Proc. Int. Symp. on Inverse Problems in Engineering Mechanics (ISIP 98), 24-27 March 1998, Nagano, Japan
Elsevier Science, (1998)

CONFIGURATION TEST CASES FOR AIRCRAFT WING ROOT DESIGN AND OPTIMIZATION

H. SOBIECZKY
DLR German Aerospace Research Center
Bunsenstr. 10, D-37073 Göttingen

ABSTRACT

Computational methods for aerodynamic inverse design and optimization need tools for the definition of input geometry and input flow variables. Challenges exist to extend three-dimensional design methods for aircraft wings to include geometrical details of the wing - fuselage juncture in the systematic design procedure. Geometrical methods to define such surfaces are presented and examples are shown for experimentally analyzed configurations DLR-F5 and DLR-F9. These are baseline for the definition of test cases for design methods with a combined data input of geometry and pressure definition.

KEYWORDS

Aerodynamic Design, Wing Roots, Fillets, Geometry Definition, Configuration Integration

INTRODUCTION

High speed aerodynamic design concepts are classically based on our knowledge of isolated compressible flow components with or without shocks, described by mathematical functions which are modelled from particular solutions to some simplified equations of flow motion.

Resulting from this knowledge are computational methods which allow for an inverse formulation of local flow parameters, like the pressure distribution along a wing or turbomachinery blade. This approach implies that we also know which type of flow quality distribution leads to optimum performance of the aerodynamic configuration to be designed. Computational Fluid Dynamics (CFD) already provides analysis tools to verify both the model flows and also the flow about a complete design-resulting configuration so that a design concept can be judged from such results prior to experimental verification.

The use of CFD for design, however, needs most of the direct (geometrical) and/or some of the inverse (flow parameter distribution) input for the flow solver. New and efficient computational aerodynamic inverse design methods have already been developed, see the review [1]. Such methods, for three-dimensional configurations, allow for design of clean aircraft wings, but the complexities of wing root junctures, wing tips and the integration of engines with pylons and nacelles still pose big problems for a systematic inclusion in an inverse design concept. Examples of a given geometry with such complexities, plus reliable computational flow analysis results at design conditions, are therefore of some value for the developer of new codes for the complexities as they occur in real complete aircraft.

In this contribution we select only one of these configurational challenges: The wing root juncture.

Any aircraft wing (except ‘Flying Wing’ aircraft) needs to be mounted onto a fuselage and it is a topic of growing importance to try a systematic optimization of the juncture surface, from simple fillet roundings of the wing-body intersection to a strongly integrated configuration. For such design efforts it is desirable to combine the results of inverse wing design with options of parametric shape variations. In this contribution we stress the role of a refined parameter-controlled input description for obtaining realistic 3D configurations. Geometry software has been developed at DLR German Aerospace Research Establishment especially for such applications. Two ‘DLR-F’ model configurations are presented and the options to use them as computational design test cases may be discussed.

FLOW STRUCTURE NEAR WING ROOTS

The usual inverse approach to design an airfoil or wing is to prescribe a pressure distribution. Numerical techniques today allow for an iterative solution to the inherently ill-posed problem for compressible and 3D flows, but the complexity of boundary conditions beyond simple configurations may lead to a rather slow design procedure because of contradictions in the prescribed boundary conditions. This is true especially near wing roots where complex phenomena dominate the flow structure. Not only the dramatic viscous phenomena in corners as known already from low speed and especially low Reynolds number flows are creating difficulties in the systematic design approach: in compressible flow also the outer inviscid flow creates problems via the occurrence of shock waves.

The quality of high speed flows is dominated by the occurrence of shocks which seriously influence aerodynamic drag and hence the efficiency of a flight vehicle operating in this speed regime. Shock waves are, in a scale comparable to characteristic geometric vehicle length, geometric structures themselves and, therefore, invite to be modelled using mathematical tools with quantitative evaluation in mind, to estimate the losses manifest in aerodynamic drag. Especially for three-dimensional flows past realistic configurations this is a nearly impossible task, our design efforts focus on more or less systematic methods to keep shock waves weak or non-existent. Inverse design methods to achieve this goal, may face some problems, though:

Prescribed smooth flow structure and expected resulting geometry (which is usually enforced already to some degree by prescribed global parameters like a wing planform, while leaving certain freedom by allowing section geometry to result from the inverse design approach) may contradict the need of the flow to be physically existent.

This general remark may be illustrated by the example of designing a swept wing mounted onto a flat (non-contoured) sidewall or fuselage, in inviscid transonic flow, with a design pressure distribution prescribed along the span of the swept wing. Such a pressure distribution may ignore the following, purely inviscid, transonic phenomena:

A check of the near sonic 3D basic equations (continuity and irrotationality) reveals the need of an oblique shock occurring at the intersection of the sonic surface (where local Mach number

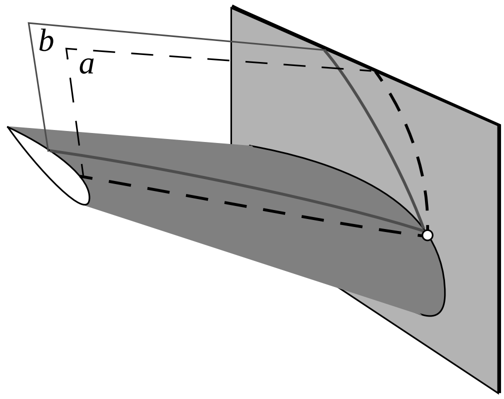

Fig. 1. Flow at wing root junction: Inviscid flow theoretical model at a sonic expansion on wall intersecting with swept wing; Sonic surface (a) and oblique ‘Leading Edge Shock’ (b)

equals unity), the swept wing surface and the wall, to help fulfilling the geometric boundary conditions. Such shock is known from numerical analysis and experiment as ‘Leading Edge Shock’.

Figure 1 illustrates this flow detail, it may be interpreted as a 3D extension of the well-known 2D wedge shoulder singularity, where the geometric discontinuity triggers occurrence of an oblique shock immediately behind the sonic expansion. In 3D flow the weak singularity is triggered by the corner and non-vanishing wing surface gradient normal to the wall. With the shock trace on the wing running far into the spanwise direction and eventually coalescing with the recompression shock, its influence on flow quality via viscous interaction should not be neglected, inverse design strategies therefore should be able to systematically suppress this theoretical possibility of unwanted effects.

Practical ways to curb this weakly singular behavior of the inviscid flow in the juncture region of wing and wall is the rounding of such junctures, called Fillets. Inviscid flow to start with, but interacting with it, boundary layer flow leading to separations in wing roots without fillets, may be smoothened quite remarkably by proper fillet design.

The question raised next may be how to use inverse concepts and optimization strategies to design wings with fillets: Target pressure distributions should be prescribed and 3D-shapes of wing including its root geometry should result. On the course to extend existing inverse methods for wing design to do this successfully we might use test examples defined in a direct approach: Surface geometry obtained by suitable geometry pre-processing and subsequent computational and experimental analysis of the flow in high speed operating conditions.

In the following, two geometric techniques are illustrated which may be used for a parameterized surface description with parameters well-suited for practical configurations and for influencing flow quality efficiently.

COMPONENT JUNCTURES

In this contribution an effort is made to stress the importance of defining not only the main component - the wing - by parameter-controlled mathematical functions: This first goal has become commonly known as a key to provide input for aerodynamic optimization. Moreover, it is desirable to tailor geometric model functions in a way that the root of the wing, where it is joining the fuselage, is equally exactly described by model functions.

In a purely analytic approach we have developed two non-iterative, direct surface definition routines to mount aircraft wings onto a fuselage or tunnel wall.

Blended projection technique

In a geometry program for surface definition we have options to define cross sectional body surfaces as well as sectional wing surfaces (Fig. 3a). The fuselage is explicitly defined by cartesian coordinates

$$y = F_1(x, z)$$

while the wing is given either by a sectional definition similar to the body, or provided from an external database, suitably described by a 3D surface grid, denoted here as

$$F_2(x, y, z) = 0.$$

This latter wing surface grid is formatted as a sectional ($m = 1,...m_e$) and spanwise ($n = 1,...n_e$) double loop of the coordinates x, y, z of the wing which has been shifted to the proper position relative to the fuselage in 3D space. A portion of wing sections at the root, ($n = 1,..n_{root}$), is shaped with thicker wing sections and larger chord length giving a ‘trumpet-like’ surface of the isolated wing. We notice that this root portion so far can be described with the spanwise variable function tools which are used for the complete wing in the spanwise direction toward the wing tip. The given wing sections need to be located at constant span stations $y_w(n)$= const, (see Fig. 2a).

A subsequent controlled deformation of the wing sections in the horizontal direction y is performed by using a blending function within the interval $0 < \eta \leq 1$

$$\eta = Fct((y_w(n) - y_w(n_{root}))/(y_w(n_{root})))$$

which controls a weighed projection of the sections $n < n_{root}$, so that the section n = 1 is moved completely onto the fuselage surface (using the explicit relation F_1) and the other sections in a gradually decreasing deformation of their spanwise coordinate, as illustrated in Fig. 2b.

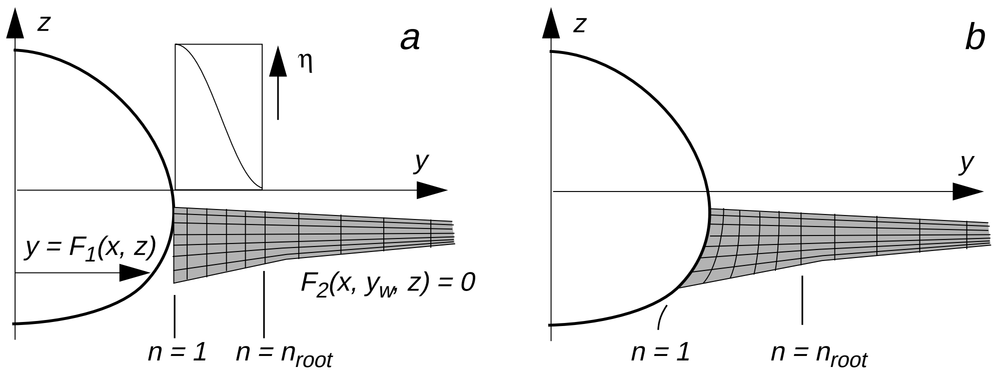

Fig. 2. Blended projection technique to mount a wing F_2 onto a body F_1. Shape F_1 remains unchanged while a portion of F_2 is moved into y-direction controlled by blending function $\eta(n)$.

This technique is simple and very efficient in creating aerodynamically favorable wing fillets, applications are easily calculated for mid-body mounted wings, and also applicable to low-mounted and high-mounted wings as long as the wing root area stays well within the fuselage crest lines.

Creating arbitrary shape junctions

A more general option has been added to our geometry tools to provide junctions between components with given analytic structure (or at least given as dense structured data sets so that surface gradients along the juncture curves can be obtained with acceptable accuracy). The idea is to connect independently given shape components F_1 and F_2 by a surface element F_3, which is defined by curves *c* (generatrices) connecting points A and B along given curves *a* and *b* on F_1 and F_2, see Fig. 3. The conditions to be fulfilled by the generatrix *c* are to observe a given tangent *t* in A on curve *a* in F2, and to observe an ending which is tangential to surface F_1 in B on curve *b*. This is done by a Bezier or higher order curve situated within a plane defined by triangle ABC, where C is the intersection of *t* with the plane β tangent to F_1 in B.

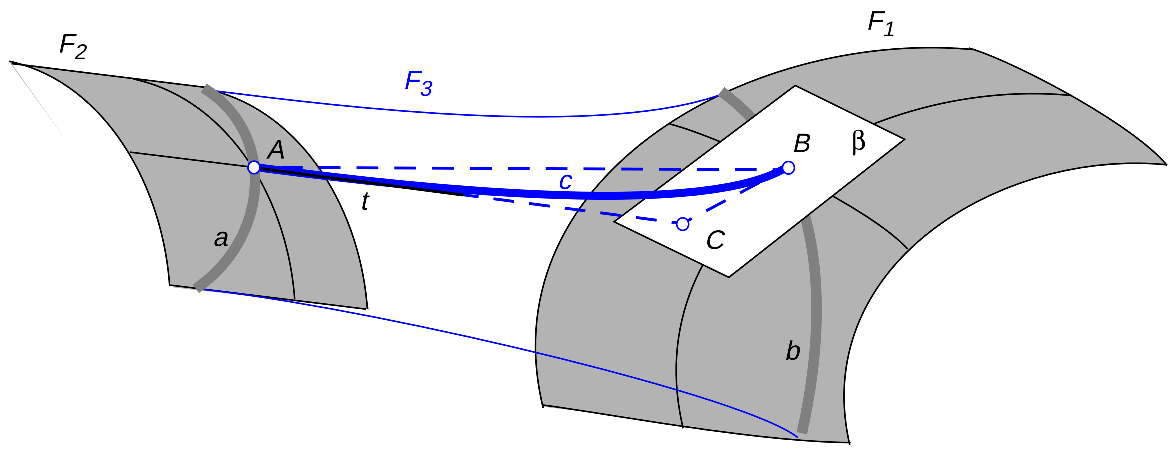

Fig. 3. Fillet definition by curves tangent to given surface gradients.

Advantages of this technique are the independence of the spanwise direction which is essential for using the explicit relation for F_1 in the former technique.

Both methods to provide smooth junctions are supporting the definition of aerodynamically efficient fillets in an attractive way because of the mathematically explicit relations used, in this regard the method is even faster than a precise intersection interpolation from given data sets for the two components.

We ask for fast and precise techniques to define these surfaces as preprocessing effort for computational optimization, either with purely direct iteration or using an inverse technique to arrive at useful configurations. Fast geometry data definition for the input of CFD analysis seems of paramount importance for an economical design and optimization procedure. The two outlined methods have been used to provide data for case studies; two of these are introduced below and are proposed as test cases for CFD and aerospace design communities.

GEOMETRY PREPROCESSOR SOFTWARE TOOLS

The ability to generate complete and realistic aero(-space) vehicles in a rapid and rational way calls for the development of software tools suited for modern design departments in aerospace industry equipped with high performance computer hardware and software. With CAD systems serving for structural design and also for the production of wind tunnel models and aircraft components, we still lack software for the aerodynamic pre-design phase, for search processes defining the baseline aerodynamics by rapid parametric variations.

Analytic techniques like the one used here for component definition including their junctions are therefore quite welcome and we have developed such computational tools, used them for examples like the ones shown below and finally have produced input data for a subsequent use in CAD systems like CATIA for model production or ICEM for unstructured grid generation [3].

DLR-F5: TEST WING FOR COMPUTATIONAL FLUID MECHANICS

More than a decade ago we have generated a test wing for CFD code validation, using an early version of this geometry preprocessor software. The example served for an attempt to compare numerical simulation algorithms for the Reynolds averaged Navier Stokes equations [4].

The wing leading edge sweep of 20 degrees and a symmetrical section along span are integrated smoothly with a large fillet providing rounded corners of the half model mounted onto the tunnel side wall, or rather a special splitter blade off the tunnel side wall. The wing is a simple example of the above mentioned option to generate the fillet as part of the wing, no subsequent blended projection is necessary because the side wall is plane, see Fig. 5

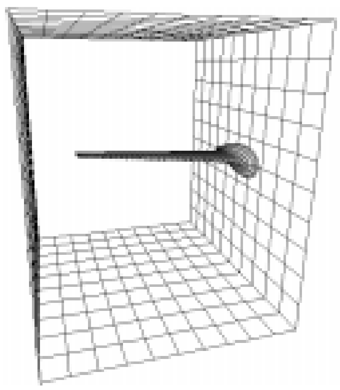

Fig. 4. Test wing DLR-F5 in a closed walls transonic wind tunnel

Experiments were carried out in the closed walls transonic wind tunnel, with all boundary and flow conditions given relevant for formulating the elliptic boundary value problem, including the location of experimentally observed transition from laminar to turbulent flow. Because of the occurring laminar separation - shock boundary layer interaction, numerical results simulating this flow were not very satisfactory and since then the test case remains to be a challenge for CFD. The data still are available [5].

In addition the case today can be generated with various airfoil, planform and wing twist modifications to serve as a varying shape for adaptation, optimization and unsteady processes simulation. Another option interesting here is a modelling and simple variation of the pressure distribution on this wing:

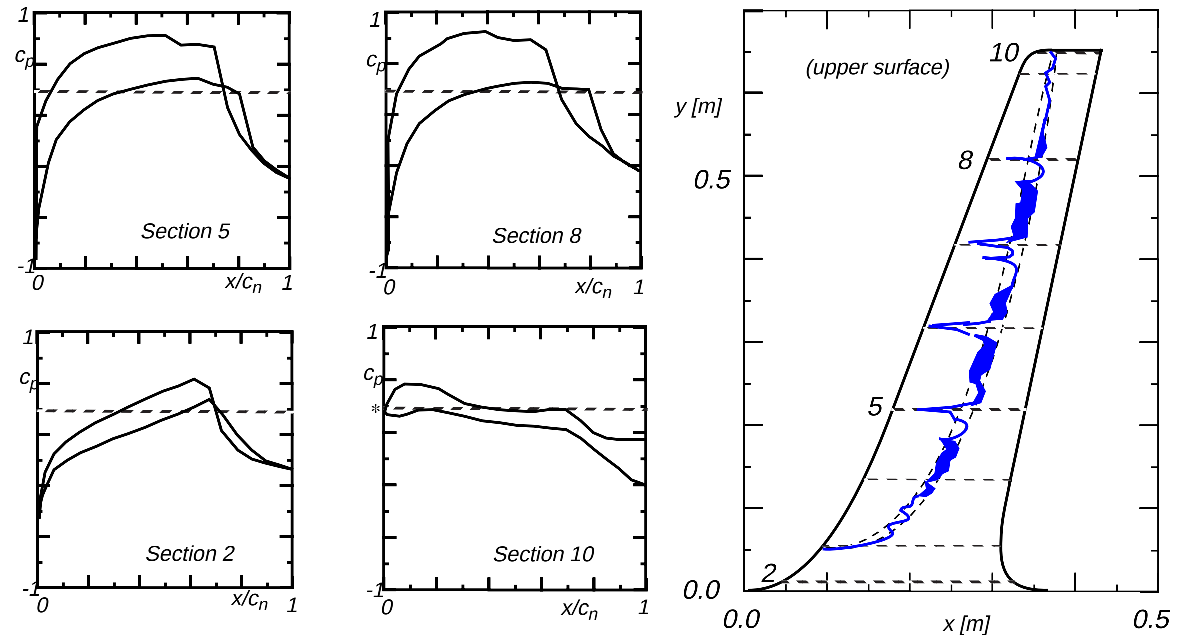

Fig. 5. Experimental pressure data on DLR-F5 wing in transonic tunnel at $M_1 = 0.82$, $\alpha = 2^o$ Re = 2 Mill., Free Transition, visualized and modeled.

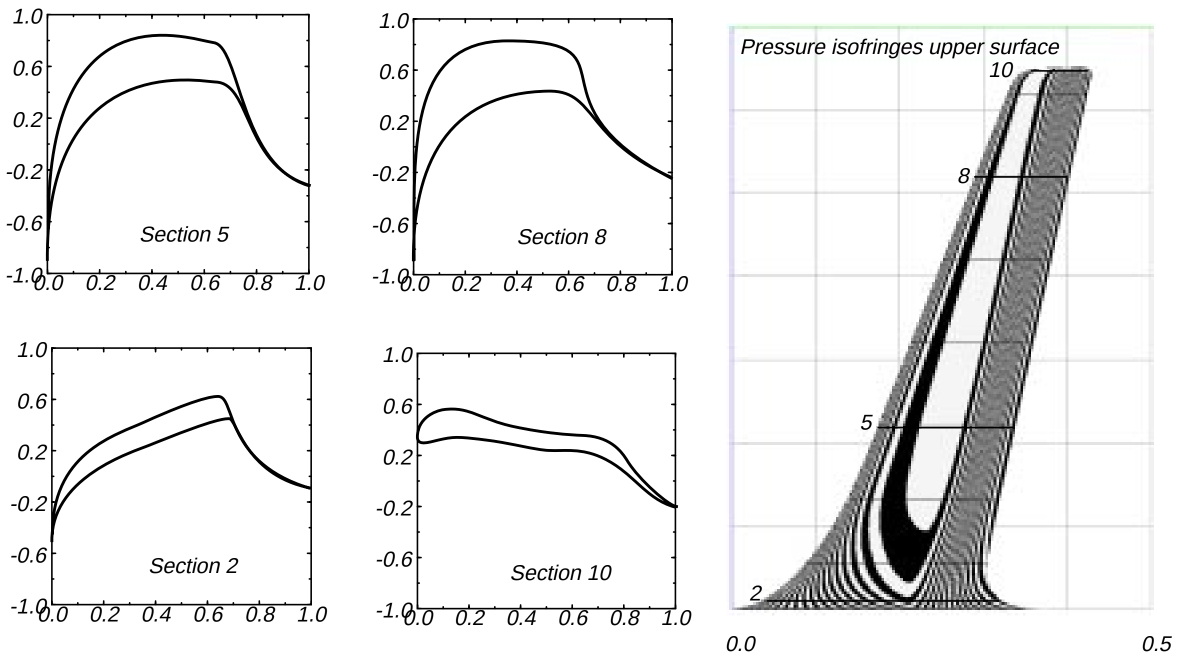

Fig. 6. Pressure coefficient distribution remodeled with geometry generator shape functions for inverse design input in free stream conditions $M_\infty \sim 0.82$

Test case for inverse design.

The original test case for CFD code developers provides all relevant input data for transonic viscous analysis in a rectangular channel with an inlet Mach number of 0.82, for the nonlifting case with angle of attack $\alpha = 0^o$ and a lifting case $\alpha = 2^o$ (Fig 5). These results have been obtained with free transition at a Reynolds number Re = 2Mill., based on a mean wing chord of 170 mm. The observed laminar separation - shock boundary layer interaction makes the test case a difficult one, as mentioned before. For redesign purposes, generated pressure data are provided, guided by the conditions of the experiment (especially in the wing root area), but ‘idealized’ based on 2D airfoil analysis and swept wing theory, see Fig. 6.

The idea is to use the test case for inverse design methods: a successful approach with perfect viscous analysis would confirm the wing geometry if the measured pressure distribution were used with the wind tunnel walls boundary condition. Using the idealized pressure distribution (or the wing geometry) with a realistic free-stream far field condition at $M_\infty = 0.82$ will result in a more or less different surface geometry (or different pressure distribution), using inverse or direct analysis, respectively.

In a first set of data complementing the earlier publication [5] we give the modelled pressure coefficients as a scalar in addition to the wing geometry along a dense set of span and section stations [6].

DLR-F9: CONFIGURATION FOR REFINED WING - BODY INTEGRATION STUDY

The second method to design junctions between given configuration components may be applied to more complex junctions where a simple projection of wing sections into one direction toward a fuselage will result in aerodynamically or structurally unwelcome shape details. Such a situation occurs in design of high wing configurations where it is desirable to shape the whole upper fuselage in strong integration with the wing.

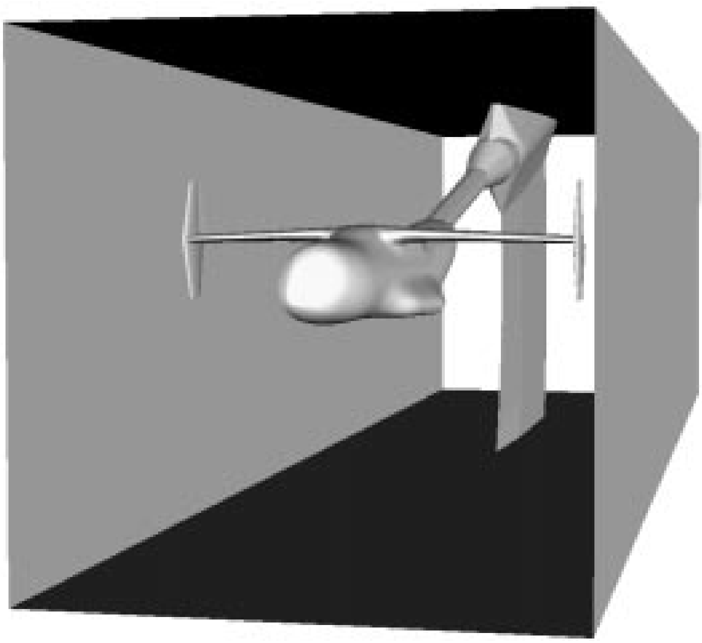

Fig. 7: DLR-F9 Model configuration for a refined experimental analysis of a generic High Wing aircraft wing root flow by adaptive components.

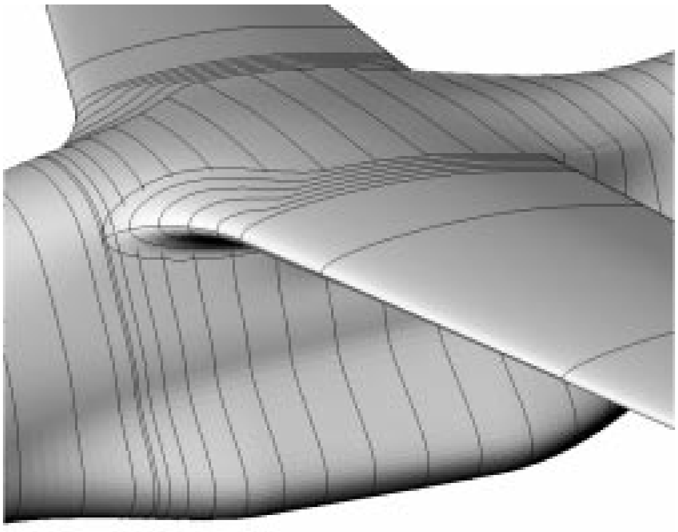

Fig. 8. Generic High Wing Transport aircraft: Shape details near the wing root. Fillet designed using new shape junction geometry tool.

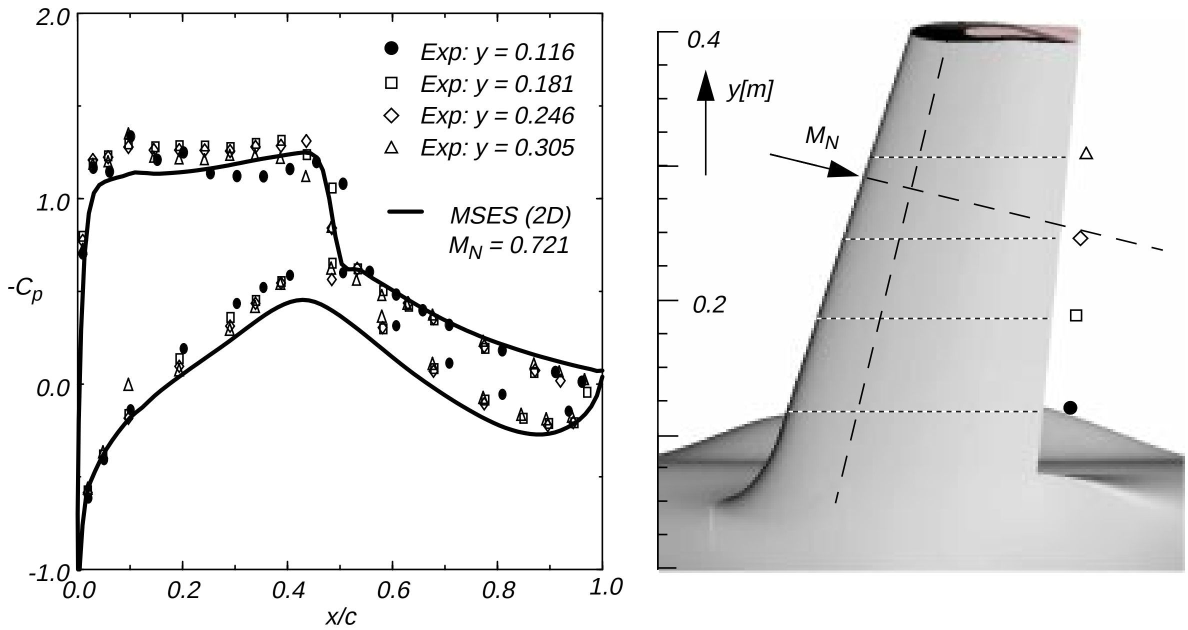

Fig. 9. Experimental results for DLR-F9 configuration in transonic wind tunnel at $M_\infty = 0.74$: Pressure measurements in 4 wing sections at optimally adjusted circulation control: pressure distribution and shock position close to isobar design concept of the wing.

Basic wing section used for 2D swept wing airfoil analysis using MSES (Drela) code.

Figure 7 illustrates the DLR-F9 configuration in the Göttingen transonic wind tunnel. Its purpose is manifold: Both new concepts for software development [3] and experimental techniques with large models and adaptive devices [7] are investigated. The high wing model with circulation control devices at the clipped wings was built and allows for a refined study of the flow quality in the area of the wing-body junction because of its larger size. In this generic airlifter configuration [8] we have the constraints of shaping a high wing in the presence of a complex fuselage, with cross sections to fit to an optimum use of the high lift system and accomodating the integration with the wing root fillet. Fig. 8 shows some details of this test case. The challenge was to integrate wing and body for an optimum load distribution, by variation of a relatively small number of certain shape parameters. The example shows that geometry preprocessing should allow to adapt both geometries, for the wing root and the body, in an optimization effort.

Experimental results are shown in Fig. 9 for the optimally adjusted circulation control device: Pressure measurements verify the isobar design concept and leave the shock position undisturbed in the inner portion of the wing. These results encourage a use of the clipped wings concept in refined investigations of the wing root area. In a subsequent experiment a shape module has been exchanged and later will be replaced by an adaptive component.

CONCLUSION

The two examples are intended to support the development of direct and inverse aerodynamic shape design concepts and also give an idea which are the relevant parameters for optimization of wing-body configurations. All knowledge, finally, must be represented in the geometry tool. New software, information and communication technology allow to make test cases like the presented ones available to any developer. An invitation to use them, with discussing the theoretical and practical geometric as well as aerodynamic aspects, is the purpose of this contribution.

REFERENCES

1. Dulikravich, G. S.: Aerodynamic Shape Inverse Design and Optimization Methods. CISM Courses and Lectures No. 366, ‘New Design Concepts for High Speed Air Transport’, Springer, Wien, NewYork (1997), pp. 159-200
2. Sobieczky, H.: Geometry Generator for Aerodynamic Design. CISM Courses and Lectures No. 366, ‘New Design Concepts for High Speed Air Transport’, Springer, Wien, NewYork (1997), pp. 137-158
3. Trapp, J., Zores, R., Gerhold, Th., Sobieczky, H.: Geometrische Werkzeuge zur Auslegung Adaptiver Aerodynamischer Komponenten. Proc. Deutscher Luft- und Raumfahrtkongress 1996, DGLR - JT96-081 (1996)
4. Kordulla, W. (Ed.), Numerical Simulation of the Transonic DFVLR-F5 Wing Experiment. Notes on Numerical Fluid Mechanics, Vol. 22, Vieweg, Braunschweig, (1988)
5. Sobieczky, H.: DLR-F5: Test Wing for CFD and Applied Aerodynamics. Case B-5 in: Test Cases for CFD Evaluation, AGARD FDP AR 303, (1994)
6. Sobieczky, H: Data Base for DLR-F5 Configuration Modifications. DLR Report in preparation (1998)
7. Zores, R., Sobieczky, H.: Using Flow Control Devices in Small Wind Tunnels, AIAA-97-1920 (1997)
8. Sobieczky, H., Geometry for Theoretical, Applied and Educational Aerodynamics. In: Computing the Future II, D. Caughey and M. Hafez (Eds.), John Wiley (1998)