An accurate RANS-based transition prediction approach (part II)

S Russo¹, N Natale¹, V Citro², F Giannetti²

¹ Dream Innovation, via F. Parri 1 81030 Sant’Arpino (Caserta) – Italy
² University of Salerno, Via Giovanni Paolo II, 132 - 84084 Fisciano (SA) – Italy

serena.russo@dreaminnovation.it

Abstract. We present a CFD analysis of a wind tunnel model of the new laminar turbo-prop aircraft designed in the frame of the European Clean Sky 2 programme. The work presents the numerical results of the aerodynamic performance of a scaled model of the innovative laminar turboprop A/C wing by using a CFD approach that includes the transition prediction. Numerical results will be compared by considering two different procedures: a) CFD analyses in which the transition location is imposed and b) CFD analyses in which the position of the transition is evaluated. The prediction of the transition location and the laminar flow extension is based on a numerical framework based on the coupling between a high-fidelity RANS tool and a Linear Stability (LNS) solver. According to this method, boundary layer equations are written in the conical formulation and the solution of RANS equations and transition onset is obtained through an eN method based on the LNS calculations.

1. Introduction

The prediction of the transition represents a demanding design issue in the framework of laminar-turbulent flows. This aspect is very important because transition influences friction drag, leading edge separation and boundary layer thickness, the latter impacting upon other key features such as shockwave position and associated wave drag in transonic flows. Although the overall lift value may be predicted with satisfactory accuracy, slight deviations between the real and the computed pressures can lead to large errors in the computed overall drag value. This issue was investigated in detail in [1] and it was shown that the overall pressure drag of a high–lift configuration, which dominates the drag value of the configuration as a whole as well as the drag of every single element, is composed of a balance of very large positive and negative contributions. One single element may be one order of magnitude larger than the resulting overall drag of the complete configuration. Thus, a relative error of 5% drag on the slat upper side may result in a change of 50% for the overall drag value [2]. For the design process of wings in industry, there exists the demand for a RANS-based computational fluid dynamics (CFD) tool that handles automatically and autonomously transitional flows, i.e. able to handle flows that experience transition from laminar to turbulent conditions. Existing transition prediction methods vary from empirical transition criteria via the local [3], linear stability equations based on small disturbance theory or non-local, linear [4][5] and non-local, non-linear stability methods using the parabolized stability equations over large eddy simulations to direct numerical simulations of the Navier--Stokes equations. Empirical transition criteria and the eN-method [6][7] based on local, linear stability theory and the parallel flow assumption represent state-of-the-art methods for the prediction of transition onset in many
industrial applications [8]. These e\textsuperscript{N}-methods are used in aircraft industry most frequently for design purposes covering transition due to Tollmien--Schlichting (TS) and cross flow (CF) instabilities.

The present work has been carried out within ESTRO Clean Sky 2 CfP project (grant agreement No 831809, [9]), in the framework of the Innovative Aircraft Demonstrator Platforms (IADP) “Regional” of the “Clean Sky” 2 Programme, whose goal was to develop an innovative future green regional aircraft configuration based on several new technologies able to match the very demanding and challenging objectives ACARE 2020. The objective of the project is to produce experimental and numerical data in low flow speed and in “cruise conditions” to validate the relevant aerodynamic performance of the Regional 90 seats turboprop A/C wing including laminar flow extension measurements and wing span load distribution. The experimental campaign will be performed in the DNW LLF Wind tunnel [10]. This work presents the numerical results of the aerodynamic performance of the scaled model of the innovative laminar turboprop A/C wing by using a CFD approach that includes the transition prediction. Numerical results will be compared by considering two different procedures: a) CFD analyses in which the transition location is imposed at 5% and b) CFD analyses in which the position of the transition is evaluated.

![Figure 1 – CLEAN-AWL (a) and CLEAN-IWT (b) configurations](image)

2. Description of the geometry

The CFD activity described in this work is dedicated to the numerical characterization of the aerodynamic performances of the following configurations of wind-tunnel model, i.e.

- Clean configuration with an advanced winglet (CLEAN-AWL);
- Clean configuration with an innovative wingtip (CLEAN-IWT).

All the configurations are depicted in Figure 1. From the comparison between CLEAN-AWL and CLEAN-IWT configurations, the main difference is represented by the wing tip: in details the CLEAN-AWL configuration is characterized by the presence of a winglet.

3. Linear Stability Analysis procedure

The approach proposed in this paper to predict the transition location and the laminar flow extension is based on a numerical framework based on the coupling between a high-fidelity, Reynolds–Averaged
Navier–Stokes (RANS) tool and Linear Stability Equations [2][5]. According to this method, boundary layer equations are written in the conical formulation [11] and the solution of RANS equations and transition onset is obtained through an eN-method based on the PSE calculations. The assessment of the tools used for the evaluation of the laminar flow extension and wing deformation is detailed in [12] and has been performed by using the results produced by ETRIOLLA project (see for details [13]). The evaluation of laminar flow extension and the prediction of laminar/turbulent transition is based on the use of an in-house numerical code that solves the linear stability problem over a CFD solution. Figure 2 shows a flowchart of the numerical procedure, in which the following numerical codes are used sequentially, i.e.

- KC-BLC (Kaups Cebeci Boundary Layer Code): Numerical Code for the solution of the compressible laminar boundary layer equations for swept and tapered wings;
- LISA-C (Linear Stability Analysis Code): Numerical Code for the Linear Stability Analysis.

The starting point is represented by the CFD solution of the flow field based on the use of RANS equations. From the distribution of the surface pressure coefficient, one section is extracted: this represents the input for the KC-BLC code. The extracted pressure distribution is divided into an upper and lower part over the wing section, by considering the stagnation point as initial point for both of them. The output of KC-BLC is represented by the boundary layer profiles along the upper and lower surfaces of the wing section. The data are transferred to LISAC, which calculates the N factor trend as a function of the normalized cord coordinate x/c for each frequency of the disturbance considered.

![Flowchart of the numerical procedure](image)

**Figure 2 – Flowchart of the numerical procedure**

4. Assessment of CFD tools

This section is devoted to the assessment of CFD tools through a grid convergence study and the comparison with available numerical data. The solutions are obtained by considering a fluid domain that has at least a dimension of 30MAC (Mean Aerodynamic Chord) in the upstream, downstream, upper and lower directions. Two different types of grids have been generated

- A **hybrid grid** for the CFD analyses in which the transition location is imposed at 5% (see Figure 3(a) for the surface grid).
  The hybrid grid has both structured (hexahedral) and unstructured (pyramidal and tetrahedral) cells: the structured grid is used to capture the gradients and resolve the boundary layer near the surface of the aircraft model and of the strut, while the rest of the domain has a mixture of unstructured grid blocks.
- A **fully-structured** grid for the CFD analyses in which the position of the transition is evaluated (see Figure 3(b) for the surface grid).
  The structured grid on the body is characterized by 70 points at the leading edge and 105 points for suction and pressure sides.

For both of them, the first layer thickness is defined to ensure a wall y+ between 0 and 1.
The Reynolds-averaged Navier-Stokes (RANS) equations are solved using implicit, upwind, second-order accurate density-based solver. The k-ω SST turbulence model is employed by integrating to the wall (i.e., without using wall functions) and fully turbulent flow is assumed. The problem is solved using a second-order discretization scheme initially adopting a CFL number of 1.0 to reduce the steady-state iterative residuals by 3 orders of magnitude and, then, a CFL number between 5 and 10 to speed up convergence. After performing an iterative error analysis, the final normalized steady-state residual tolerance criteria used in this study is a 7 order of magnitude reduction ($10^{-7}$).

For the grid convergence study, the angle of incidence is set to $\alpha=0$ deg and $\alpha=4$ deg. Table 1 shows a summary of the main results by using four meshes having an increasing number of cells (see column #2) for $\alpha=0$ deg. The same table, also, includes the corresponding drag coefficients (column #3) and lift coefficients (column #4) percentage differences by considering numerical reference data obtained from private communication with Leonardo. Looking at the last two columns, it is clear that the last grid, i.e., ESTRO4, allows a good match with the available results because it ensures a percentage error for $C_L$ lower than 1% and does not unnecessarily increase the computational time. For this reason, ESTRO4 grid is used for successive calculation.

| Name of the hybrid grid | # cells | $|\Delta C_D|$ (%) | $|\Delta C_L|$ (%) |
|-------------------------|---------|-------------------|-------------------|
| ESTRO1                  | 8E6     | 8.4               | 2.1               |
| ESTRO2                  | 10E6    | 5.9               | 4.2               |
| ESTRO3                  | 11E6    | 4.5               | 5.2               |
| ESTRO4                  | 12E6    | 3.8               | 0.5               |

Table 1 – Results
5. CFD Analysis

This section is devoted to the description of the numerical results by considering two different procedures:
- CFD analyses in which the transition location is imposed at 5%);
- CFD analyses in which the position of the transition is evaluated.

The free stream flow conditions are reported in Table 2.

|    | T [K] | P [Pa] | M_∞ | Re [mio] | q_∞ [Pa] |
|----|-------|--------|------|----------|----------|
| A  | 288.15| 101325 | 0.2  | 6.5      | 2800     |
| B  | 288.15| 101325 | 0.35 | 6.5      | 8700     |

Table 2 - Flow conditions

Figure 4 shows the trend of the aerodynamic coefficients. As expected, the main differences are observable between M=0.2 and M=0.35 calculations. If the two configurations of wing are compared, the global coefficients are very similar demonstrating that the effect of the different wingtips is limited to a small portion of the wing. This behaviour is also confirmed by the pressure coefficient distributions depicted in Figure 5: this figure shows that the differences become evident starting from y/b=91% section.

Figure 4 - Aerodynamic coefficients: C_L(α), C_D(CL) and C_M(α)
The results reproduced in Figure 6 and Figure 7 refer to the CFD analyses in which the position of the transition is evaluated by using KC-BLC and LISAC codes. More in details, Figure 6(a) shows an example of the boundary layer profiles computed into two points of the upper surface of one section of the wing model. These data are reconstructed with KC-BLC code by considering the pressure coefficient distribution derived from CFD simulations. According to the flowchart depicted in Figure 2, the boundary layer profiles are used for the evaluation of the N-factor curves, that is reported in Figure 6(b).

Figure 6(b) shows the N-factor scan for 5000<fr<20000 Hz and 0<β<2000. The envelope of the upper surface of the ESTRO wing section can be simply obtained by analyzing the evolution of the N-factor as a function of the streamwise coordinate x/c depicted in this figure. Following the value of N factor of the DNW wind-tunnel (N=9) [10], we end up with an estimation of the transition location of x/c=0.52. Moreover, a dedicated parametric stability analysis has been carried out on the span of the ESTRO wing. The results are documented in Figure 7 and the ESTRO-project approach (represented by the dashed lines) is compared against a CFD simulation based on the use of Langtry-Menter 4-equation Transitional SST Model [14]. In details, we depict the transition location (xtr/c) as function of y/b. This figure shows that the LISAC transition locations agree well with the global results obtained using the Langtry-Menter 4-equation Transitional SST Model. We note that the LISAC points are in the region where the gradient of the skin friction coefficient is very high.
6. Conclusions
This work focuses on the CFD analysis and the preliminary evaluation of laminar flow extension by using an in-house numerical code based on the Linear Stability Analysis over a CFD solution. Numerical results will be compared by considering two different procedures: a) CFD analyses in which the transition location is imposed and b) CFD analyses in which the position of the transition is evaluated. The prediction of the transition location and the laminar flow extension is based on a numerical framework based on the coupling between a high-fidelity RANS tool and Linear Stability Equations (LNS). Two configurations of wing model are considered: CLEAN-AWL and CLEAN-IWT. The CFD analyses in which the transition location is imposed demonstrate that the two configurations of wing have very similar behaviours in terms of global aerodynamic coefficients and local quantities such as pressure coefficient distributions. The CFD analyses in which the position of the transition is evaluated thanks to the use of our in-house codes (KC-BLC and LISAC) forecast a position of the transition location around the 50% of the streamwise chord. Further investigations will be performed in the comparison of the results between ESTRO-project approach and the use of Langtry-Menter 4-equation Transitional SST Model.

7. Acknowledgement
This project has received funding from the Clean Sky 2 Joint Undertaking under the European Union’s Horizon 2020 research and innovation programme under grant agreement No 831809.
8. Disclaimer
Any dissemination of results must indicate that it reflects only the author’s view and that Clean Sky 2 Joint Undertaking is not responsible for any use that may be made of the information it contains.

References
[1] R. Rudnik, A. Ronzheimer, M. Schenk, Berechnung von zwei- und dreidimensionalen Hochauftriebskongurationen durch Losung der Navier-Stokes Gleichungen, in: Jahrbuch 1996, Bd. II der Deutschen Gesellschaft für Luft- und Raumfahrt, Lilienthal Oberth e.V. (DGLR), Jt96-104, Deutscher Luft- und Raumfahrtkongre, DGLR-Jahrestagung Sep. 1996, Dresden, Deutsche Gesellschaft für Luft- und Raumfahrt, Lilienthal Oberth e.V. (DGLR), Godesberger Allee 70, D-53175 Bonn, Germany, 1996, pp. 717-726.
[2] A. Krumbein, eN transition prediction for 3D wing configurations using database methods and a local, linear stability code, Aerospace Science and Technology 12 (2008) 592-598.
[3] L Siconolfi, V Citro, F Giannetti, S Camarri, P Luchini, Towards a quantitative comparison between global and local stability analysis, Journal of Fluid Mechanics 819, 147-164.
[4] D Fabre, J Tchoufag, V Citro, F Giannetti, P Luchini, The flow past a freely rotating sphere, Theoretical and Computational Fluid Dynamics 31 (5), 475-482.
[5] D Fabre, R Longobardi, V Citro, P Luchini, Acoustic impedance and hydrodynamic instability of the flow through a circular aperture in a thick plate, Journal of Fluid Mechanics 885.
[6] A.M.O. Smith, N. Gamberoni, Transition, pressure gradient and stability theory, Douglas Aircraft Company, Long Beach, Calif. Rep. ES 26388, 1956.
[7] J.L. van Ingen, A suggested semi-empirical method for the calculation of the boundary layer transition region, University of Delft, Dept. of Aerospace Engineering, Delft, The Netherlands, Rep. VTH-74, 1956.
[8] V Citro, P Luchini, Multiple-scale approximation of instabilities in unsteady boundary layers European Journal of Mechanics-B/Fluids 50, 1-8.
[9] https://cordis.europa.eu/project/id/831809.
[10] W1-DNW-05 Requirements for wind tunnel models to be tested at DNW, DNW, Version 2017.
[11] K. Kaups, T. Cebeci, Compressible laminar boundary layers with suction on swept and tapered wings, Journal of Aircraft, Volume 14, Number 7, 1977.
[12] S Russo, N Natale, V Citro, F Giannetti, An accurate RANS-based transition prediction approach (part I), EASN 2021.
[13] https://cordis.europa.eu/project/id/323460.
[14] Menter, F. R., Langtry, R., and Volker, S., "Transition Modelling for General Purpose CFD Codes," Flow, Turbulence and Combustion, Vol. 77, No. 1, Nov. 2006, pp. 277-303.