AN INNOVATIVE CFD METHODOLOGY FOR EVALUATION OF AIRFOILS AERODYNAMIC CHARACTERISTICS

Francesco Manara, Leonardo S.p.A., Cascina Costa, Italy
Gregorio Frassoldati, Leonardo S.p.A., Università degli Studi Roma Tre
Daniele Malacrida, Leonardo S.p.A., Genova, Italy
Stefan Platzer, Leonardo S.p.A., Munich, Germany

Abstract

The present study aims to define an innovative and reliable CFD methodology for the definition of C81 tables describing aerodynamic properties of airfoils. Nowadays CFD computations are challenging the quality of experimental results, given their increasing reliability for such applications. In the present work Leonardo Helicopters CFD methodology for 2D airfoil simulations was reviewed, updated and validated against literature data available for the SC1095 airfoil at Reynolds number varying from $1 \cdot 10^6$ and $10 \cdot 10^6$ (depending on the Mach number). Different turbulence models were tested, including the innovative Generalized k-ω (GEKO) model. An improved methodology was selected, which was found to predict with satisfying accuracy the main aerodynamic characteristics of the SC1095 at different Mach numbers and angles of attack. The same methodology was then applied to a series of legacy airfoils, used for several Leonardo Helicopters rotors.

1. INTRODUCTION

As stated in [1], a C81 airfoil performance table is a text file that lists coefficients of lift, drag, and pitching moment of an airfoil as functions of angle of attack for a range of Mach numbers. Being kept constant the reference length (namely the airfoil's chord) and the environmental conditions, the Reynolds/Mach ratio is constant. The C81 tables supply aerodynamic 2D data to comprehensive analysis codes which are employed to predict helicopter rotor performance, aeroelastic properties and vibratory loads [2]. For historical reasons, wind tunnel measurements – as opposed to CFD simulations - are typically used, when possible, to characterize aerodynamic features of the airfoils. Nevertheless, it has been already proven that CFD computations are as good as experimental data in predicting many of the aerodynamic performance characteristics [3].

In the present work Leonardo Helicopters CFD methodology for 2D airfoil simulations was reviewed, updated and validated against literature data available in [2] and [4] for the SC1095 airfoil at Reynolds number varying from $1 \cdot 10^6$ and $10 \cdot 10^6$ (depending on the Mach number). Several turbulence models available in *ANSYS Fluent* © were tested, including the innovative Generalized k-ω (GEKO) model [5]. 3D simulations of an extruded wing were performed as well. An improved methodology was selected, which was found to predict with satisfying accuracy the main aerodynamic characteristics of the SC1095 at different Mach numbers and angles of attack. The same methodology was then applied to a series of legacy airfoils, used for several Leonardo Helicopters rotors.

2. PROBLEM DESCRIPTION

2.1. Reference Data

2.1.1. SC1095

In order to validate the CFD results against literature data, the Sikorsky SC1095 airfoil was chosen, based on relevance for helicopter applications and availability of aerodynamic databases.

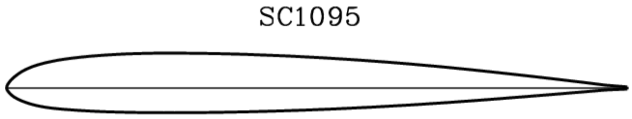

Figure 1: SC1095 airfoil

The aforementioned airfoil, having maximum thickness of 9.5%, was used for the UH-60A "Black Hawk" main rotor, and [2] provides a critical review of the data measured during ten wind tunnel tests from different authors and in different facilities, presenting the global aerodynamic characteristics for each experiment and comparing them. Moreover, for every experiment the authors defined the reliability of the data and stated if it was necessary to correct them before use. [4] was one of the analyzed experiments, it was conducted at the Glenn L.Martin (GLM) wind tunnel at the University of Maryland, and the results were made publicly available by the authors. The results obtained in that experiment were deemed reliable by the reviewer without any corrections on all angles of attack and they were used in this study as a reference for the detailed curves of aerodynamic coefficients vs. angle of attack at low Mach numbers, with chord Reynolds numbers between $1 \cdot 10^6$ and $3 \cdot 10^6$ and Mach numbers up to 0.28. Data from other experimental works at highly subsonic and transonic Mach numbers (in the range 0.5-0.9) were used comparing the summary results trend with the outcomes of this investigation.

2.1.2. Legacy airfoils

In order to further check the CFD results obtained in this work, a family of airfoils currently in use for Leonardo Helicopters rotorcrafts was selected.

Two airfoils of the same family were selected, having respectively 9% and 12% maximum thickness. To protect sensitive data, they will be addressed as Legacy-9 and Legacy-12 in the following, and their comparison of their properties will be shown as trends and in unscaled graphs.

As a reference for those airfoils, experimental data obtained in Aircraft Research Association wind tunnel facility in Bedford (UK) were used. Mach number ranging from 0.2 to 0.7 were analyzed. For Mach numbers 0.4 (only Legacy-9 airfoil), 0.6 and 0.7 an additional set of experimental data was available, which were obtained using strips to force an early transition from laminar to turbulent regime. Out of those datasets, only the drag results obtained with transition strips were retained reliable, therefore the drag data without strips will be omitted in the following.

2.2. Computational Domain

2.2.1. O-grid

The mesh was built using commercial software *Pointwise* ©. The domain used for the SC1095 analysis was a two-dimensional, circular one (so-called O-grid), having a radius of 40 chord lengths, and the mesh was a fully structured, quadrilateral one. For the airfoil discretization, after an analysis of mesh sensitivity, the best compromise between accuracy and computational cost was found at 300 points on each of the two sides of the airfoil, and 10 points on the trailing edge tab (see Figure 2). For each Mach number, the first extrusion step was set in order to have Y+=1 at the first cell center. The resulting meshes totaled ~200k cells.

The boundary condition on the external circular patch representing inlet and outlet was pressure farfield, while adiabatic wall boundary condition was set for the airfoil surface.

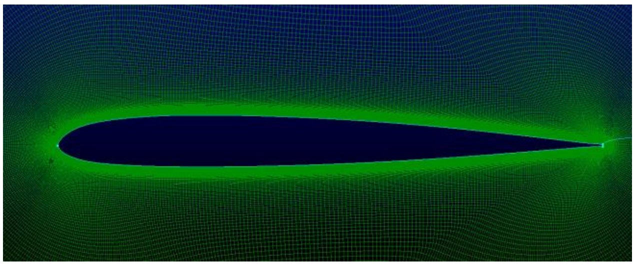

Figure 2: Example of CFD mesh on SC1095 airfoil

A three-dimensional domain was built as well, in order to assess the impact of adding third dimension on flow features predicted by the solver. For those cases, the 2-D mesh was extruded in span-wise direction with up to 400 points, using periodic boundary conditions on side surfaces. Drag, lift and moment coefficients were obtained from integration over the whole wing surface. Moreover, the three-dimensional domain was built in two versions: the first one had the same mesh as the two-dimensional case, targeting Y+=1; the second one had a coarser boundary layer mesh, with the first extrusion step set in order to have Y+ between 30 and 80 at the first cell center, thus exploiting the wall functions capabilities embedded in the turbulence model.

Different angles of attack were simulated imposing the correct direction to the pressure-farfield boundary condition. Simulations were performed with a step of 1 degree of angle of attack, covering all the linear part of the polar, positive and negative stalls and post-stall regions.

2.2.2. Rectangular domain

As a further step to guarantee a higher simulation robustness, a rectangular domain was built, with a circular hole in it. The domain had a length of 40 chord lengths before and after the airfoil. An O-grid having a radius of 20 chord lengths was inscribed in the rectangular domain and they were coupled through non-conformal interfaces. The inner O-grid was built using the same strategy and prism layer law as the one described in the previous section. Moreover, its generation was recorded and reproduced in a Glyph script, in order to be able to reproduce the same kind of mesh on different airfoils. The non-conformal interfaces allowed to rotate the inner grid while keeping the external domain in the same position. This allowed to adopt a pressure-inlet/pressure-outlet boundary conditions scheme, which proved to be more robust than the pressure-farfield one. This permitted to adopt higher under-relaxation factors in the solver scheme and reach convergence with less computational time.

After verifying that the results obtained with the pure O-grid mesh and with the rectangular domain mesh were the same, the legacy airfoils were simulated using the rectangular domain one.

2.3. Numerical Setup

The study mostly involved steady RANS simulations, along with a limited number of URANS simulations.

The instrument of choice to perform the simulations was *Ansys Fluent* and a comparison with *Star-CCM+* © was performed. The coupled pressure-based solver was used, with energy equation always enabled and ideal gas formulation for air. Various 2-equations turbulence models were tested, such as Realizable k-ε, SST k-ω, Generalized k- ω (GEKO) [5]. Along with these, 3-equations and 4-equations formulations for transitional RANS were also tested, without adding a sensible increase in accuracy.

GEKO model is a turbulence model based on k-ω, but formulated in such a way that it has some tunable parameters which are easily customizable by the user to better respond to different flows. In particular, the parameter named C_{sep} is the main parameter for adjusting separation prediction for wall-bounded flows. The default value for C_{sep} is 1.75, which should mimic the separation prediction of the SST k-ω model [5]. Decreasing such parameter, the solver should predict lower levels of separation, and the other way around when it gets increased. In subsequent paragraphs, the model is cited as “GEKO N”, where N is the value set for C_{sep} coefficient (e.g. GEKO 1.0 when C_{sep} was set to 1.0).

2.4. Unsteady Setup

A certain number of 2D URANS simulations were performed, while it would not have added value to exploit Scale-Resolving Simulations in two dimensions. In fact, it does not make sense to use 2D for any other flavor of Scale-Resolving Simulations, as turbulence energy cascade is working backwards in 2D with respect to 3D.

For practical reasons, unsteady simulations were performed only at Mach 0.16 and in 2D. The estimated Strouhal number for a flat plate at the corresponding Reynolds number was 0.22, implying a shedding frequency of approximately 12 Hz and therefore a period of 0.08 s. In order to simulate 10 times the vortex shedding estimated time, a total sampling time of 0.8 s was then chosen.

A time step of $1 \cdot 10^{-3}$ s was selected, with 10 spatial sub-iterations per each time step to ensure convergence. To help unsteady simulation initialization and startup, the RANS result was used as a starting point, and a start-up time of 0.2 s was allowed before sampling.

All unsteady simulations were based on SST k-ω turbulence modeling.

3. RESULTS

3.1. SC1095, Mach < 0.3

3.1.1. Ansys Fluent vs Star CCM+

As shown in Figure 3, for the SC1095 at Mach number 0.16 the Fluent SST k-ω model is accurately predicting the maximum lift coefficient, with an error lower than 1%. Maximum lift is predicted at 14 degrees, with a satisfying accuracy with respect to the experiment, which detected it at 15 degrees. On the other hand, the lift-curve slope is overestimated of about 5%, and lift coefficient at negative stall is more than 10% higher (in module) than the experimental reference. Moreover, peak negative lift is predicted at -13 degrees, while the experiment measured it at -11 degrees.

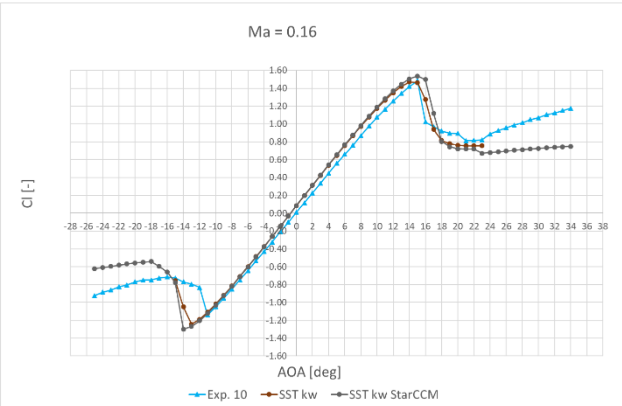

Figure 3: SC1095 Lift Coefficient vs Angle of Attack at Mach 0.16 – Fluent vs StarCCM RANS

When using Star CCM++ with the same turbulence model, the maximum lift coefficient is over-predicted by 3.5%, while the curve slope remains the same as the one obtained with Fluent. The over-prediction of negative lift coefficient is further increased.

Therefore, in the analyzed conditions the two commercial solvers StarCCM and Ansys Fluent are pretty much equivalent for the same turbulence model.

3.1.2. GEKO turbulence model

In Figure 4, drag coefficient curves obtained at Mach 0.16 with SST k-ω and with GEKO turbulence model are displayed. In particular, three different values of GEKO C_{sep} coefficient were tested: 1.00, 1.75, 2.50.

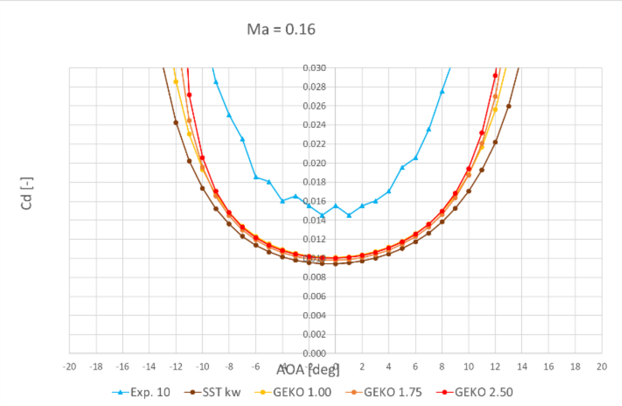

Figure 4: SC1095 Drag Coefficient vs Angle of Attack at Mach 0.16 – SST vs GEKO

Depending on the turbulence model, the minimum drag is simulated between 0.0094 (SST) and 0.0100 (GEKO 2.50). The variation of minimum drag with turbulence model is low. Despite being somewhat scattered, the experimental value for minimum drag is around 30% higher than the CFD values, even if GEKO models slightly reduce the distance from the experimental reference. It has to be underlined that a fully-turbulent setup is adopted in CFD, while in reality laminar flow can be expected for a portion of the airfoil surface, before transition to turbulence. This could explain at least in part the discrepancy between simulation drag coefficient prediction and experimental measurement. Based on the data publicly available from references, it was not possible to better assess the mentioned discrepancy.

In Figure 5, lift coefficient curves obtained at Mach 0.16 with SST k-ω and with GEKO turbulence model are displayed. Accuracy of SST simulation had already been discussed in Paragraph 3.1.

The choice of a different turbulence model has completely no effect on the lift curve slope, while as expected it affects the maximum lift coefficient.

While in [5] C_{sep} 1.75 was established as the coefficient which could tweak the GEKO turbulence model to mimic SST, for this case the maximum and minimum lift coefficients predicted by GEKO model are nearly the same as SST model when C_{sep}=1.00. Anyway, the stall predicted by GEKO 1.00 is smoother than SST, meaning that lift coefficient drops more slowly after peak value.

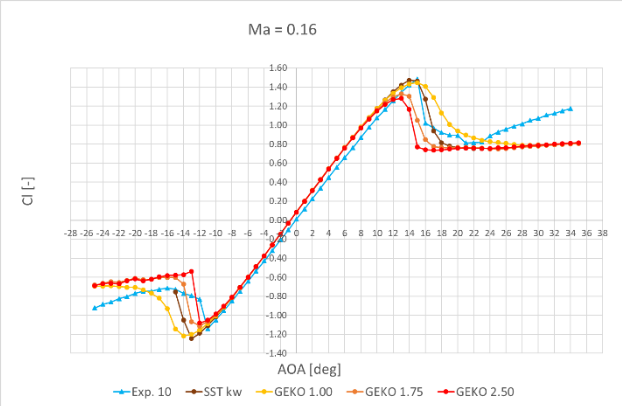

Figure 5: SC1095 Lift Coefficient vs Angle of Attack at Mach 0.16 – SST vs GEKO

As expected, higher values of C_{sep} (1.75 and 2.50) are linked to progressively lower maximum lift and to stall at lower angles of attack. Moreover, stall is more abrupt when C_{sep} value is higher.

To further clarify the effect of C_{sep} coefficient, in Figure 6 and Figure 7 contours of velocity magnitude are shown at Mach 0.16, angle of attack 15 degrees, for GEKO 1.0 and GEKO 2.5 respectively.

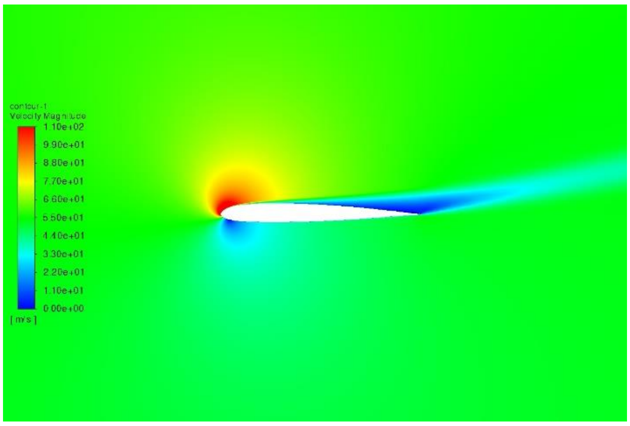

Figure 6: SC1095 Velocity Magnitude Contour At AoA 15deg, Mach 0.16 – GEKO 1.0

In this condition, if C_{sep} is set at 1.0 (Figure 6), a flow detachment is present, which occurs around 50% of the chord. The airfoil is not in the linear part of its alpha sweep anymore, but most of its surface is still generating lift, and the integrated lift coefficient is close to its maximum value. If C_{sep} is increased to 2.5 (Figure 7), the flow detachment is occurring at the leading edge and the whole airfoil is now experiencing a detached flow, not being able to generate as much lift. The turbulence model is hence predicting a deeply stalled condition, and the integrated lift coefficient is almost halved with respect to the GEKO 1.0.

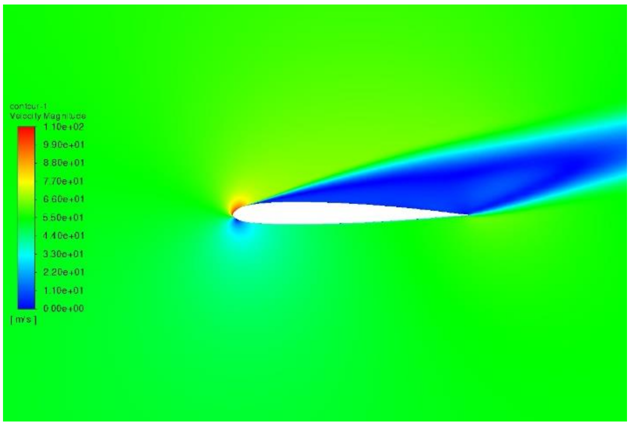

Figure 7: SC1095 Velocity Magnitude Contour At AoA 15deg, Mach 0.16 – GEKO 2.5

In Figure 8, moment coefficient curves obtained at Mach 0.16 with SST k-ω and with GEKO turbulence model are displayed.

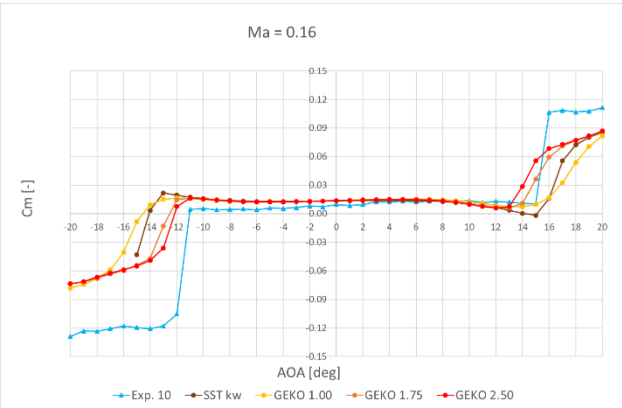

Figure 8: SC1095 Moment Coefficient vs Angle of Attack at Mach 0.16 – SST vs GEKO

All turbulence models seem to under-predict the moment coefficient curve slope in the linear part, and the choice of the turbulence model has very little effect in that range. In addition, CFD over-predicts the moment coefficient at zero lift (at AOA=-1 deg). The impact of turbulence model on moment coefficient is only evident when flow detachment appears, and in that regard moment coefficient curves are coherent with lift coefficient ones.

3.1.3. Unsteady RANS

In Figure 9 the comparison between steady and unsteady RANS modeling is shown.

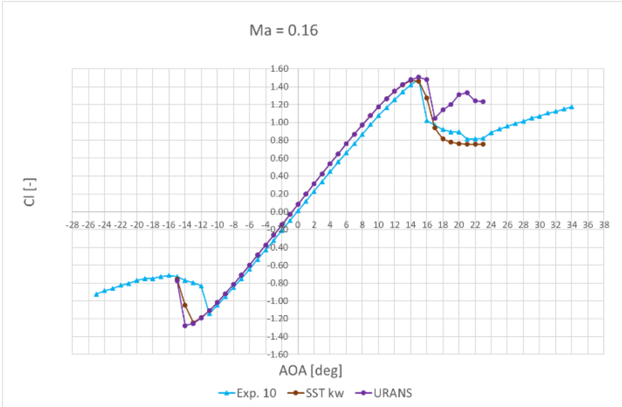

Figure 9: SC1095 Lift Coefficient vs Angle of Attack at Mach 0.16 – Unsteady vs Steady RANS

Despite switching to an unsteady formulation, the lift coefficient curve slope remains the same, while the maximum and minimum lift coefficients are slightly increased (in module). Moreover, the unsteady simulations predict higher lift in the post-stall condition.

Despite a greater computational cost, the unsteady simulations are unable to improve correlations in terms of lift curve slope and negative lift peak value, henceforth they do not provide a clear advantage in accuracy with respect to the steady formulation.

Drag and moment coefficient for unsteady simulations are not shown here for the sake of brevity, as they do not differ significantly from steady simulations results.

3.1.4. 2D vs 3D and Y+=1 vs Wall Functions

In Figure 10 the results of 3D simulations are compared to the 2D simulations.

The curve in gold (SST k-ω 3D) displays the results obtained on the 3D mesh having the same prism layer mesh law as the 2D one.

The difference in results is very little, and limited to near-stall and post-stall regions, where its discrepancy from the 2D reference is somewhat comparable to the standard deviation of a RANS simulation when flow detachment occurs. Drag and moment coefficient curves, which are not shown here, confirm the same conclusion.

On another hand, the results obtained using wall function approach display lower absolute peak values of positive and negative lift, deteriorating significantly the correlation with the experiment for positive angles of attack.

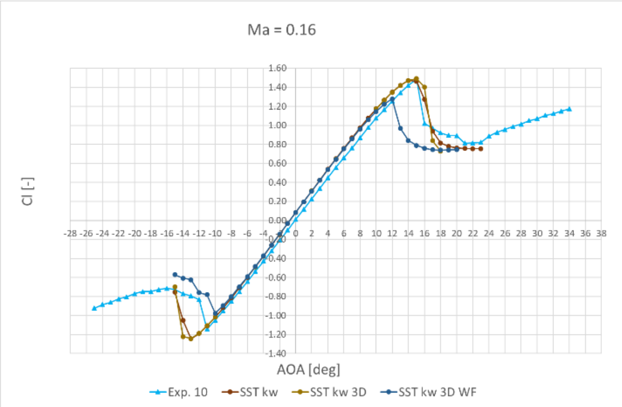

Figure 10: SC1095 Lift Coefficient vs Angle of Attack at Mach 0.16 – 2D RANS vs 3D RANS with different prism layer laws

3.2. SC1095, Mach > 0.3

Even if experimental data were not available as a function of angle of attack, higher Mach numbers were explored with 2D simulations, with the purpose of comparing summary data, such as maximum lift coefficient, stall angle, lift curve slope, minimum drag, etc., against experimental data shown in [3].

3.2.1. Lift curves analysis

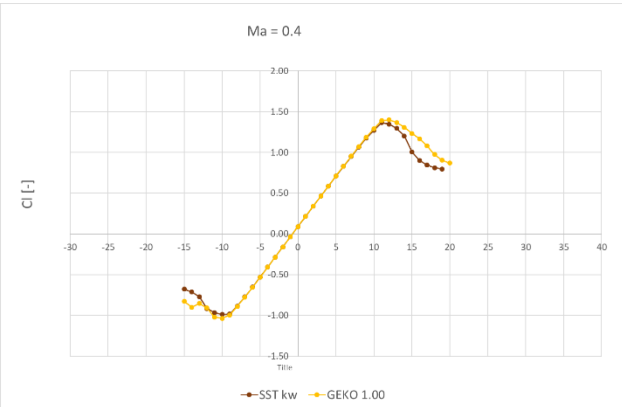

Figure 11: SC1095 Lift Coefficient vs Angle of Attack at Mach 0.4 – SST vs GEKO 1.0

Figure 11, Figure 12, Figure 13 display a comparison of lift coefficient vs angle of attack curves with SST and GEKO turbulence models.

In the same manner as Mach 0.16, at Mach 0.4 the GEKO model with C_{sep}=1.00 is still quite similar to the SST in terms of lift slope and positive and negative stall. At Mach 0.6 instead, a C_{sep}=1.75 is mimicking the SST behaviour, while lower values of C_{sep} lead to considerably higher maximum lift coefficient. This holds true for Mach 0.8 as well.

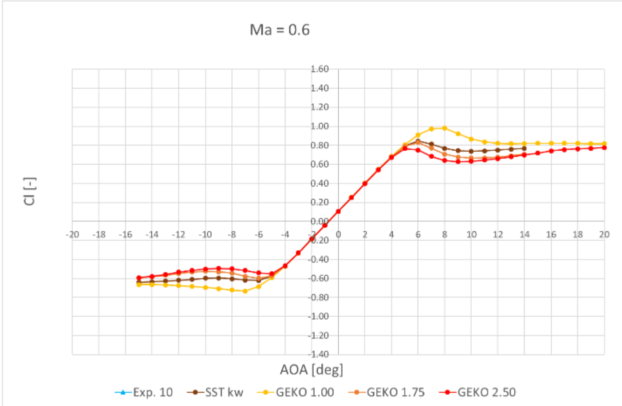

Figure 12: SC1095 Lift Coefficient vs Angle of Attack at Mach 0.6 – SST vs GEKO

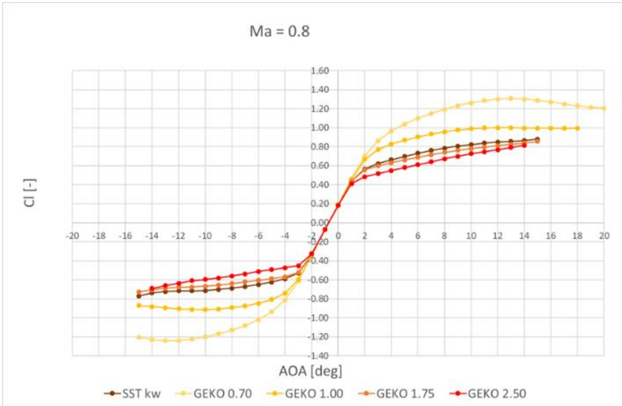

Figure 13: SC1095 Lift Coefficient vs Angle of Attack at Mach 0.8 – SST vs GEKO

3.2.2. Analysis of aggregated data

In the following, data aggregated over multiple Mach numbers are displayed. This allows to compare simulation results against data reported in [2]. Among all the experimental data available in the reference, it was chosen to report here only the ones that in the review were deemed as reliable without any corrections.

Figure 14 displays the maximum lift coefficient as a function of Mach number. Please note that for high Mach numbers (>= 0.8) the maximum lift coefficient is selected as the one at which the lift-curve slope ceases to be constant.

As already seen, at Mach 0.16 SST k-ω and GEKO 1.00 models are the ones who better predict the maximum lift.

At Mach 0.4 both those models overestimate the maximum lift, but SST is closer to the experimental value. This trend is reversed at Mach 0.6, where GEKO 0.70 and GEKO 1.00 are in better agreement with the experiment. At Mach 0.8 SST, GEKO 1.75 and GEKO 2.50 are the best turbulence models, while lower values of C_{sep} lead to a large over-prediction of maximum lift.

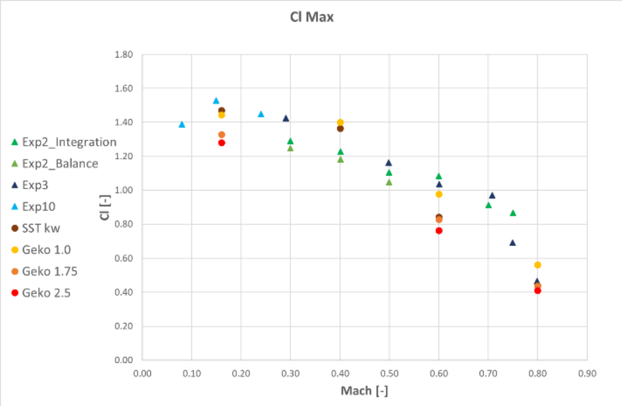

Figure 14: SC1095 Maximum lift coefficient vs Mach number

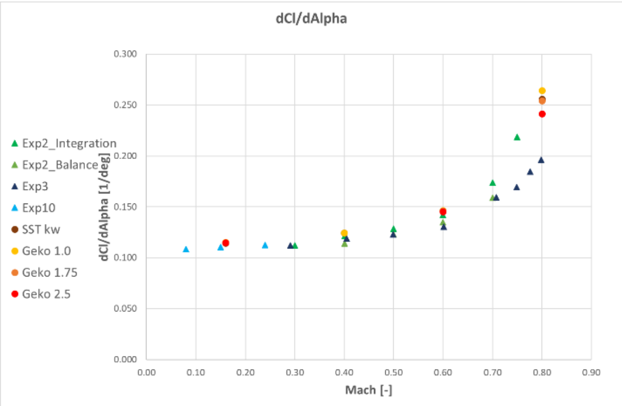

Figure 15 SC1095 Lift curve slope vs Mach number

Figure 15 displays the lift curve slope as a function of Mach number. It can be noted that up to at least Mach 0.6, the turbulence model has negligible effect on this parameter. Depending on Mach number, an overestimation of lift curve slope between 5% and 10% is present.

Figure 16 displays the minimum drag coefficient as a function of Mach number. As already shown in Figure 4, minimum drag coefficient at low Mach number seems to be underestimated by CFD with respect to experimental data. The trend is reversed at Mach 0.4, while at higher Mach numbers experimental data become more scattered.

Figure 17 and Figure 18 display respectively the zero-lift moment coefficient and moment curve slope minimum drag coefficient against undisturbed flow Mach number.

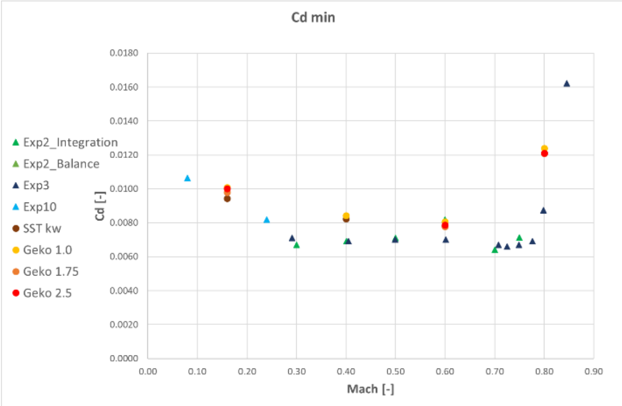

Figure 16: SC1095 Minimum Drag Coefficient vs Mach number

The zero-lift moment coefficient is over-predicted by CFD, while predicted moment curve slope is in between the different experimental analyses. For both those parameters, the impact of the turbulence model is practically negligible.

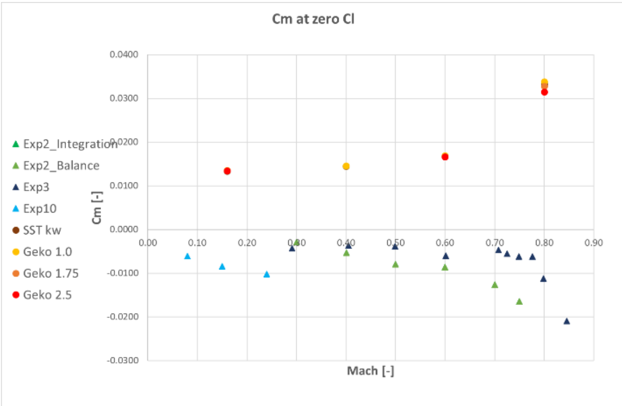

Figure 17: SC1095 Zero-Lift Moment Coefficient vs Mach number

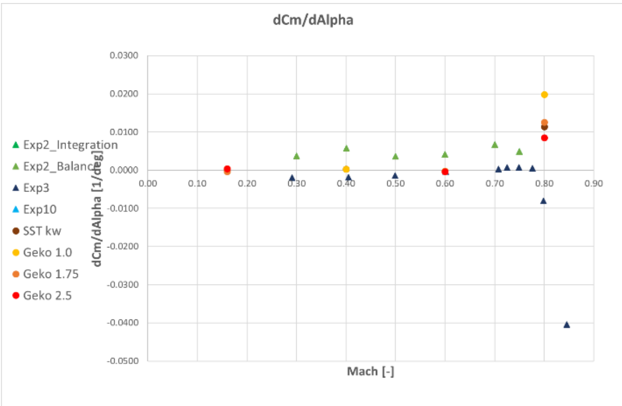

Figure 18: SC1095 Moment Curve slope vs Mach number

3.2.3. Proposed turbulence model

Based on the analysis performed on SC1095 airfoil, it appeared that not all the Mach numbers could be accurately predicted using the same turbulence model. For that reason, a scheme of turbulence model choice as a function of Mach number was proposed, and it is shown in Table 1.

Table 1: Turbulence model choice

Mach number	Turbulence model
Ma <= 0.4	SST k-ω
0.4 < Ma < 0.7	GEKO 1.0
Ma >= 0.7	SST k-ω

The methodological setup was then applied to Legacy-9 and Legacy-12 airfoils, in order to verify its accuracy on a different kind of airfoil.

3.3. "Legacy-9" Airfoil

In Figure 19, and Figure 20 the results obtained on the 9% thick legacy airfoil with SST k-ω turbulence model at Mach 0.2 are compared with experimental data in terms of lift and moment coefficients respectively.

The maximum lift coefficient is accurately predicted by CFD (with an error close to 1% with respect to WT data), even if the lift curve slope is overestimated and the stall angle of attack is not correctly predicted. Negative lift peak is underestimated as well.

Moment coefficient slope is underestimated, as already observed on SC1095 airfoil.

At Mach 0.4 the lift coefficient curve CFD prediction displays the same level of accuracy already noted at Mach 0.2, with accurate maximum lift and overestimated lift curve slope, as showed in Figure 21.

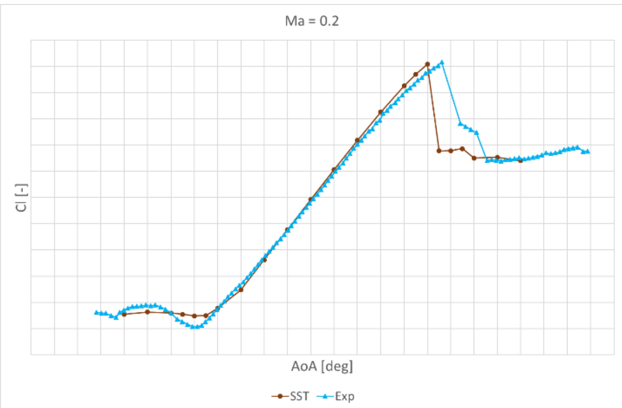

Figure 19: Legacy-9 Lift Coefficient vs Angle of Attack at Mach 0.2

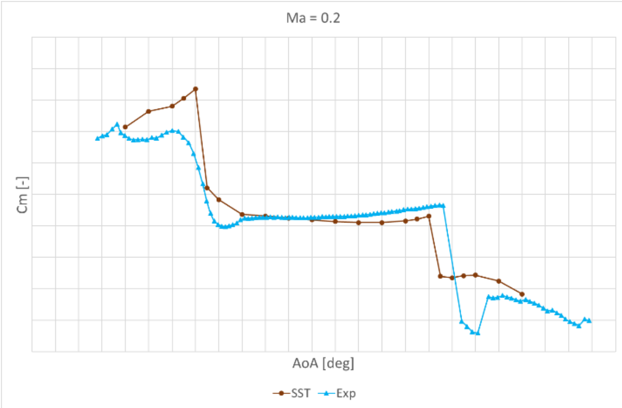

Figure 20: Legacy-9 Moment Coefficient vs Angle of Attack at Mach 0.2

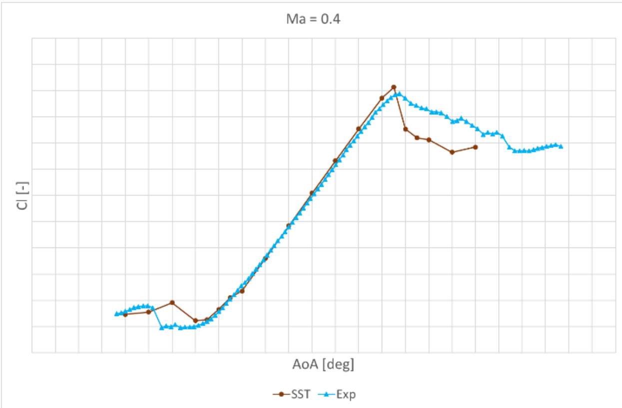

Figure 21: Legacy-9 Lift Coefficient vs Angle of Attack at Mach 0.4

At Mach 0.6 GEKO 1.0 appeared to be the most accurate turbulence model on SC1095 analysis, therefore it was selected for Legacy airfoils as well.

As can be noted in Figure 22, the CFD is accurate enough in predicting positive and negative stall angles, as well as peak lift values, even if the curve slope is still overestimated.

In Figure 23 it is interesting to note that the CFD estimation of minimum drag coefficient is around 30% higher than the one obtained with transition strips (green curve). Minimum drag overestimation at high Mach number was already observed on SC1095 airfoil (see Figure 16).

Despite the great accuracy in lift prediction, the moment coefficient curve slope is still underestimated by CFD (Figure 24).

At Mach 0.7, both GEKO 1.0 and SST model were tested. As shown in Figure 25, at this Mach number SST turbulence model is representing lift peaks in a more accurate way and it should be selected for use.

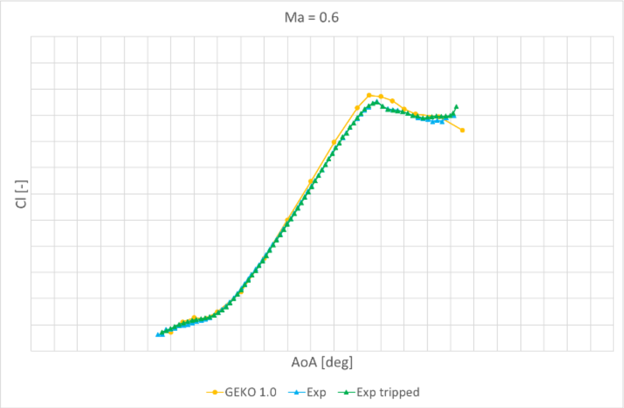

Figure 22: Legacy-9 Lift Coefficient vs Angle of Attack at Mach 0.6

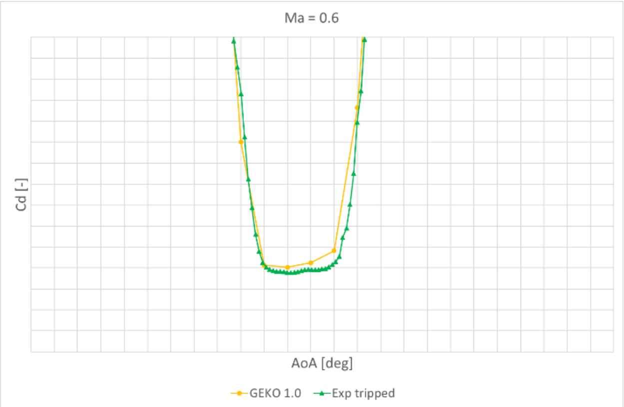

Figure 23: Legacy-9 Drag Coefficient vs Angle of Attack at Mach 0.6

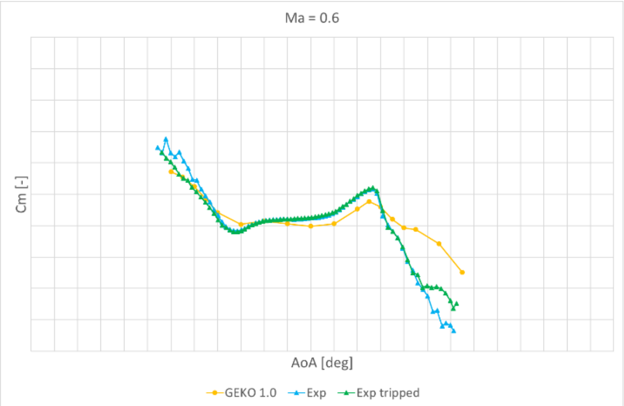

Figure 24: Legacy-9 Moment Coefficient vs Angle of Attack at Mach 0.6

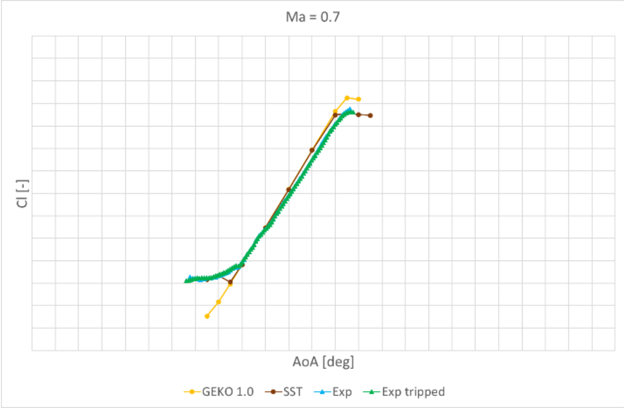

Figure 25: Legacy-9 Lift Coefficient vs Angle of Attack at Mach 0.7

3.4. "Legacy-12" Airfoil

In Figure 26 and Figure 27 the results obtained on the 12% thick legacy airfoil with SST k-ω turbulence model at Mach 0.2 are compared with experimental data in terms of lift and moment coefficients respectively.

In this case, the CFD is over-predicting both stall angle (by 3 degrees) and maximum lift coefficient (by around 13%). This appears as a noticeable difference from the SC1095 reference case and from the legacy airfoil having 9% thickness.

Negative lift peak is under-predicted, while moment-coefficient curve slope is under-predicted, as it happened for previously analyzed airfoils.

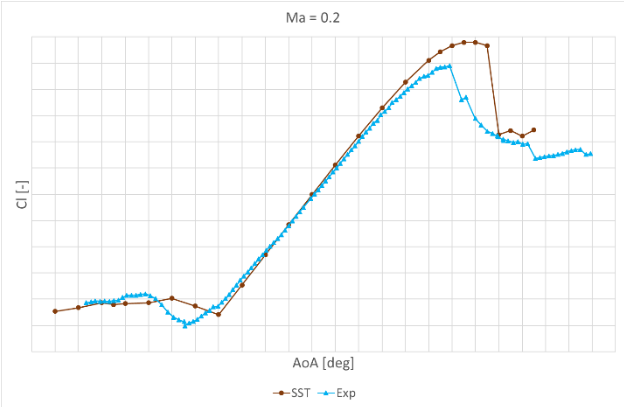

Figure 26: Legacy-12 Lift Coefficient vs Angle of Attack at Mach 0.2

At Mach 0.4 (Figure 28), the maximum lift is still over-predicted by around 8%, even if the stall angle is more in line with the experimental reference. The minimum drag coefficient is overestimated by around 6% with respect to the experiment with transition strips (see Figure 29).

As already noted for Legacy-9 profile at Mach number 0.6, in Figure 29 it is easily seen that at Mach 0.4 the CFD drag prediction is quite in line with experimental values obtained with transition strips.

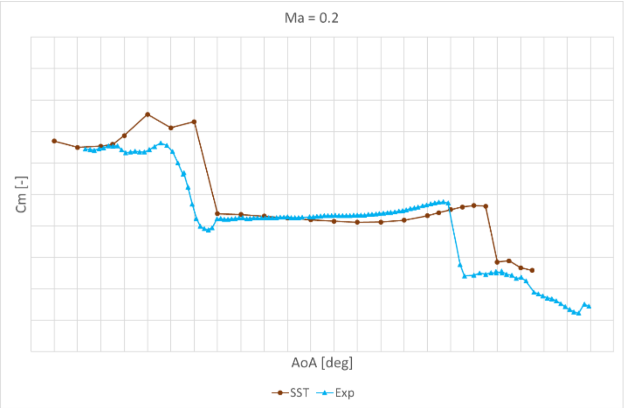

Figure 27: Legacy-12 Moment Coefficient vs Angle of Attack at Mach 0.2

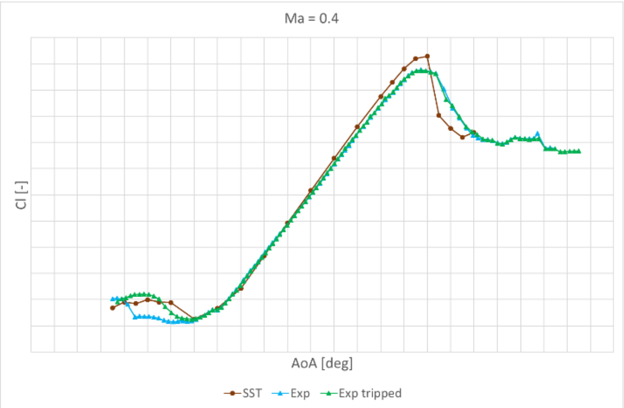

Figure 28: Legacy-12 Lift Coefficient vs Angle of Attack at Mach 0.4

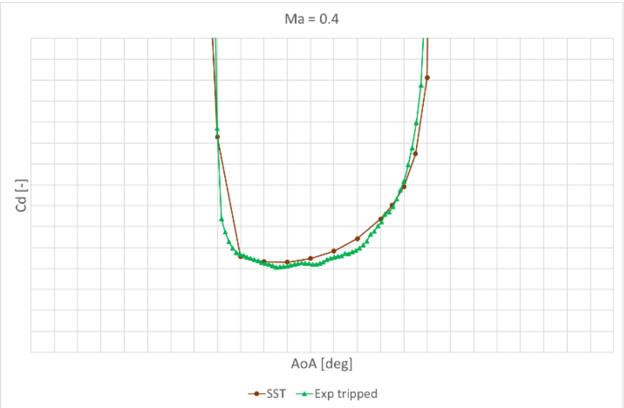

Figure 29: Legacy-12 Drag Coefficient vs Angle of Attack at Mach 0.4

At Mach 0.6, GEKO 1.0 turbulence model is confirmed to be capable of predicting lift coefficients peaks, even if a slight overestimation of maximum one still exists (see Figure 30).

At Mach 0.7 (Figure 31), differently from what observed on 9% thick airfoils, it looks like GEKO 1.0 is predicting maximum lift coefficient in a more accurate way than the SST turbulence model. Anyway, GEKO 1.0 fails to accurately predict negative stall angle.

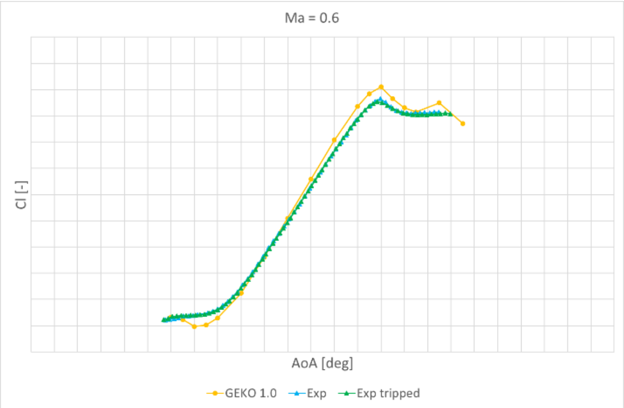

Figure 30: Legacy-12 Lift Coefficient vs Angle of Attack at Mach 0.6

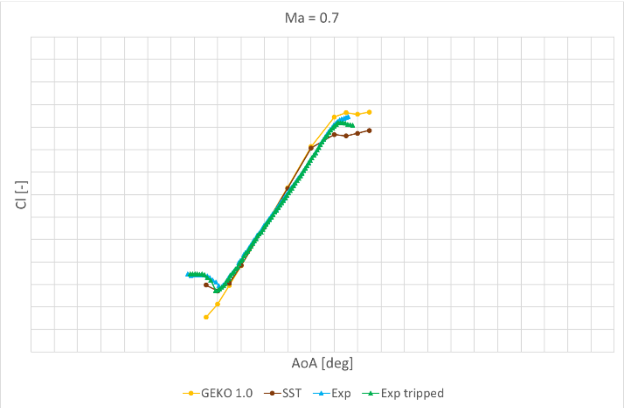

Figure 31: Legacy-12 Lift Coefficient vs Angle of Attack at Mach 0.7

4. CONCLUSIONS AND FUTURE WORK

The present study allowed to draw the following conclusions regarding prediction of airfoils aerodynamic characteristics:

1. A prism layer mesh allowing to fully resolve the boundary layer up to Y+=1 distance from the wall is mandatory. An approach based on wall functions presents the risk of altering the most significant characteristics of aerodynamic coefficient curves for this application.

2. Adding the third dimension (spanwise) to the computational domain does not improve the quality of the solution if RANS models are used.

3 In a two-dimensional computational setup, URANS do not represent a significant improvement over steady RANS in terms of aerodynamic coefficients prediction.

4. Prism layer mesh, turbulence model, two or three-dimensional domain, time-dependence, all have almost negligible effect on lift and moment coefficients slope against angle of attack.

5. From detailed analyses of the CFD simulations against experimental references of an airfoil having maximum thickness of 9.5%, it can be concluded that there is not a single turbulence model satisfactory for all conditions. Depending on the undisturbed flow Mach number, SST or GEKO models are the most suitable option to accurately predict peak values of Cl and positive and negative stall angles. For this reason, it is recommended to adopt a different turbulence model depending on the Mach number, according to Table 1.

6. Regardless of the turbulence model and solution strategy, an overestimation of lift curve slope between 5% and 10% with respect to experimental data should be expected, depending on Mach number.

7. Regardless of the turbulence model and solution strategy, an underestimation of moment curve slope with respect to experimental data should be expected.

8. A maximum overestimation of peak lift coefficient with respect to experimental data of 13% was observed for an airfoil having a maximum thickness of 12%, significantly higher than the reference one. The extent of maximum lift over-estimation appears higher at low Mach numbers.

9. Depending on Mach number, a value of C_{sep} in the range from 1.00 to 1.75 would allow GEKO turbulence model to match the stall behavior of SST k-ω turbulence model.

The new C81 tables obtained for legacy Leonardo profiles with the updated CFD methodology have been released internally to the Department and they are currently being used for comprehensive analysis on legacy rotors. The identified methodology itself is now considered a standard for the production of new C81 tables in the aerodynamic office.

It has to be noted that the discrepancy between CFD and experimental references is hardly explained with sole numerical modeling error, especially for what concerns lift coefficient slope in the linear part. This is also underlined by the small effect on this feature of any change in the numerical modeling.

For the time being, several actions are currently in place to corroborate the obtained results. In order to extend the applicability of CFD validation for airfoils characterization, the range of simulations are being extended from steady and unsteady RANS to three-dimensional Scale-Resolving Simulations, in Fluent formulations such as SAS and SBES. At a higher computational cost, the objective is to explore those regions where the present model in unsuited for accurate predictions, extending the methodology the furthest possible. An ideal hybrid method, with educated corrections of the raw 2D-obtained database, could represent the best possible compromise between accuracy and computational time.

In the future, different tools suited for large scale 3D CFD simulations and domain discretization could also take place into the comparison. This can help identify the most accurate settings for finite volumes simulations for capturing massive separations, a feature directly transferrable onto full airframe simulations.

Author contact:

Francesco Manara

francesco.manara@leonardo.com

Gregorio Frassoldati

gregorio.frassoldati@leonardo.com

Daniele Malacrida

daniele.malacrida.ext@leonardo.com

Stefan Platzer

stefan.platzer@leonardo.com

5. REFERENCES

[1] R. Strawn, E. Q. Mayda and C. P. vanDam, "Automated CFD for Generation of Airfoil Performance Tables," NASA Tech Briefs, 2013.

[2] W. G. Bousman, "Aerodynamic Characteristics of SC1095 and SC1094 R8 Airfoils," NASA/TP–2003-212265, 2003.

[3] M. J. Smith, T.-C. Wong, M. Potsdam, J. Baeder and S. Phanse, "Evaluation of CFD to Determine Two-Dimensional Airfoil Characteristics for Rotorcraft Applications," in *60th American Helicopter Society Annual Forum*, Baltimore, MD, 2004.

[4] K. W. R. a. J. G. Leishman, "Effects of Ballistic Damage on the Aerodynamics of Helicopter Rotor Airfoils," *JOURNAL OF AIRCRAFT,* pp. 695-703, 1998.

[5] F. Menter, R. Lechner and A. Matyushenko, "Best Practice: Generalized k-omega (GEKO) Two-Equation Turbulence Modeling in Ansys CFD," fluidcodes.com, 2019.