NUMERICAL SIMULATION OF THE FLOW AROUND CIRCULATION CONTROL AIRFOILS

K.-C. Pfingsten, C. Jensch, K. W. Körber and R. Radespiel Institute of Fluid Mechanics, Technical University at Braunschweig Bienroder Weg 3, 38106 Braunschweig, Germany

Abstract

Numerical two-dimensional simulations with a RANS flow solver are conducted to assess the aerodynamic sensitivities of a gapless high-lift system. The capability of the numerical flow solver to simulate the flow around a circulation control airfoil is verified by comparing experimental and numerical results for an elliptical airfoil utilising trailing edge blowing. For the investigated high-lift configuration a small fraction of the engine flow is used for circulation control. The air is blown from a slot directly upstream of the flap and thus the flap can bear large adverse pressure gradients without separation. It is found that the use of circulation control yields lift coefficients which are comparable to those generated by conventional highlift systems.

1 Introduction

In recent years noise pollution from aircraft, especially around airports, has become a huge problem. Hence there is an increasing interest in reducing the noise emitted during take off and landing. The conventional high-lift systems, consisting of slats and slotted flaps, are a major contributor of airframe noise. Therefore a gapless high-lift system without slats has a potential of reducing the overall noise, emitted by an aircraft. With active flow control, like trailing edge blowing, a gapless high-lift device is capable of generating the high lift coefficients needed for climb and landing.

For circulation control a small fraction of the cold engine flow (about 5%) is used for blowing. The bleed air is pipelined from the engine to a slot directly upstream of the flap and thus the flow over the flap can bear large adverse pressure gradients without separation. Thus a gapless high-lift device with circulation control is able to generate the required lift.

The low drag coefficients during climbing, achievable with this powered high-lift system, could also allow the use of new low-noise trajectories, which would further reduce the noise exposure at ground level. The absence of slats might allow laminar flow in cruise flight, thereby reducing the drag in this flight segment. Even with taking into account the additional system weight associated with the bleed air distribution for a gapless high-lift system, there is a chance of reducing the total weight of the aircraft and possibly the cost, because slats and fowler flaps are now redundant.

First experiments to understand the principal of circulation control were conducted in the thirties of the last century [1], [2]. At Braunschweig Technical University systematic measurements and theoretical considerations for wings with blown flaps yielded lift increase versus neccesary momentum coefficients [3], [4].

In the seventies two demonstrator aircraft using circulation control to allow for short take off and landing abilities were built in the United States [5], [6] and went through extensive flight testing. Today a wide range of literature concerning circulation control is available [7].

Because of the high costs involved in windtunnel experiments numerical flow simulation can play a major role in the development of an effective circulation control high-lift system. Therefore it is important to investigate if the flow solver is capable of simulating the highly curved flow around a circulation control airfoil correctly. Numerical flow simulations have to prove their ability to predict the location of the jet detachment from the Coanda surface accurately, as it has a significant influence on the generated lift.

The accurate determination of the detachment is particularly difficult for a profile with a round trailing edge, such as that used by Novak [8] in the experimental assessment of an airfoil with trailing edge blowing. Especially for strongly curved flow previously developed adaptions of the standard Spalart-Allmaras turbulence model [9], which incorporate curvature effects on the production of turbulence, have to be used to simulate the flow around these profiles ([10],[11],[12], [13]).

In this paper comparisons are made between computed and experimental pressure distributions, velocity profiles and the position of flow detachment. Furthermore the numerical results are compared to the simulations of Swanson [14] conducted with the same modification of the turbulence model but with a different flow solver. These comparisons are used to assess the capability of the flow solver to simulate the highly curved flow around a profile with circulation control.

In order to address aerodynamic design trades for circulation control applied to a modern transonic airfoil, this paper also presents results of twodimensional flow field simulations around profiles utilising blowing over the flap surface at high-lift and analysis at cruise conditions.

2 Coanda-Effect

Profiles with trailing edge blowing use the well known Coanda principle to generate high lift coefficients: A high velocity, tangentially blown air jet remains attached to a convex surface due to the balance between centrifugal forces and the sub-ambient pressure in the jet sheet. The Coanda-effect works best when the slot height is about 1% to 5% of the curved surface radius and the slot height is between one and two per mil of the chord length [6].

The driving parameter for the Coanda-effect is the dimensionless momentum coefficient c_{μ} of the jet, which is defined as follows:

$$c_{\mu} = \frac{v_{jet} \ \dot{m}_{jet}}{\frac{1}{2} \rho_{\infty} v_{\infty}^2 S}$$

It is important to notice that the increase of the lift coefficient is much higher than the used dimensionless momentum coefficient. The augmentation can be as large as eighty times the applied c_{μ} [7]. So the lift gain is due to flow separation control and supercirculation and does not arise because the momentum of the jet is directed downwards.

3 Numerical Simulation

The flow around profiles utilising trailing edge blowing is simulated by solving the Reynolds-averaged Navier-Stokes equations using the DLR hybrid unstructured flow solver TAU ([15], [16]), which is based on a finite volume scheme. The code processes meshes with different types of cells and combines the advantages of structured grids to resolve boundary layers with the flexible grid generation of unstructured grids. To accelerate the convergence to steady state, techniques like local time stepping, residual smoothing and multigrid technique based on agglomeration of the dualgrid volumes are available. All computations are undertaken assuming the boundary layer to be fully turbulent.

3.1 Turbulence Model

The Spalart-Allmaras (SA) turbulence model [9] is used for all computations. In the standard Spalart-Allmaras turbulence model $\nu_t = f(\tilde{\nu})$ and the turbulence transport equation is given by:

$$\frac{\partial \rho \tilde{\nu}}{\partial t} + \frac{\partial \rho u_i \tilde{\nu}}{\partial x_i} = P + D_{diff} - D_{diss} \; .$$

The terms on the right hand side represent production, gradient diffusion and the wall destruction of the turbulent kinematic viscosity. For the SARC model (SARC: *Spalart-Allmaras* model for Rotation and/or Curvature effects) the production term is multiplied by the rotation function f_{r1} [11]:

$$P = c_{b1}\rho S\tilde{\nu}f_{r1} \quad \text{with}$$

$$f_{r1} = (1 + c_{r1})\frac{2r^*}{1 + r^*} \left(1 - c_{r3}tan^{-1} \left(c_{r2}\tilde{r}\right)\right) - c_{r1} .$$

The nondimensional quantities \tilde{r} and r^* are given by the following formulas:

$$r^* = \frac{S}{\omega} \qquad \text{and} \\ \tilde{r} = 2\omega_{ik}S_{jk} \left[\frac{\mathrm{D}S_{ij}}{\mathrm{D}t} + (\epsilon_{imn}S_{jn} + \epsilon_{jmn}S_{in})\Omega_m \right] \frac{1}{D^4} \\ \text{Where } \epsilon_{jmn} \text{ is the tensor of Levi-Civita, } \Omega_m \text{ represent}$$

where ϵ_{jmn} is the tensor of Levi-Civita, Ω_m represents the system rotation and $D^2 = 0.5 (S^2 + \omega^2)$. DS_{ij}/Dt are the components of the Lagrangian derivative of the strain tensor. Strain rate $S^2 = 2S_{ij}S_{ij}$ and rotation $\omega^2 = 2\omega_{ij}\omega_{ij}$ can be computed with the following equations:

$$S_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \text{ and } \omega_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i} \right)$$

The predetermined empirical coefficients are: $c_{r1} = 1$, $c_{r2} = 12$, $c_{r3} = 1$.

A simplified SARC model (SSARC) which is based upon an unpublished work of Spalart [17] is also implemented in the TAU code. Assuming a steady state flow and small values of \tilde{r} and $(r^* - 1)$ a correlation between r^* and \tilde{r} has been derived by Spalart:

$$\tilde{r} = \frac{2r^{*2} \left(1 - r^{*}\right)}{\left(1 + r^{*2}\right)^{2}} \ .$$

Thus f_{r1} only depends on r^* :

$$f_{r1} = (1 + c_{r1}) \frac{2r^*}{1 + r^*} \left(1 - c_{r3} tan^{-1} \left(2c_{r2} r^{*2} \frac{1 - r^*}{(1 + r^{*2})^2} \right) \right) - c_{r1} .$$

Strictly speaking it is only valid for steady and onedimensional flows where U(y) or $U_{\theta}(r)$.

4 Profile with Blowing over a Round Trailing Edge

A circulation control profile with a round trailing edge (**Figure 1**) was investigated by Novak et al. in a wind tunnel with a closed measurement section [8]. Pressurised air is blown from an internal plenum through a slot on the upper surface directly upstream of the round trailing edge.

4.1 Experiment

The model for the wind tunnel experiments had a chord length of c = 0.38m and a span of b = 0.61m, which results in an aspect ratio of $\lambda = 1.6$. The height of the slot used for blowing was h/c = 0.002. Dimensionless momentum coefficients of $c_{\mu} = 0.03$ and $c_{\mu} = 0.1$ were investigated. The windtunnel experiments were accomplished for $\text{Re} = 1 \cdot 10^6$, Ma = 0.12and a geometric angle of attack of $\alpha_{geo} = 0^{\circ}$. In addition to the surface pressure distribution Novak measured velocity profiles on the Coanda surface by using Laser doppler velocimetry. The measured surface pressure distributions were used to compute the integrated force coefficients in tangential (c_t) and normal (c_n) direction according to the chord of the model. Due to the interaction of the sidewall boundary layer with the model, vortical structures were generated. These three-dimensional structures induced a strong downwash along the span, which reduced the effective angle of attack.

4.2 Comparison of Numerical and Experimental Results

Here the numerical results for the three used turbulence models are compared with the experimental results and the results of the flow simulation conducted by Swanson [14]. Force coefficients in tangential and normal direction as well as the position of the separation are discussed. The position of the separation is indicated by the angle θ_{sep} which is referenced to the beginning of the circular trailing edge on the upper surface.

In **Figure 1** the mesh for the numerical simulation around the profile investigated by Novak is shown. For all computations the total number of points is about 10^5 . The structured area close to the surface has a thickness of 45 cells. In the vicinity of the jet slot the grid for the numerical simulation is clustered to capture the jet behaviour correctly.

The numerical investigations using TAU simulate the flow around the profile at free stream conditions. To account for the wind tunnel influence the effective angles of attack are used as given by the experimenters ($\alpha_{eff} = -2.46^{\circ}$ for $c_{\mu} = 0.03$ and $\alpha_{eff} = -5.86^{\circ}$ for $c_{\mu} = 0.1$) [18]. Free stream values as well as total temperature and total pressure in the plenum are chosen according to the experiments.

Figure 2 shows the measured flow field around the wind tunnel model for two different momentum coefficients ($c_{\mu} = 0.03$ and $c_{\mu} = 0.1$). Separation from the Coanda surface occurs in the experiments at $\theta_{sep} \approx 85^{\circ}$ for $c_{\mu} = 0.03$ and at $\theta_{sep} \approx 115^{\circ}$ for $c_{\mu} = 0.1$. In **Figure 3** the computed flow field using the unmodified SA turbulence model is plotted. For $c_{\mu} = 0.03$ the flow in the simulation separates rather late at $\theta_{sep} = 91^{\circ}$. Also in the flow field for $c_{\mu} = 0.1$ the jet stays attached too long to the Coanda surface. It separates at $\theta_{sep} = 134^{\circ}$. This creates a small separation bubble on the lower surface of the airfoil close to the trailing edge, which does not appear in the experiments. It becomes obvious that this flow cannot be predicted correctly by using the unmodified SA turbulence model

The SARC model yields a good prediction for the position of the separation for both momentum coefficients. In **Figure 4** the flow fields are plotted. The flow separates at $\theta_{sep} = 84^{\circ}$ for $c_{\mu} = 0.03$. This is close to the experimental results. The simulations conducted by Swanson [14] with the SARC model yield a separation at $\theta_{sep} = 90^{\circ}$. For $c_{\mu} = 0.1$ the SARC predicts the separation at $\theta_{sep} = 116^{\circ}$. The simulation given in [14] yields a separation at $\theta_{sep} = 113^{\circ}$ with SARC.

Also the simplified version of SARC allows the flow solver to compute the position of the separation properly (see **Figure 5**), here $\theta_{sep} = 82^{\circ}$ ($c_{\mu} = 0.03$) and $\theta_{sep} = 114^{\circ}$ ($c_{\mu} = 0.1$) respectively.

In Figure 6-9 the computed velocity profiles for $c_{\mu} = 0.03$ are compared to the measured results at different positions on the round trailing edge. Here only the velocity components parallel to the surface are considered. The ordinate d/c is the distance normal to the surface divided by the chord length. The velocity profiles for $c_{\mu} = 0.1$ are displayed in Figure 10-13.

In Figure 6 the velocity directly downstream of the slot is plotted. It can be seen that the computed velocity is slightly slower than the measured velocity. To increase the total pressure in the plenum above the total pressure used in the experiments increases the velocity in the slot to the experimental values, but this also enlarges the dimensionless momentum coefficient by 30%, hence delays the separation and increases the force coefficient in normal direction. Therefore the total pressure in the plenum is chosen to match the value of the experiment.

Swanson increases the total pressure in the plenum to get the correct velocity distribution at the slot position, consequently the computed flow for $c_{\mu} = 0.03$ separates later.

In Figure 8 and Figure 7 the velocity profiles computed with the SARC fits quite well to the experimental data, the differences in Figure 9 arise because the measured flow separates slightly earlier. The same was found in [14].

For $c_{\mu} = 0.1$ the velocity profiles can be seen in **Figure 10-13**. Here the velocity profile in the slot corresponds well to the measured velocity profile. Also the computed velocity profiles at $\theta_{sep} = 80^{\circ}$, $\theta_{sep} = 110^{\circ}$ and $\theta_{sep} = 120^{\circ}$ match the measurements as well as the values computed by Swanson [18]. The TAU code predicts the separation at a slightly larger θ_{sep} (**Table 1**).

In Table 1 force coefficients as well as the posi-

tion of the separation are gathered for all investigated cases using all three turbulence models. The numerical results are compared to the values measured in the experimental investigation by Novak [8] and to the numerical results computed by Swanson [14], [18]. It can be seen that with the unmodified SA turbulence model the predicted θ_{sep} is too high. Thus the computed force coefficients in normal direction are too high as well. The simulations using the SARC predict the position of the separation only with a small offset. However the predicted force coefficients in normal direction are still about 10% to high.

The reason for rather large differences in tangential forces can be found in the pressure distributions. In **Figure 14** the surface pressure distribution of the numerical simulations are compared to the measured pressure on the surface of the wind tunnel model [8] for $c_{\mu} = 0.03$. The numerical simulation with the unmodified SA turbulence model results in a too low pressure along the upper surface, hence the force coefficient in normal direction is too large. For SARC and SSARC the pressure distributions fit much better to the experimental data, here the computed force coefficient in normal direction is only slightly too high. In the simulations the pressure on the Coanda surface is still lower than it was measured in the experiments, therefore the predicted value of the force coefficient in tangential direction is too high. The c_p distributions for $c_{\mu} = 0.1$ are compared in Figure 15. The results of the simulation using SARC show similarly improved agreement to the experimental pressure distribution as for $c_{\mu} = 0.03$.

When SARC is used the flow solver is able to predict the highly curved flow quite well. However the accurate prediction of the tangential and normal force coefficient is still a problem. This shows the general ability of the flow solver TAU to simulate the flow around a circulation control profile with a round trailing edge, where the separation point is not predetermined by a sharp trailing edge.

5 Profile with Blowing over a Flap

For Circulation Control only a small fraction of the cold engine flow (about 5%) is used for blowing. The bleed air is led via a duct from the engine to a slot directly upstream of the flap and thus prevents separation on the flap. Due to the Coanda effect the high velocity air jet stays attached to the convex wall behind the slot. Hence the flow on the upper surface of the flap is able to bear severe adverse pressure gradients. Thus a slatless airfoil can produce high lift coefficients without a gap between the main wing and the flap.

5.1 Geometry

As a starting point for the two-dimensional investigation of circulation control a modern transonic airfoil was chosen. The slot has a height of one per mill of the chord length. The construction of the profile with circulation control has been done in a way to keep the characteristics of the basic profile.

First the x-wise position of the slot is defined, which will also determine the length of the flap. Upstream of this position the original upper surface is used. Downstream of the vertical slot the upper surface of the flap has to be shaped. Thus the slot is positioned directly upstream of the flap.

There are two basic design constraints for the flap geometry. On the one hand the Coanda radius should be as large as possible to enable the Coanda effect, on the other hand the profile should be as well optimised for cruise flight, which means it needs a thin trailing edge like the basic airfoil and in cruise the new profile should be as close as possible to the shape of the basic profile. There are three general options of designing the upper surface of the flap, which are on display in **Figure 16**.

In version A the Coanda radius is hidden in the profile whilst in cruise flight. When the slot is closed for cruise the circulation control profile and the basic airfoil are identical. If the flap is deflected for starting and landing the Coanda radius appears downstream of the slot

In version B the the upper surface downstream of the flap is translated downwards to create the slot. So in cruise the upper surface of the flap is slightly lower and at the position of the slot is a small step. As for version A the Coanda radius is hidden when the flap is not defected. Note that the exact position of the hinge line can be used to obtain continuity in surface slope with version A and B. This feature is not shown in **Figure 16**.

Version C describes a dual-radius flap, which was first mentioned by Englar [19]. Here the upper surface of the flap is composed out of two circle segments. The first circle segment is inside the geometry whilst in cruise. It is always chosen as large as possible, so the radius is about the local thickness of the profile at the slot position as for version A and B. The radius of the second segment is chosen in a way that the position of the trailing edge stays unchanged. As the second segment is from a very large circle, the new shape of the upper surface stays very close to the original contour (**Figure 16**).

A large high-lift flap with a flap length of $c_{flap}/c = 0.3$ is designed according to version B to numerically assess its high-lift characteristics. Furthermore a dual-radius flap (type C) is realised with a flap length of $c_{flap}/c = 0.07$. Both flap geometries can be seen in (Figure 17).

To get a realistic flow at the slot exit a converging

duct is designed which has a length of seven times the height of the slot. At the inflow position the duct is two times as high as the slot to ensure the jet is accelerated until it reaches the exit. The duct can be seen in **Figure 18**.

5.2 Numerical Results

The numerical investigations using TAU simulate the flow around the profile at free stream conditions. Free stream values were chosen according to the standard atmosphere (i.e. 0m for start and 10500m for cruise flight). The total pressure and total temperature of the pressurised air used for circulation control were chosen corresponding to the flow conditions in the exhaust of a modern shrouded high-bypass engine. For take off condition that is $p_{t,jet}/p_{\infty} = 1.68$ and $T_{t,jet}/T_{t,\infty} = 1.63$.

In **Figure 18** the mesh around a circulation control profile with flap is displayed. For all geometries the total number of points is about $0.7 \cdot 10^5$. The structured area on the surface has a thickness of 30 cells. In the vicinity of the jet slot the grid for the numerical simulation is clustered to capture the jet behaviour correctly. Behind the trailing edge and behind the upper boundary of the duct a wake plane with structured cells is added to get a better resolution at these sharp edges.

For the large high-lift flap and a slot height of h/c = 0.001 the momentum coefficient is $c_{\mu} = 0.04$. When the flap is deflected by $\eta = 60^{\circ}$, lift coefficients of up to $C_l = 4.4$ can be achieved. Higher deflection angles shift the angle of maximum lift to smaller values. For a deflection angle of $\eta = 60^{\circ}$ the angle of maximum lift is already negative. By doubling the slot height the generated lift is increased by about 7%. Nevertheless a profile with a small slot height like h/c = 0.001 is preferred compared to a larger slot size, because as a rough estimate the percentage of engine bleed air corresponds to the percentage of overall thrust reduction. It can be seen that the much shorter and lighter dual-radius flap achieves lift coefficients of $C_l = 3.2$ for $c_{\mu} = 0.04$. The lift coefficients for the two flap geometries shown in Figure 17 are compared in Figure 19. In Figure 20 the computed flowfield around a circulation control airfoil with a dual-radius flap is displayed.

Detailed analysis of the airfoil stall with circulation control shows that stall is caused by reversed flow above the flap. Here the Coanda jet still follows the flap contour whereas flow reversal of the wake from the main wing takes place. Note that the implementation of circulation control does not generate higher pitching moments than a modern profile with slat and Fowler flaps.

All results for the investigated large high-lift flap (type B) are also applicable for a flap of type A, because the high-lift characteristics of version A and B are almost identical. The cruise performance of a version A flap is identical to the performance of the basic airfoil, as in cruise the geometries are the same. Thus the improved high-lift performance of the investigated large high-lift flap can be achieved without degrading the cruise performance if a flap of version A would be used. Therefore no numerical simulations to assess the cruise performance of the large high-lift flap are conducted here.

For the dual-radius flap the slot is closed with a straight cap in cruise. The reference profile generates a drag with $c_d = 0.0073$ at its design point (Ma_{∞} = 0.72, $C_l = 0.4$). The dual-radius flap has a drag coefficient of $c_d = 0.0075$ at the design point and thus increases the drag by two drag counts.

These results show the high potential of the circulation control concept in comparison to a conventional as well as other powered high-lift systems.

The results shown in Figure 19 were computed with the standard SA turbulence model. For these applications the separation point is defined by the geometry and the influence of the curvature correction on the lift coefficient is small. The flow computed with the SA, SARC or SSARC model stays attached until it reaches the sharp trailing edge for both configurations. For the profile with the large high-lift flap the lift coefficient is nearly independent of the curvature correction. For the dual-radius flap the lift coefficient is increased by $\Delta c_l = 0.1$ when SARC is used. The eddy viscosity close to the upper surface of the flap is reduced by the adapted turbulence model, but further away from the surface the production term of the turbulence model is amplified. Thus the momentum exchange between the wake of the main airfoil and the wall jet is enhanced. As long as the flow is attached to the flap surface the generated lift is slightly higher than without the curvature correction model.

However, for a final assessment of these results and the capability of the flow solver detailed experimental data of a circulation control profile utilising blowing over a flap with a sharp trailing edge are needed.

6 Conclusion

Steady-state Reynolds-averaged Navier-Stokes calculations of the flow around a circulation control profile with a round trailing edge are compared to experimental results. The standard SA turbulence model does not predict the position of the separation correctly. Using the SARC turbulence model the flow solver is able to predict the position of the detachment quite well. This shows the ability of the flow solver TAU to simulate the flow around a circulation control profile with a round trailing edge for small to medium momentum coefficients. The results indicate that the flow solver can simulate the flow around profiles which blow pressurised air over a flap, as well. Thus steady-state Reynolds-averaged Navier-Stokes calculations are used to assess a supercritical twodimensional circulation control airfoil with two different flap geometries. The simulations show minor or no disadvantages in cruise and the capability of generating high lift coefficients with a gapless high-lift device. This documents the high potential of circulation control as a high-lift system. While these simulations are very encouraging, additional calculations are needed to further define the optimum flap geometry and to determine the minimum total pressure and mass flow rates needed to achieve the necessary lift. Moreover, more experimental data to validate the numerical flow predictions and especially the turbulence models are recommended.

Acknowledgements

The present work was performed as part of the German research project "Innovative Hochauftriebskonfigurationen", which is part of the research programme Lufo III of the BMWA. The authors thank Dr. Swanson of the NASA Langley Research Center, USA, for general discussions of the topic and for making the experimental data he used as a reference available and Dr. Weinman of DLR, German Aerospace Center, Göttingen, Germany, for implementing the curvature correction models in the TAU code and for helpful discussions about the implemented models.

References

- [1] BAMBER, M. J.: Wind tunnel tests on airfoil boundary layer control using a backward-opening slot. NACA Report 385, 1932
- [2] HAGEDORN, H.; RUDEN, P.: Windkanaluntersuchungen an einem Junkers-Doppelflügel mit Ausblaseschlitz am Heck des Hauptflügels. Bericht A 64 der Lilienthal-Gesellschaft für Luftfahrtforschung, 1938
- [3] THOMAS, F.: Untersuchungen über die Grenzschichtbeeinflussung durch Ausblasen zur Erhöhung des Auftriebes, Institut für Strömungsmechanik der TU Braunschweig, dissertation, 1961
- [4] KÖRNER, H. ; LÖHR, R.: Dreikomponentenmessungen am Modell eines leichten STOL-Flugzeuges mit Ausblasen in Flügeltiefenrichtung / Deutsche Forschungsund Versuchsanstalt für Luft- und Raumfahrt. 1975 (DLR-FB 75-74). – technical report
- [5] LOTH, J. L.: Advantages of combining BLC suction with circulation control high-lift generation.
 In: Applications of circulation control technology

Vol. 214. Reston : American Institute of Aeronautics and Astronautics, 2006 (Progress in astronautics and aeronautics), pp. 3–21

- [6] ENGLAR, R. J.; HEMMERLY, R. A.: Design of the circulation control wing STOL demonstrator aircraft. In: *Journal of Aircraft* 18, No. 1 (1981), pp. 51–58
- [7] JOSLIN, R. D. ; JONES, G. S.: Applications of circulation control technology. Progress in Astronautics and Aeronautics Vol. 214, Reston, American Institute of Aeronautics and Astronautics, 2006
- [8] NOVAK, C. J. ; CORNELIUS, K. C. ; ROADS, R. K.: Experimental investigations of the circular wall jet on a circulation control airfoil. AIAA Paper 87-0155, 1987
- [9] SPALART, P. R. ; ALLMARAS, S. R.: A oneequation turbulence model for aerodynamic flows. AIAA 92-0439, 1992
- [10] SPALART, P. R.; SHUR, M.: On the sensitization of turbulence models to rotation and curvature. In: Aerospace Science and Technology No. 4 (1997), pp. 297–302
- [11] SHUR, M. L. ; STRELETS, M. ; TRAVIN, A. K. ; SPALART, P. R.: Turbulence modeling in rotating and curved channels: Assessing the Spalart-Shur correction. In: *AIAA Journal* 38 (2000), Nr. 5, pp. 784–792
- [12] RUMSEY, C. L. ; GATSKI, T. B. ; ANDERSON, W. K. ; NIELSEN, E. J.: Isolating curvature effects in computing wall-bounded turbulent flows. In: *International Journal of Heat and Fluid Flow* 22 (2001), pp. 573–582
- [13] MANI, M.; LADD, J. A.; BOWER, W. W.: Rotation and curvature correction assessment for oneand two-equation turbulence models. In: *Journal* of Aircraft 41, No. 2 (2004), pp. 268–273
- [14] SWANSON, R. C. ; RUMSEY, C. L. ; ANDERS, S. G.: Progress towards computational method for circulation control airfoils. AIAA Paper 2005-0089, 2005
- [15] DLR: Technical Documentation of the DLR TAU-Code / Institut für Aerodynamik und Strömungstechnik. Braunschweig, Göttingen, 2006. – technical report
- [16] GERHOLD, T.: Overview of the hybrid RANS code TAU. In: *MEGAFLOW - Numerical Flow Simulation for Aircraft Design* Vol. 89, Springer-Verlag, 2005 (Notes on Numerical Fluid Mechanics and Multidisciplinary Design), pp. 81–92

- [17] SPALART, P. R.: Evaluation of simplified S-A rotation/curvature correction. Personal Communication, 2006
- [18] SWANSON, R. C. ; RUMSEY, C. L.: Numerical issues for circulation control calculations. AIAA Paper 2006-3008, 2006
- [19] ENGLAR, R. J.; HUSON, G. G.: Development of advanced circulation control wing high-lift airfoils. In: AIAA Journal 21, No. 7 (1984), pp. 476–483

Tables

c_{μ}		Experiment [8]	SA	SA[14]	SARC	SARC[14]	SSARC
0,03	c_n	1.500	1.791	1.921	1.670	1.740	1.616
0,03	c_t	0.037	0.1077	0.1072	0.0997	0.0980	0.0970
0,03	θ_{sep}	$\approx 85^{\circ}$	91°	-	84°	90°	82°
0,10	c_n	3.575	4.740	4.742	4.055	4.028	3.933
0,10	c_t	0.298	0.5800	0.5389	0.4886	0.4573	0.4727
0,10	θ_{sep}	$\approx 115^{\circ}$	134°	-	116°	113°	114°

Table 1: Force coefficients and position of separation

Figures



Figure 1: Mesh for the numerical simulation of the flow around the profile experimentally investigated by Novak [8]



Figure 2: With LDA evaluated streamlines [8]



Figure 3: Numerically simulated streamlines using SA



Figure 4: Numerically simulated streamlines using SARC



Figure 5: Numerically simulated streamlines using SSARC



Figure 6: Velocity profiles $(c_{\mu}=0.03, \theta=0^{\circ})$



Figure 7: Velocity profiles ($c_{\mu} = 0.03, \theta = 50^{\circ}$)



Figure 8: Velocity profiles $(c_{\mu} = 0.03, \theta = 80^{\circ})$



Figure 9: Velocity profiles ($c_{\mu} = 0.03, \theta = 90^{\circ}$)



Figure 10: Velocity profiles $(c_{\mu}=0.1, \theta=0^{\circ})$



Figure 11: Velocity profiles $(c_{\mu}=0.1, \theta=80^{\circ})$



Figure 12: Velocity profiles ($c_{\mu} = 0.1, \theta = 110^{\circ}$)



Figure 13: Velocity profiles $(c_{\mu} = 0.1, \theta = 120^{\circ})$



Figure 14: c_p distribution for $c_{\mu} = 0.03$



Figure 15: c_p distribution for $c_{\mu} = 0.1$



Figure 16: Three general options for the flap geometry



Figure 17: Investigated flap geometries for circulation control



Figure 18: Mesh for circulation control profile with large high-lift flap



Figure 19: c_l over α for different flap geometries (h/c = 0.001): Ma_{∞} = 0.21, Re = 29 · 10⁶, $c_{\mu} = 0.04$



Figure 20: Dual-radius flap (h/c = 0.001): Ma_{∞} = 0.21, Re = 29 · 10⁶, $c_{flap}/c = 0.07$, $\eta = 60^{\circ}$, $\alpha = 9^{\circ}$, $c_{\mu} = 0.04$, $c_l = 3.3$, $c_d = 0.054$, $c_m = 0.48$