the two different solvers gave similar averaged lift coefficients, as shown in Figure 5. Then the 2D simulations were run for other AOAs on the medium mesh. The results for time-averaged lift coefficients are shown in Fig. As can be seen, the computed lift coefficients agree well with experimental results at lower AOAs under 7 deg , are mildly overestimated at midrange AOAs deg where the flow is characterized by transitional separation bubbles. It is clear that the 2D simulations failed to predict stall at high AOAs above 11 deg. In order to take a closer look at the flow fields, the pressure coefficients on the airfoil surface and the averaged streamlines for AOA of 6 degrees are shown in Figs. Note that at AOA of 6 degrees, the pressure coefficient agrees well with experimental data. For the 9 deg AOA case, both the pressure distribution and average streamline comparison with the experiment suggested that the numerical simulations tended to predict longer separation bubble than that observed in the experiment, and this overprediction caused lift to be over-predicted too. The most interesting cases were those at post-stall AOAs. The computed averaged Cp profile on the airfoil surface is compared with the experimental measurement in Fig. They are completely different! The suction peak near the leading edge remains intact while it has already collapsed from experimental data. The large separation region is nowhere to be seen. Instead, a small separation bubble is observed near the leading edge, as seen from the averaged streamlines shown in Fig. This completely wrong 2D numerical solution seems independent of the mesh resolution and flow solver. Therefore, it is reasonable to conclude that 2D Navier-Stokes model itself is not capable of predicting stall at high AOAs. To further verify this, a 3D numerical study needs to be performed. The current study aims at resolving as much flow details as possible and revealing some of the effects of 3D structures. First, the 16 deg AOA case was studied. When the lift coefficient history was compared with its 2D counterpart, the 3D simulation yielded much closer lift to the experimental average, as shown in Fig. Furthermore, a completely different flow field is obtained with the 3D simulation, as shown in

Fig. The 3D simulation was able to predict a large circulation zone which is very similar to PIV measurements. Therefore, the 3D simulation was capable of predicting the correct flow physics qualitatively even on the coarse mesh, unlike the 2D simulations. The simulations at other AOAs were also performed, and the computed lift coefficients are shown in Fig. The 3D simulations predicted a stall AOA of roughly 10 degrees, while 11 degree is the experimentally observed stall angle. In the two-dimensional simulation, the vorticity was concentrated and located in a few very large vortex structures, and these large-scale vortices propagated in a path that is very close to the upper surface, and were never stationary. Whereas in a 3D simulation, concentrated vorticity only existed near the leading edge, after that, the vorticity is spread in the whole region above the airfoil resulting in a large recirculation region and no concentrated vortices were present. The above observation seemed to suggest that the diffusion of the vorticity is completely different for 2D and 3D simulations. Explanation for this can be found from the vorticity transport equation incompressible below: In a 3D flow field, large vortices would be distorted and even intertwined by the growing three-dimensional disturbance, and would then break down to smaller scale structures in all the directions and be finally diffused to the whole region. This process also delivers energy to the smaller scale motions, and finally causes the transition to turbulence. Therefore, it is speculated that in a real flow, the 3D structures cause the vorticity and the energy to diffuse to smaller scale motions, which form a large low speed area above the airfoil, and this prevents the pressure from building up from the trailing edge and thus cause the suction peak to collapse and the airfoil to stall Fig. While in 2D simulation the successive large vortices shedding behavior build a high speed zone Fig. However, at lower AOAs, as can be seen from Fig. For the mid-range AOAs, both 2D and 3D predicted longer separation bubble, but seemingly for different reasons: As is known, the reattachment is associated with the energy entrainment of smaller scale motions, and the results from 2D simulation is more or less a converged solution, and the lack of smaller scale motion is probably due to its incorrect energy diffusion behavior; while for 3D simulations, it is likely that the restriction of resolution that caused the lack of smaller scale motions, and this lack of ability to reattach is also a possible cause of the earlier separation predicted. In addition to all the above mentioned numerical verification, special attention was also paid to the sharp corners of the airfoil, since ideal sharp corners do not exist in any real case. Both sharp-corner and blunt-corner versions of the mesh were used but the results showed almost no difference. This time, a 3rd order spectral difference solver was used to verify the results. As shown, the two solvers as well as the two meshes produced quite similar lift coefficients history. Not surprisingly, it shows a good agreement with experimental data at lower AOAs, mild overestimation at mid-range AOAs and the missing of stall behavior at higher AOAs. And the large errors in post-stall behaviors should be again because of the incapability of 2D Navier-Stokes simulation. Again, 3D simulation gave better agreement with the experiments, even with a coarser mesh Fig. And when the transient vorticity field was examined in Fig. Conclusion Numerical results seem to indicate that 2D simulations are not adequate for the prediction of unsteady flow structures around the low-Reynolds-number airfoils at relatively high AOA with large scale separations.

Chapter 4 : Spectral content and spatial scales in unsteady rotationally augmented flow fields - IOPscience

The occurrence of large scale structures in the post stall flow over a rectangular wing at high angles of attack was investigated in a small-scale subsonic wind tunnel. Mean and time dependent measurements within the separated flow field suggest the existence of two distinct angle of attack regimes.

Federal Aviation Administration Aircraft icing has been an active area of research for many years; however, the vast majority of research has focused on iced airfoil aerodynamics. We are currently in the early stages of a multiyear program to study the effects of ice on three dimensional swept wings. A swept wing model, representative of a typical commercial airliner wing, has been built and is being tested in the UIUC wind tunnel in order to develop the necessary experimental methods to properly study the effects of ice on swept wings. Methods currently being used include force balance measurements, surface pressure measurements, wake surveys using hot wires and five-hole probes, pressure sensitive paint, surface oil flow visualization and smoke flow visualization. The tests at the larger wind tunnels will be used to create a database of experimental measurements for swept wings with ice accretions at high Reynolds number. In addition to creating this database, we also hope to determine the ice shape features that are critical to the performance of swept wings to aid in future simulations. The aircraft will allow various electrical propulsion system technologies to be tested to determine performance, reliability, safety, and cost. These include various battery, fuel cell, super capacitor, and motor technologies. Additionally, the new aircraft could be used to study energy-harvesting solutions including photovoltaics, vortex energy extraction, and piezoelectrics. The overall technical objective of the Phase I project is to perform a multidisciplinary analysis of the various aircraft available to be purchased and those available at the NASA Dryden Flight Research Center to determine which vehicle would provide the most desirable and versatile testbed for electric aircraft propulsion. This testbed will provide a modular system capable of testing the widest range of components and architectures in a cost effective and timely manner. Envelope Protection System M. Rolling Hills Research Corporation, NASA STTR The objective of this research is to quantify the relation between aircraft stall and aerodynamic performance, with particular interest given to changes in control surface hinge moment. The envelope protection system will be designed in such a fashion to predict stall under clean, iced, damaged, and other flow-inhibited conditions. Thus, this system could potentially provide warnings of unexpected, premature stall to a pilot, who could then take corrective action. Wind tunnel testing at UIUC will be performed to obtain airfoil aerodynamic performance measurements under various conditions of airfoil contamination. Aft Guide Vanes M. Due to current FAA regulations, which limit civilian jets to subsonic speeds, Gulfstream Aerospace Corporation has proposed an engine nacelle design aimed at reducing the sonic boom signature of the propulsion system. The external engine profile, which has a large protuberance due to a gearbox and other engine hardware that contributes greatly to the noise signature, is circularized by wrapping an external cowling around the entire engine. This allows for a more uniform external profile. Tip radial defected flow, originating from the supersonic inlet, is diverted into the nacelle-bypass that is formed between the core turbofan and the new cowling. A fairing is used to divert the nacelle-bypass flow around the gearbox, leading to a highly complex, 3D internal flow. This experiment studies the flow that passes through the aft portion of the nacelle-bypass, where, with the aid of thick guide vanes, the flow is redistributed to a fully annular cross-section and reaccelerated to supersonic conditions. Experimental data comes mainly in the form of static pressures within the model and total pressure measurements at the inlet and exit of the model. Federal Aviation Administration The purpose of this research is to use pressure sensitive paint techniques to visualize pressure distributions and characterize flows over 3-D wings with ice shapes. Pressure sensitive paint provides continuous pressure measurements, which are advantageous when compared to discrete surface pressure taps. The current work focuses on using this method on a 3-D swept wing with a leading edge ice shape and testing this model in the UIUC wind tunnel. These tests will determine the effectiveness of this method for 3-D icing applications at low speed, and will aim to obtain quantitative pressure distribution data for the model. Additionally, this method will enable flow characteristics such as separation bubbles, leading edge vortices, and trailing edge separation to be analyzed visually. The Boeing Company Active control methods, such as servo tabs, have already been implemented on aircraft such as the Boeing and For next generation air vehicles, such as the blended-wing body aircraft, an active-control tab-assisted control surface would be an effective method of reducing flight control power requirements and thermal impact, but would require a more complex method of application than that previously used or studied. The initial wind-tunnel experiment served as a preliminary verification of the power-savings capability of a tab-assisted flap and of a computational fluid dynamic CFD analysis. Furthermore, methods of independent flap and tab movement were analyzed and optimized for continued use in the realistic simulation or dynamic phase of the experiment. The second phase, or dynamic simulation phase, will simulate the proposed mode of operation providing validation of math-model simulations and lead the way to an in-flight application. Forward Guide Vanes M. Due to current FAA regulations, which limit civilian jets to subsonic speeds, Gulfstream Aerospace Corporation has proposed an engine nacelle design aimed at reducing the sonic boom signature of the engine housing. This continuing experimental research focuses on the integration of guide vanes within the bypass region in order to investigate their effect on subsonic annular flow around a circumferential blockage. After upgrading the existing wind tunnel facility to accommodate the design inlet Mach number, experimental flow analysis is aimed at characterizing the complex flow regime, validating computational models, and providing insight into techniques that may improve the flow characteristics within the bypass region. A notable feature of the axisymmetric test section used in this experiment is its ability to rotate, which allows for detailed analysis of both radial and circumferential boundary layer and shear layer profiles. Results of this research will also be used to define inlet boundary conditions for the supersonic bypass flow experiment. Federal Aviation Administration Recent examples of significant airspeed reductions for propeller driven airplanes operating in icing conditions have raised concerns about propeller ice accretion, ice protection and performance degradation. Cases like these and the need to augment current certification guidelines have prompted the FAA to develop methods for better characterizing propeller ice accretion and the resulting performance effects. The objectives of the work being conducted at Illinois are to develop a methodology to analyze propeller performance and to quantify the losses in propeller performance in icing conditions. Initially, degradations in blade section aerodynamic performance due to ice accretion were determined experimentally using artificial ice shapes and data from a propeller icing test. The blade section performance degradation was related to the overall propeller performance degradation using a 2-D strip analysis propeller code. This data will be input into the propeller code and compared with experimentally measured propeller performance. Sonic boom attenuation is a considerable design challenge to enable civilian aircraft to operate at supersonic flight conditions. One technology proposed by Gulfstream Aerospace for the production of low-noise supersonic aircraft is the high-flow nacelle bypass concept in which an outer nacelle surface is used to encircle the asymmetric external engine protuberances of a traditional turbine engine. Although this bypass flow may reduce the overall sonic boom signature of the vehicle, the engine mechanics and mounting create a highly complex 3-D flow in the annular bypass region. Pressure measurements were taken over a range of circumferential angles and radial positions to investigate characteristics of the fundamental steady state annulus flow such as pressure losses and flow uniformity. NASA Glenn Research Center Understanding the relationship between ice accretion geometry and the resulting aerodynamic penalty is important for many applications, including establishing procedures to determine the most critical ice accretion shape for aircraft certification. Research is being conducted under this grant to improve our ability to accurately measure and predict airfoil performance with simulated ice. The presence of the simulated ice causes large regions of unsteady separated flow that make measurement of the aerodynamic performance and computational predictions difficult. First, this study is using detailed measurements of the turbulent wake and a reevaluation of the wake-survey method to improve wake measurement for airfoils with large unsteady wakes. Unsteady pressure measurements and PIV techniques are being used to understand the flowfield over an iced airfoil near stall including the 2-D and 3-D characteristics. NASA Glenn Research Center The objective of this research is to develop accurate aerodynamic simulations of ice accretions for aircraft within a known uncertainty. Currently aircraft certification and other tests use various means to simulate ice accretion, many of which are not well documented. Results from this research will also be used to define the fidelity needed when simulating complex 3-D ice features when performing CFD modeling or computational ice accretion methods. Unsteady Flow about an Iced Airfoil M. NASA Glenn Research Center The prediction of the aerodynamic performance of airfoils and wings with ice accretion is complicated by the unsteady flow that exists near maximum lift. A major feature of this flow is the large separation bubble that occurs aft of the leading-edge ice horn. This experimental study will characterize the unsteady behavior of this bubble using a specially constructed wind tunnel model. High-response pressures and unsteady forces while PIV, surface hot films, and hot wires in the flowfield will provide information on flow reattachment and details of the vortex structures. NASA Glenn Research Center The objective of this research is to use experimental data to determine the relationship and sensitivity of iced-airfoil performance to icing cloud parameters. This study will use ice shape tracings taken from a previous study produced by NASA. Results of this research will also be used to link measurable or significant aircraft safety changes to the underlying changes in airfoil aerodynamic performance. The separated flow regions are studied by both steady and unsteady approaches. Laminar Airfoil Flow Control M. EideticsI Flow control can be used to enhance turbulent boundary layer attachment and enable longer runs of laminar flow on highly optimized low-drag airfoils. Airfoils are designed and tested to develop this capability.

DOWNLOAD PDF STUDY OF THE UNSTEADY FLOW FEATURES ON A STALLED WING

Chapter 5 : Aircraft Icing & Aerodynamics Research Group | Research

Title: Study of the Unsteady Flow Features on a Stalled Wing: Authors: Yon, Steven A.; Katz, Joseph Publication: AIAA Journal, vol. 36, issue 3, pp.

Aerodynamic Optimization of Hyperelliptic Cambered Wings Aerodynamics of Slotted Natural Laminar Flow Airfoil In order to improve the flight efficiency of future commercial transport vehicles, this study seeks to develop and mature a series of slotted, natural laminar flow SNLF airfoil sections. The Aerodynamics and Unsteady Flows Research Group is responsible for providing experimental diagnostics of the SNLF airfoil to understand the airfoil performance and flow physics. Additionally, a new SNLF configuration will be developed that incorporates the benefits of active flow control for low-speed, high-lift applications. Support by NASA is gratefully acknowledged. The fundamental advantage of EMD is its ability to represent nonlinear processes in nonstationary signals. While typically used with point-based measurements or simulation data, the extension of EMD to handle n-Dimensional full-field data has many advantages to understanding complex fluid flow interactions. Furthermore, when used with a multivariate approach, modal content can be represented that retains coherence between flow field variables e. This multivariate, multi-dimensional EMD methodology is currently under development at Illinois. After development, this novel approach to signal analysis will be utilized to understand the nonlinear interactions associated with laminar-turbulent boundary layer transition. The emergence of a turbulence spectrum by way of these nonlinear interactions has long stood as an important problem in fluid flows. Support by ARO is gratefully acknowledged. Return to top of page Flow Physics of the Onset of Dynamic Stall Dynamic stall is a complex phenomenon that can occur in the flowfield of airfoils in unsteady motion. When dynamic stall occurs, the flowfield is dominated by a dynamic stall vortex, which acts to augment the lift and drag of the airfoil. However, there are several aspects about the emergence of this dynamic stall vortex that merit additional study. The current understanding about dynamic stall is focused on the flow features and unsteadiness that occurs at the same time scales as the unsteady motion of the body. This study seeks to understand the unsteadiness in the flowfield, leading up to the dynamic stall vortex, which occurs as sub-scale flow features at much shorter time scales. By understanding these aspects of the dynamic stall flowfield, this study opens up opportunities to develop new methods for dynamic stall control, or introduce significant improvement of existing control methods. Return to top of page Stalled Airfoil Oscillations and Hysteresis Separated flow fields associated with airfoil stall are very complex. Numerous sources of unsteadiness can be present all at once, making it difficult to characterize the physical mechanisms which lead to large-scale oscillations in pressure, velocity, and performance. Previous studies have identified a circulation-driven low-frequency oscillation which can occur in airfoil flows near stall. Recent experiments have revealed that these oscillations can also occur in stalled airfoil flow fields. The purpose of this investigation is to characterize this low-frequency unsteadiness in the flow field about an NACA airfoil, and to understand the physics associated with the unsteady flow field. Return to top of page Closed-Loop Control of Trailing-Edge Separation In general, flow control acts to manipulate an existing flowfield through some type of actuation in order to achieve a more desirable flow state than would occur without forcing. If an active flow control method is used, this forcing can be provided through steady or unsteady actuation. When this actuation is unsteady, there tends to be a wide parametric space of variables that can dictate how the actuation is modulated. For example, for certain actuation methods the frequency, amplitude, duty cycle, and waveform of the forcing can all be varied independently. Active flow control is typically placed into two categories: When open-loop flow control is used, the actuation parameters are provided a priori to the actuation system. When a closed-loop flow control architecture is used, however, measured sensory information of the flowfield is utilized to institute the desired effects on the flowfield. This study seeks to utilize a closed-loop flow control system, comprising of a set of pulsed blowing slots on an airfoil and a series of unsteady pressure transducers, in order to predict what actuation parameters provide the desired control of trailing-edge separation. Results from this study have utilized adaptive modal decomposition techniques in order to produce a system that predicts the necessary blowing amplitude and frequency to achieve a desired lift coefficient on an airfoil through manipulation of the trailing-edge separation. Return to top of page Innovative Flow Control Actuators Turbulent separation remains one of the main limiting factors to the high-lift capabilities of modern aircraft. Active flow control provides a means to mitigate turbulent boundary-layer separation, though many actuation techniques either require complex, heavy infrastructure to be effectively feasible, or are limited to low speeds due to low actuation amplitudes. During this study, a new set of simple flow control actuators are being developed which leverage the formation of natural vortical flow structures to enhance flow mixing and manage turbulent boundary-layer separation. In addition, these devices can be actively deployed, providing on-demand performance with no perceptible cruise penalty, as opposed to passive devices. This study is also being conducted in collaboration with CU Aerospace. Return to top of page Distributed Propulsion Testbed In order to understand the vast potential of a distributed propulsion system, a UAV-scale testbed is currently being developed. This vehicle will represent a modified version of a dynamically-scaled Cessna SR22 aircraft. The aircraft will feature a series of electrical ducted fans distributed across the wing span, which enable significant improvements in propulsive efficiency and vehicle capabilities beyond the baseline aircraft system. This testbed will be utilized in order to understand how a distributed propulsion system can be utilized as control effectors in a fully propulsion-airframe integrated configuration. This study is also being conducted in collaboration with ES Aero. Return to top of page Flow Physics of the Tiltrotor Fountain Effect When a tiltrotor aircraft is situated in a hovering configuration, a complex flow interaction can be produced where the induced velocity across both rotors are deflected inboard along the span of the wing, with a sudden acceleration in the upward direction when these two streams meet. The subsequent jet that is produced can then be re-ingested by the rotors. The detailed flow physics associated with this flow field, however, are not very well understood. This study seeks to quantify the unsteady flow between the rotors using a basic wing and dual-rotor geometry in hover and in a cross-flow. Return to top of page Design, Analysis, and Evaluation of a Novel Propulsive Wing Concept Several recent studies into alternative aircraft configurations and propulsion technologies have shown significant promise. However, outboard of the fuselage or main body of the aircraft, most of these configurations utilize a conventional wing surface. This study is also being conducted in collaboration with Rolling Hills Research Corporation. Return to top of page Sensitivity Analysis of Hybrid-Electric Aircraft Systems In order to meet the ambitious goals in fuel burn reduction in place for the next generation of aircraft, the possibilities provided by hybrid-electric propulsion systems are currently being explored. Traditionally, hybrid-electric systems can be laid out into three system architectures. However, little is understood about how the respective efficiencies of these hybrid-electric architectures change from one aircraft platform, or mission, to another. In order to understand this sensitivity, this study seeks to develop a simulation system for a hybrid-electric aircraft propulsion system that can be used to analyze the various trade-offs associated with various components and hybrid architectures. Return to top of page Analysis of Optimum Wing Spanloads using Multi-Fidelity Methods Several classic investigations have led to an improved understanding of how induced drag is produced and how it can be minimized. Undoubtedly, the most significant of these studies were those conducted by Prandtl and Munk, where the elliptical wing was identified as producing the minimum induced drag for a fixed value of lift and span. Several subsequent studies have also used various methods to determine wing spanloads that minimize induced drag for wings with fixed structural constraints, rather than a fixed span constraint. These efforts span from classic solutions derived from circulation theory to those resulting from the rise of modern multi-disciplinary design optimization methods. However, there currently exists little understanding on how the sensitivity of a minimum drag solution is tied to the limitations of the modeling method. In order to better understand this coupling, the focus of this study will be to develop a series of optimum spanload wing configurations that have been designed using tools of varying fidelity, and performing wind-tunnel experiments on the resulting designs. The result of this study will lead to an improved understanding of how design and analysis tools of varying fidelity can be used for conceptual wing design. Return to top of page Aerodynamic Optimization of Hyperelliptic Cambered Wings The introduction of wingtip devices has allowed for unprecedented improvements in the aerodynamic efficiency of aircraft wings. Various wingtip devices, such as winglets, have now reached common use in commercial aircraft applications. This study seeks to understand how improved aerodynamic performance of a wing can be achieved by blending the non-planar wingtips into a continuous spanwise camber of the wing design. This continuous distribution of spanwise camber can be achieved using a hyperelliptic function, which is then optimized to produce a minimum drag solution under a set of fixed performance constraints.

Chapter 6 : "An Experimental Study of Bio-Inspired Force Generation by Unsteady Flo" by Wesley N. Fass

This study of the separated flow over unswept wings is a result of the cooperation between SDSU and NASA Ames Research Center. The research had elements of wind-tunnel and numerical.

Chapter 7 : Aerodynamics and Unsteady Flows Research Group, University of Illinois at Urbana-Champaig

The structure of Stall Cells (SCs) on wings is analyzed on the basis of stereo particle image velocimetry measurements. All experiments regard a Reynolds number Ã— flow around a rectangular wing with endplates and an aspect ratio of

Chapter 8 : Stall (fluid mechanics) - Wikipedia

Numerical study of airfoil stall cells using a very wide computational domain studies about stall cells is the unsteady flow features on a stalled wing.