AIAA-2002-2931

AERODYNAMIC OPTIMIZATION OF SUBSONIC FLYING WING CONFIGURATIONS

B. Mialon, T. Fol, C. Bonnaud
ONERA, Applied Aerodynamics Department, F92320 Châtillon, France
Airbus France, 316, Route de Bayonne, 3060, Toulouse Cedex 03, France
Seditec, Groupe Aeroconseil, 3, rue Dieudonné Costes, BP75, 31703, Blagnac Cedex, France

Abstract[1]

The paper presents some aerodynamic results of initial design studies carried out at ONERA and Airbus France, on flying wing configurations. A description of the analysis and the design process, using CFD tools, is given and some results regarding the effects of the planform are presented.

Introduction

Passenger and cargo air traffic is expected to grow about 5% a year over the next 20 to 30 years. The conventional jetliner configuration, with a cylindrical fuselage, a swept wing and empennage and engines mounted on pylons under the wings, was developed nearly 50 years ago. Over the years, jetliners have grown considerably to meet market demand, to reach more than 550 passengers in a 3-class layout for the next generation. Designing such a large capacity aircraft brings up new technical challenges and the size limit for the "tube and plank" configuration is probably reached. In addition to the size limit, future aircrafts will have to meet the demands of increased economic efficiency and reduced environmental impact.

As a consequence, novel configurations for subsonic transport have been the subject of a renewal of interest in the last decade[1,2,3,4]. The flying wing, or blended wing body, seems to be one of the most promising concept regarding very high capacity aircrafts. However, no experience exist with such unconventional configurations, which makes their design very challenging in a number of disciplines. The classical design methodologies do not apply and the high rate of integration of the shape makes even stronger the interdependency between aerodynamics, aeroelasticity, flight mechanics and structure. A program of fundamental research has been launched on these configurations within Airbus. One element of this program has been carried out by Airbus France and ONERA, with two main objectives :

- to assess the cruise aerodynamics performance of a viable flying wing, in order to prepare future experimental work ;
- to get basic knowledge on the sensitivities to geometrical parameters, on the design process and on the existing tool relevance.

The focus here is on high speed, clean wing aerodynamics design. The paper presents the initial optimization study carried out on two configurations, in order to produce a viable design, valuable for wind tunnel testing. The engine installation is not considered in this paper, although in parallel with the clean wing optimization, the problematic of the engine integration on such a novel configuration has been investigated, in order to identify the major technical issues and to find out a range of appropriate solutions.

The first part of the paper is dedicated to the description of the optimization problem specifications while the second part focuses on the tool description. The third and fourth parts present the design process for the configurations at hand.

Specifications

Two planforms have been proposed by Airbus France (figure 1).

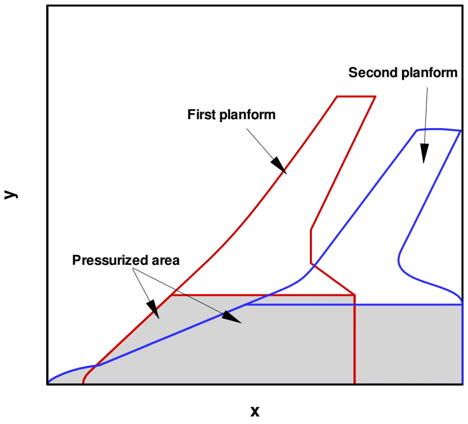
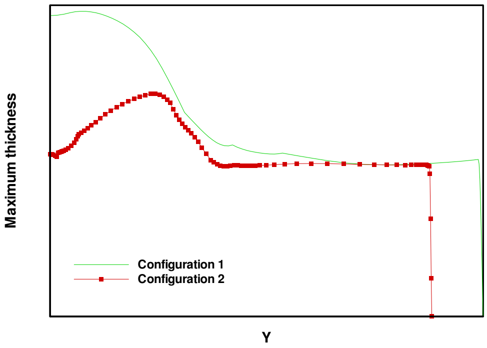


Figure 1: Top view of the configurations studied

---

[1] Copyright © 2002 by ONERA. Published by the American Institute of Aeronautics and Astronautics, Inc., with permission.

The first configuration is composed of an inner body generated by rather thick airfoils (figure 2), appropriate for lodging passenger and/or freight, and of two wings mounted outboard. The maximum thickness is located near the centerline. The leading edge sweep angle is almost constant all along the span, while the trailing edge sweep angle is 0° in the inner and central areas, linked through a region with highly negative sweep angle. The shape was initially generated using rather conventional airfoils, producing an untrimmed, nose-down, configuration.



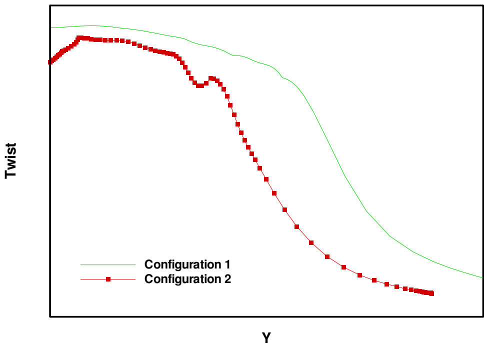


Figure 2 : Thickness-to-chord ratio and twist distributions

The second configuration includes more constraints. The sweep angle at leading edge is much higher in the inner part than in the outer wing. Moreover, the aspect ratio is lower for the second shape than for the first one, due to the shorter span and to the longer local chord in all the inner area. The thickness-to-chord ratio of the airfoils are much lower than the first configuration ones. The maximum thickness-to-chord ratio is located close to the cabin/non pressurized area interface, due to the constant absolute maximum thickness of the cabin.

The design problem objective is to increase the lift to drag ratio L/D in cruise condition and to determine the CL corresponding to the maximum L/D ratio. The problem is constrained by some aerodynamic and geometric considerations:

- The trim: the aircraft should be longitudinally trimmed at cruise, which means that the pitching moment must be zero naturally (without tail);
- The longitudinal stability: a zero static margin is the target for the optimization output;
- The maximum angle of attack is imposed, in order to respect an acceptable deck angle;
- The volume of the pressurized area : a minimum value is imposed for each configuration;
- The maximum thickness of airfoils is fixed on the outboard wing;
- The planform is imposed.

Finally, the degrees of freedom of the design problem are basically the airfoil shapes (thickness and camber) and the twist distribution.

Numerical methods used

Analysis

In total, not less than three aerodynamic codes have been used for the analysis of the initial and modified shapes.

The CANARI code, developed at ONERA[5], is a general Euler and Navier-Stokes solver for structured multi-block meshes. It has been validated for a large number of fundamental and complex practical applications, including external and internal flows for subsonic, transonic and supersonic flows. This code has been used as a Euler solver. The viscous effects are taken into account through a weak coupling with an ONERA boundary layer code, 3C3D[6]. This coupled tool has been validated, in particular for civil aircraft applications, for non separated flows.

The NSMB code[7] has been used by Airbus France for the analysis of both configurations. This code is a structured multi-block Navier-Stokes solver. It has been used with the Spalart-Allmaras turbulence model because of its reasonable results obtained on a wide range of flows and its good numerical properties.

The *elsA* platform, under development at ONERA[8], has been used for the analysis of the initial configuration 2. The *elsA* software is a structured, multi-block, multi-application new generation code. A number of turbulence models is implemented, among which some are based on the wall functions[9]. The *elsA* software results have been compared with experimental data obtained at high Reynolds number in the framework of the European project HiReTT[10] (High Reynolds Number Tools and Techniques for Civil Transport Aircraft Design). For the application at hand, some computations have been performed with three two-equations turbulence models.

Structured C-type meshes have been generated for both configurations. The meshes used with the CANARI code (Euler type) represent five blocks and 800 000 nodes for configuration 1, and seven blocks, 700 000 nodes for configuration 2. The meshes used for Navier-Stokes computations differ only in the normal grid spacing near the walls, which is refined in order to get y^+ values adapted to the turbulence modeling. For example, y^+ mean value observed on the computation of configuration 2 with the wall functions on a 1 550 000 nodes mesh, is about 30 on the wing, which is acceptable for this type of turbulence modeling[9]. It can be pointed out that the use of simple C-type meshes for blended wing bodies does not allow the same absolute grid spacing in the chordwise direction to be obtained inboard and outboard, due to the high taper ratio specific to these configurations. The patch grid technique, presently under development in the *elsA* software, should allow local mesh refinement to be considered in the near future.

Design

The major steps in the shape optimization process have been obtained using the "cut and try" approach. A 3D surface generator has been developed in order to modify the surfaces. A number of master sections has been preliminarily defined on the wing (figure 3). This module allows modified 2D wing sections to be integrated on the wing. Once the master sections were modified, an interpolation of the vertical deviation from the initial shape is carried out on the whole wing surface. The twist distribution can be modified by the module too.

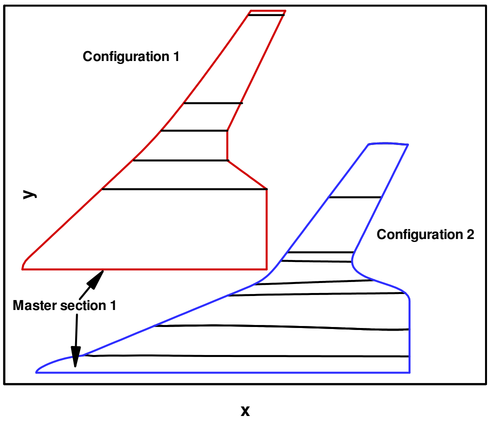


Figure 3: Master sections for both configurations

The new CFD meshes are generated using an in house mesh deformation tool, based on the spring analogy[11]. The analysis of the modified shapes is carried out with the Euler+boundary layer code mentioned above.

A post processor[12,13] has been used in order to extract the drag terms separately through a far-field approach. This tool allows physical contributions to the drag to be computed accurately, which is helpful for the designer, especially for drag reduction purposes.

Numerical optimization has also been applied in this study. The tool in use at ONERA is based on a minimization programme[14] and on the modules described above (CANARI code used in the Euler option only). In the near future, gradient computations through adjoint state and linearized equations will be available in the *elsA* software.

Design of the first configuration

The analysis of the initial configuration 1 confirmed the inadequacy of the initial airfoils, especially inboard (figure 4): the shock wave location on the upper surface is very aft and the negative pressure gradient upstream the shock is unfavorable to the drag rise. Local 3D Mach numbers raise up to 1,28 in the central region. Supersonic flow appear on the lower surface too. It should be noticed that the flow is fully non-separated.

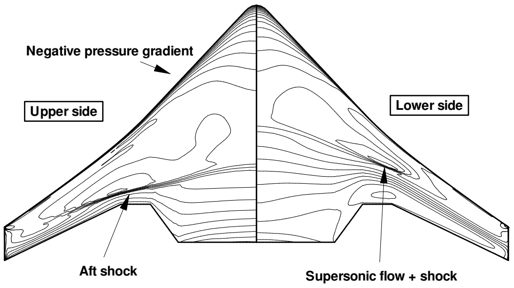


Figure 4: Pressure contours at cruise for initial configuration 1

The pitching moment of the initial configuration 1 is highly negative. Basically, the global coefficients result from the combination of local pitching moment and local lift distribution in the spanwise direction. A simple model, taking into account the planform and the local force distributions, has been developed in order to assess the influence of modifications of local force distributions on the global coefficients.

The figure 5 shows the example of two variations of pitching moment distribution which produce the same effect on the global coefficients of the configuration: a modification of the wing inboard is much more efficient than outboard, to obtain a given effect on the pitching moment. The same kind of result is obtained considering modifications of the lift distribution alone.

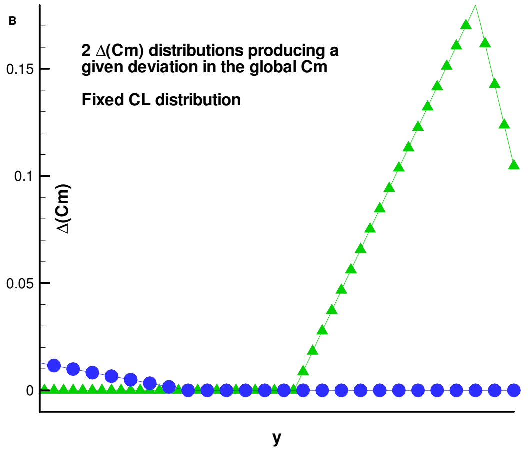


Figure 5 : Example of two modifications of the pitching moment distribution producing the same deviation of the global pitching moment

The relationships between 2D and 3D flows has also been investigated, in order to determine if the 3D design process can be initiated by a 2D airfoil optimization and continued by a 3D refinement. We did not find any rules which allow 2D flows to represent correctly 3D flows. Especially on the lower surface, it seems that the inner sections have a very strong influence on the flow all along the span.

From these preliminary investigations, we decided to work directly in 3D, through a process made of the modification of the master sections and of the surface mesh generation. Then the volume grid is deformed and the aerodynamic analysis of the new shape is performed. Moreover, the design process was organized in three stages :

1. To reduce the pitching moment (absolute value), with the target "naturally trimmed aircraft". These modifications are mainly located inboard ;
2. To improve the pressure distribution, with the objective to reduce the shock strengths and to delay the separation onset. These modifications are mainly located in the central area and outboard ;
3. To increase the maximum L/D ratio. Numerical optimization was used for this stage.

The first two stages of this strategy have been carried out using 37 iterations of modifications. The root master section camber has been increased between the leading edge and 25% of the chord and decreased aft, resulting in an increase of front loading and a decrease of rear loading, while moving upstream the shock location on the suction surface and eliminating the shock on the lower surface. In the central region of the wing, the same type of modifications were introduced, with an additional increase of the thickness near trailing edge. Finally, the master sections located outboard have also been slightly reshaped in order to tailor the supersonic recovery on the upper surface. Some resulting airfoil modifications, as well as their effect on pressure distributions, are presented in the figures 6 and 7.

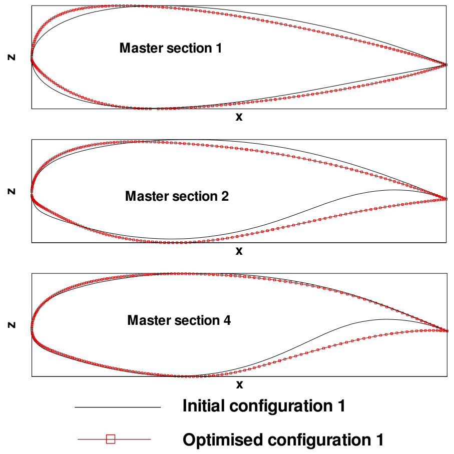


Figure 6 : Modifications on three master sections

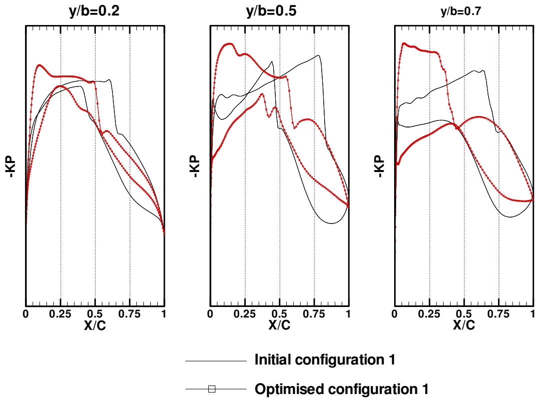


Figure 7 : Pressure distributions on three sections distributed on the wing

The numerical optimization has been used in the third step of the design procedure. The number of nodes of the meshes has been reduced in order to save CPU time, paying attention to maintain the mesh refinements in the regions where strong pressure gradients are observed. The agreement between the force coefficients and the drag components obtained with both the refined and the coarsened grids is acceptable. Especially for the lift induced and wave drag, the use of a far field type post processor allows the dependence on the grid size to be drastically reduced. Numerical optimization have been performed, with a drag objective (sum of the lift induced and wave drag, the latter being a relatively small contribution) under lift and pitching moment constraints. The twist distribution as well as the camber distributions of the master sections located inboard were considered as degree of freedom. This numerical optimization stage has produced a slightly improved design.

Finally, the optimized shape is much better regarding the compressibility effects (figure 8). The shock on the upper surface has been swept aft and moved upstream. However, its intensity remains quite strong, due to the influence of the inboard wing and to the high thickness of the airfoils inboard. The pressure contours at lower side are much more regular and the supersonic recovery in the kink region is tailored.

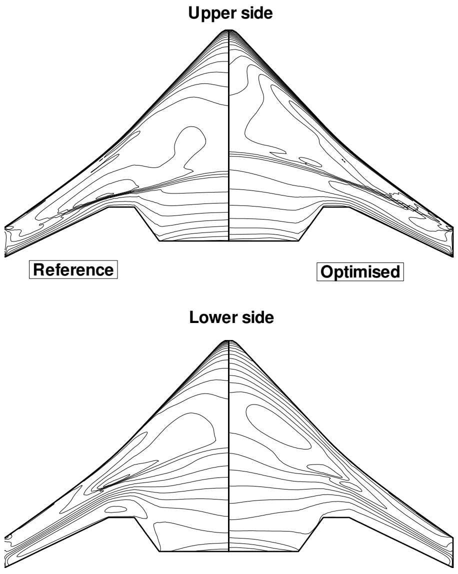


Figure 8 : Initial and optimized pressure contours on the configuration 1 – M=0,85 – CL=0,22

The shape optimization results in a significant improvement of the pitching moment and the maximum L/D ratio (figures 9).

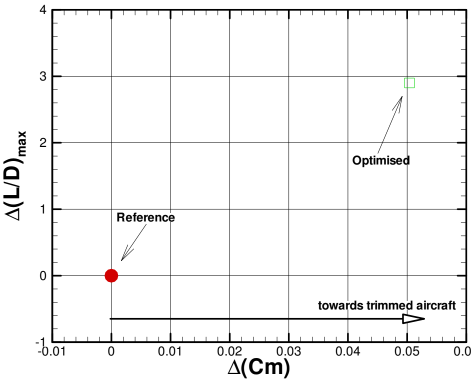


Figure 9 : Performance gain between the initial and optimized shapes for the configuration 1 – M=0,85

The distribution of load and pitching moment in the spanwise direction is presented in figure 10. The optimized shape is subject to a more elliptical load distribution than the initial one; the local lift is decreased inboard and increased outboard, what corresponds to a nose-down effect. However, even combined with the reduction of the local nose-up pitching moments, the final shape remains longitudinally untrimmed. This is because it was chosen to avoid negative aft camber on the airfoils inboard.

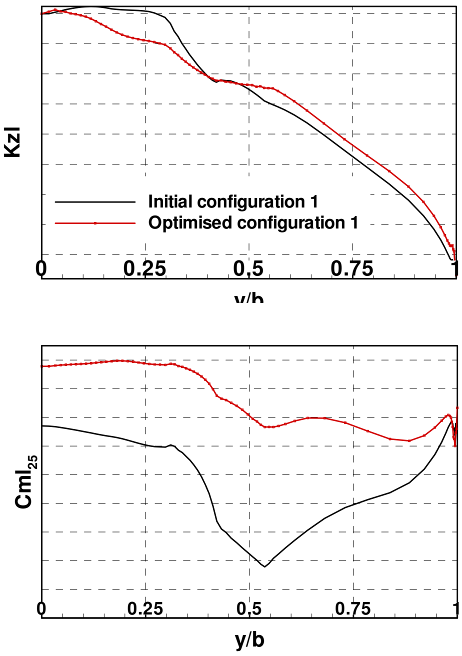


Figure 10 : Load and pitching moment distributions in the spanwise direction for the initial and optimized configuration 1

The incidence for the maximum lift-to-drag ratio has been increased by one degree while the corresponding lift was unchanged. As far as the static stability is concerned, it has been checked that the aerodynamic center is at the same position relative to the assumed center of gravity. Additional constraints on the volume of the pressurized area and the deck angle are satisfied too. Extensive off-design computations showed that the flow remains attached at CL close to 1,3CL_{cruise}, which is promising for meeting the classical buffet onset requirement. The shock location and strength, and thus the performance, are very sensitive to the aerodynamics conditions. Low speed computations have shown that the optimized shape present pressure distributions rather safer than the initial shape, especially in the outboard wing where the pressure level near the leading edge, on the upper side, is higher.

Design of the second configuration

Flight Reynolds numbers based on the mean aerodynamic chord are about 250 millions for high capacity flying wings. The aerodynamics performance predictions accuracy determined for conventional configurations might be influenced by these comparatively very high Reynolds numbers. Thus, Navier-Stokes solutions have been obtained for three different turbulence modeling: the Wilcox k-ω model with and without wall functions (with the *elsA* software), and the Spalart-Allmaras model (with the NSMB code). The wall function approach is highly interesting because the appropriate y+ values, at the center of the first cells, are in the region [10-50] instead of 1 when the wall functions are not used. Thus, the grid density in the wall boundary layer region can be reduced. Pressure distributions are compared in three sections in figure 11. The main differences between the aerodynamic solutions compared are concentrated on the shock location, for which deviation can reach 5% of the local chord outboard. The influence of the wall functions on the pressure distributions and on the aerodynamic forces is not significant.

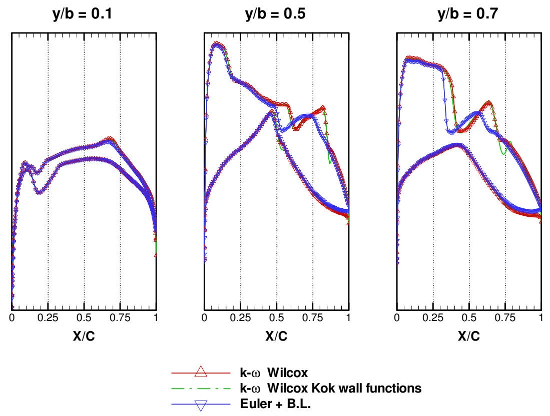


Figure 11 :Influence of the turbulence modeling on the pressure contours for initial configuration 2

This shape is better suited than the initial first planform (figure 12); no critical supersonic region exists inboard, thanks to the rather low relative thickness of airfoils and to the high sweep angle. The outer wing is subject to a double shock pattern, which poses the problem of the aft shock sweep.

The CL_α slope is low due to the large inboard surface and the high sweep angle. The span loading indicates a highly loaded inner wing, although local lift is very low (figure 13). The lower aspect ratio relative to configuration 1 leads to an increased lift induced drag. Although the rear loading is low, the initial configuration 2 is untrimmed. Further improvement is needed in the tailoring of compressibility effects too.

Configuration 2 has been optimized under the same constraints as the first configuration : again, the first set of optimizations was dedicated to satisfy the trim constraint while the second set aimed at the L/D maximization.

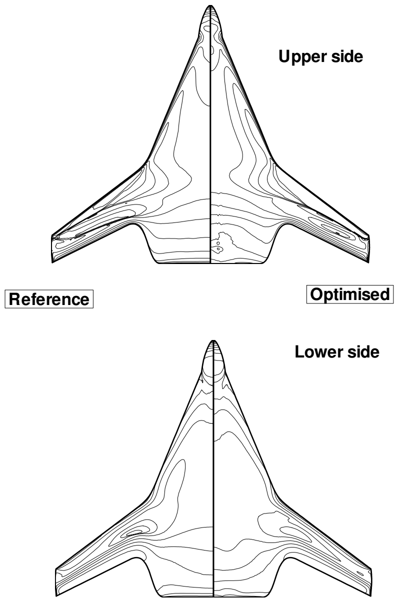


Figure 12 : Cruise pressure distribution on the configuration 2

Thus, the modifications focused on the inboard airfoils, with an increase of front camber and the introduction of negative aft camber, associated to a re-twisting aimed at limiting the decrease of the local lift coefficient. This results in a positive pressure gradient in the supersonic region and an inverse rear loading on the pressure area. Although the inboard wing twist has been increased, the maintained global lift coefficient requires an increase of the angle of attack of (0,4°). In the kink region, the shock tendency on the lower surface has also been reduced mainly through the local decrease of thickness. The resulting configuration is almost trimmed and satisfies the constraints.

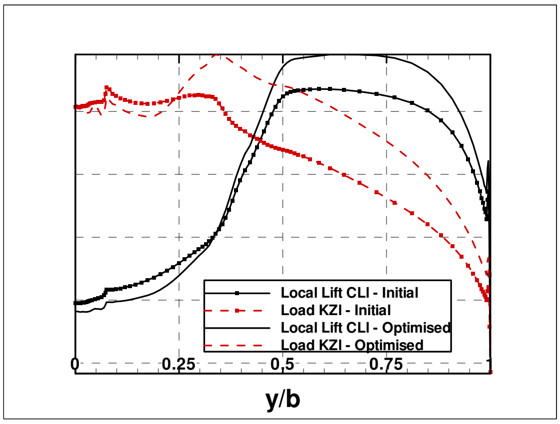


Figure 13 : Cruise span loading and lift distribution for configuration 2

Conclusions

As expected, these flying wing studies have allowed basic knowledge on the design constraints relevant to tailless high capacity aircrafts to be acquired. A new family of airfoils, better suited for this kind of tailless configurations, has been generated. A database has been produced too, with two different configurations and three different aerodynamic codes. The importance of geometrical parameters such as sweep angle at leading edge, aspect ratio or shape of the generated airfoils has been investigated. Two major concerns have especially been addressed :

1. The trim is considered to be the hardest constraint to satisfy, as well as the most costly in terms of aerodynamics performance;
2. The tailoring of compressibility effects inboard is difficult and can be dealt with an increase of the sweep angle.

At the same time, an optimization strategy has been validated, either using manual modifications or numerical optimization capabilities. The trade-off between balance and aerodynamic performance has been investigated. The future work in aerodynamics concerns engine installation and study of appropriate mounting arrangements for wind tunnels models. Studies in flight mechanics and structure will also be launched with the objective to proceed to multidisciplinary optimizations.

Acknowledgements

The authors would like to acknowledge the French Ministry of Defense and the French Civil Aviation Authorities for the funding of the studies presented in this paper.

References

[1] *John McMasters, Ilan M. Kroo, "Advanced Configurations for Very Large Transport Airplanes (Invited Paper)", AIAA Paper 1998-0439, 36th AIAA Aerospace Sciences Meeting and Exhibit - Reno - January 1998*

[2] *Mark A. Potsdam, Mark A. Page, and Robert H. Liebeck, "Blended Wing Body Analysis and Design", AIAA Paper 1997-2317*

[3] *H. Smith, "College of Aeronautics Blended Wing Body Development Programme", ICAS 2000 Congress*

[4] *V.E. Denisov, A.L. Bolsunovsky, N.P. Buzoverya, B.I. Gurevich, "Recent Investigations of the Very Large Passenger Blended-Wing-Body Aircraft", ICAS-98-4,10,2.*

[5] *A.M. Vuillot, V. Couaillier, N. Liamis, "3D Turbomachinery Euler and Navier-Stokes Calculations with a Multidomain Cell Centered Approach.", AIAA Paper 93-2576 (1993).*

[6] *R. Houdeville, "Three-dimensional Boundary Layer Calculation by a Characteristic Method", Fifth Symposium on Numerical and Physical Aspects of Aerodynamic Flows, Long Beach, January 1992.*

[7] *C. Gacherieu, R. Collercandy, P. Larrieu, S. Soumillon, L. Tourrette, S. Viala,, "Navier-Stokes Calculations at Aerospatiale Matra Airbus for Aircraft Design", ICAS 2000 Congress*

[8] *L. Cambier., M. Gazaix, "An Efficient Object-Oriented Solution to CFD Complexity". AIAA Paper 2002-0108, 40th AIAA Aerospace Sciences Meeting and Exhibit, 14-17 Jan. 2002 Reno, Nevada, United States.*

[9] *E. Goncalvès., R. Houdeville, "Reassesment of the Wall Function Approach for RANS Computations", Aerospace Science and Technology, 5, pages 1-14, 2001.*

[10] *S. Ben Khelil, J.-L. Gervois, G. Carrier, F. Moens, P. Viscat, C. François, "Assessment of elsA software through civil transport configurations", CEAS Aerospace Aerodynamics Research Conference, 10-12 June 2002, Cambridge, United Kingdom*

[11] *A. Dugeai, A. Madec, A.-S. Sens, " Numerical Unsteady Aerodynamics for Turbomachinery Aeroelasticity.", In 9th International Symposium ISUAAAT, Ecole Centrale Lyon, France, September 2000*

[12] *V. Schmitt and D. Destarac, "Recent Progress in Drag Prediction and Reduction for Civil Transport Aircraft at ONERA", AIAA paper 98-0137*

[13] *D. Destarac, "Drag Extraction from Solutions of the Euler and Navier-Stokes Equations", ONERA-DLR Aerospace Symposium, Paris, June 1999*

[14] *G.N. Vanderplaats, "Numerical Optimization Techniques for Engineering Design: With Applications", McGraw-Hill Book Company*