Editors
E.H. Hirschel/München
W. Schröder/Aachen
K. Fujii/Kanagawa
W. Haase/München
B. van Leer/Ann Arbor
M. A. Leschziner/London
M. Pandolfi/Torino
J. Periaux/Paris
A. Rizzi/Stockholm
B. Roux/Marseille

New Results in Numerical and Experimental Fluid Mechanics V

Contributions to the 14th STAB/DGLR Symposium Bremen, Germany 2004

Hans-Josef Rath
Carsten Holze
Hans-Joachim Heinemann
Rolf Henke
Heinz Hönlinger
(Editors)

Springer

Prof. Dr. Hans-Josef Rath
Carsten Holze
ZARM - University of Bremen
Am Fallturm/Hochschulring
D-28359 Bremen

Rolf Henke
Airbus Deutschland GmbH
Senior Project Manager
Hünefeldstraße 1-5
D-28199 Bremen

Dr. Hans-Joachim Heinemann
Co-Managing Editor
„Aerospace Science and Technology, AST"
DLR
Bunsenstraße 10
D-37073 Göttingen

Professor Dr. Heinz Hönlinger
DLR - AE
Bunsenstraße 10
D-37073 Göttingen

Library of Congress Control Number: 2006923556

ISBN-10 3-540-33286-3 Springer Berlin Heidelberg New York
ISBN-13 978-3-540-33286-2 Springer Berlin Heidelberg New York

This work is subject to copyright. All rights are reserved, whether the whole or part of the material is concerned, specifically the rights of translation, reprinting, reuse of illustrations, recitation, broadcasting, reproduction on microfilm or in other ways, and storage in data banks. Duplication of this publication or parts thereof is permitted only under the provisions of the German Copyright Law of September 9, 1965, in its current version, and permission for use must always be obtained from Springer. Violations are liable to prosecution under German Copyright Law.

Springer is a part of Springer Science+Business Media
springer.com

© Springer-Verlag Berlin Heidelberg 2006
Printed in Germany

The use of general descriptive names, registered names, trademarks, etc. in this publication does not imply, even in the absence of a specific statement, that such names are exempt from the relevant protective laws and regulations and therefore free for general use.

Final processing by PTP-Berlin Protago-TEX-Production GmbH, Germany (www.ptp-berlin.com)
Cover-Design: deblik, Berlin
Printed on acid-free paper 89/3141/Yu – 5 4 3 2 1 0

NNFM Editor Addresses

Prof. Dr. Ernst Heinrich Hirschel
(General editor)
Herzog-Heinrich-Weg 6
D-85604 Zorneding
Germany
E-mail: e.h.hirschel@t-online.de

Prof. Dr. Kozo Fujii
Space Transportation Research Division
The Institute of Space
and Astronautical Science
3-1-1, Yoshinodai, Sagamihara
Kanagawa, 229-8510
Japan
E-mail: fujii@flab.eng.isas.jaxa.jp

Dr. Werner Haase
Höhenkirchener Str. 19d
D-85662 Hohenbrunn
Germany
E-mail: werner@haa.se

Prof. Dr. Bram van Leer
Department of Aerospace Engineering
The University of Michigan
Ann Arbor, MI 48109-2140
USA
E-mail: bram@engin.umich.edu

Prof. Dr. Michael A. Leschziner
Imperial College of Science
Technology and Medicine
Aeronautics Department
Prince Consort Road
London SW7 2BY
U. K.
E-mail: mike.leschziner@ic.ac.uk

Prof. Dr. Maurizio Pandolfi
Politecnico di Torino
Dipartimento di Ingegneria
Aeronautica e Spaziale
Corso Duca degli Abruzzi, 24
I-10129 Torino
Italy
E-mail: pandolfi@polito.it

Prof. Dr. Jacques Periaux
Dassault Aviation
78, Quai Marcel Dassault
F-92552 St. Cloud Cedex
France
E-mail: jperiaux@free.fr

Prof. Dr. Arthur Rizzi
Department of Aeronautics
KTH Royal Institute of Technology
Teknikringen 8
S-10044 Stockholm
Sweden
E-mail: rizzi@kth.se

Dr. Bernard Roux
L3M – IMT La Jetée
Technopole de Chateau-Gombert
F-13451 Marseille Cedex 20
France
E-mail: broux@l3m.univ-mrs.fr

Prof. Dr. Yurii I. Shokin
Siberian Branch of the
Russian Academy of Sciences
Institute of Computational
Technologies
Ac. Lavrentyeva Ave. 6
630090 Novosibirsk
Russia
E-mail: shokin@ict.nsc.ru

FOREWORD

This volume contains the paper presented at the 14th DGLR/STAB-Symposium held at the ZARM, Universität Bremen, Germany, November, 16 to 18, 2004. STAB is the German Aerospace Aerodynamics Association, founded towards the end of the 1970's, whereas DGLR is the German Society for Aeronautics and Astronautics (Deutsche Gesellschaft für Luft- und Raumfahrt - Lilienthal Oberth e.V.).

The mission of STAB is to foster development and acceptance of the discipline “Aerodynamics” in Germany. One of its general guidelines is to concentrate resources and know-how in the involved institutions and to avoid duplication in research work as much as possible. Nowadays, this is more necessary than ever. The experience made in the past makes it easier now, to obtain new knowledge for solving today's and tomorrow's problems. STAB unites German scientists and engineers from universities, research-establishments and industry doing research and project work in numerical and experimental fluid mechanics and aerodynamics for aerospace and other applications. This has always been the basis of numerous common research activities sponsored by different funding agencies.

Since 1986 the symposium has taken place at different locations in Germany every two years. In between STAB workshops regularly take place at the DLR in Göttingen. The changing meeting places were established as focal points in Germany's Aerospace Fluid Mechanics Community for a continuous exchange of scientific results and their discussion. Moreover, they are a forum where new research activities can be presented, often resulting in new commonly organised research and technology projects.

It is the fifths time now that the contributions to the Symposium are published after being subjected to a peer review. The material highlights the key items of integrated research and development based on fruitful collaboration of industry, research establishments and universities. Some of the contributions still present results from the "Luftfahrtforschungsprogramm der Bundesregierung (German Aeronautical Research Programme)". Some of the papers report on work sponsored by the Deutsche Forschungsgemeinschaft (DFG, German Research Council) in some of their Priority Programs (Verbundschwerpunkt-Programm) as well as in their Collaborative Research Centres (Sonderforschungsbereiche). Other articles are sponsored by the European Community and are therefore results of cooperation among different organisations. The main areas include numerical simulation and mathematics, aeroelasticity, small and large aspect ratio wings in the context of leading-edge vortices, wake vortices, high lift systems and propulsion integration, and new developments in wind tunnel facilities and measurement techniques. Therefore, this volume gives an almost complete review of the ongoing aerodynamics research work in Germany. The order of the papers in this book corresponds closely to that of the sessions of the Symposium.

The Review-Board, partly identical with the Program-Committee, consisted of A. Altminus (München), G. Ashcroft (Köln), J. Ballmann (Aachen), R. Behr (München), Chr. Breitsamter (München), A. D'Alascio (München), J. Delfs (Braunschweig), G. Eitelberg (Emmeloord), M. Fischer (Bremen), R. Friedrich (München), R. Grundmann (Dresden), A. Gülhan (Köln), H. Hansen (Bremen) , S. Haverkamp (Aachen), St. Hein (Göttingen), P. Hennig (Unterschleißheim), F. Holzäpfel (Weßling), H. Hönlinger (Göttingen), G. Koppenwallner (Katlenburg-Lindau), W. Kordulla (Noordwijk), H. Körner (Braunschweig), E. Krämer (Stuttgart), H.-P. Kreplin (Göttingen), N. Kroll (Braunschweig), D. Kröner (Freiburg), J. Longo (Braunschweig), Th. Lutz (Stuttgart), F. Menter (Otterfing), E. Meyer (München), C. Naumann (Stuttgart), G. Neuwerth (Aachen), W. Nitsche (Berlin), H. Olivier (Aachen), R. Radespiel (Braunschweig), H.-J. Rath (Bremen), U. Rist (Stuttgart), H. Rosemann (Göttingen), R. Schnell (Köln), G. Schrauf (Bremen), W. Schröder (Aachen), D. Schwamborn (Göttingen), J. Sesterhenn (München), E. Steinhardt (München), Chr. Stemmer (Dreseden), P. Thiede (Bremen), J. Thorbeck (Berlin), C. Tropea (Darmstadt), R. Voß (Göttingen), C. Wagner (Göttingen), C. Weiland (München), C. Weishäupl (München), and W. Würz (Stuttgart). Nevertheless, the authors sign responsible for the contents of their contributions.

The editors are grateful to Prof. Dr. E. H. Hirschel as the General Editor of the "Notes on Numerical Fluid Mechanics and Multidisciplinary Design" and to the Springer-Verlag for the opportunity to publish the results of the Symposium.

H.-J. Rath, Bremen
C. Holze, Bremen
H.-J. Heinemann, Göttingen
R. Henke, Bremen
H. Hönlinger, Göttingen

October 2005

CONTENTS

Page

High Aspect-Ratio Wings

Low Aspect-Ratio Wings

Contents (continued)

Contents (continued)

Page

Laminar Flow Control and Transition

Active Flow Control

Contents (continued)

Page

Hypersonic Flows

Aeroelasticity

Aeroacoustics

Contents (continued)

Contents (continued)

High Reynolds-Number Windtunnel Testing for the Design of Airbus High-Lift Wings

Daniel Reckzeh[1], Heinz Hansen[2],
AIRBUS, Aerodynamic Design & Data Domain,
D- 28183 Bremen, Germany

Summary

Present aircraft design methods must be continuously improved. This is due to environmental and future transport market requirements. Promising results are expected from development of advanced high lift systems for civil transport aircraft. At present the validation of the numerical design of a high lift / low speed system is done with subscale aircraft models. The final performance of the new aircraft is derived by extrapolation. This extrapolation to a full-scale aircraft poses a high risk in time and money. The accurate prediction of low speed / high lift performance early in the design process will, therefore, be a key asset in the development of competitive future aircraft. Consequently the Airbus approach for future design verification concentrates on earlier design verification by the use of high-Re facilities, especially the ETW. The experience gained in the qualification and development phase of low-speed testing in ETW via R&T programmes (as EUROLIFT) will be exploited in future in aircraft development work by early use of high-Re benchmark testing in close combination to the design work with advanced CFD-tools.

Drivers for high-Reynolds-number windtunnel testing for design verification

Within the scope of constant pressure to improve aircraft products concerning costs, performance, reliability and emissions, the development and production of advanced high lift systems for new or modified aircraft will play an important role in the future [1]. Design methods for such high lift systems have to be continuously improved due to these environmental and future transport market demands. A crucial prerequisite for the design of efficient and competitive aircraft is a comprehensive understanding of the flow physics of such systems, as well as the ability to optimise these systems in terms of more efficient, yet simpler designs. This, in combination with a high accuracy of flight performance prediction in an early stage of development will provide a strong contribution to the competitiveness of European aircraft manufacturers. Accurate flight performance prediction is a challenging task, this is due to the fact that most of the high lift testing to date has been done at sub-scale conditions [2]. Field performance and handling qualities for the aircraft are then derived by extrapolation. Many of these

[1] Head of high-lift devices group, Daniel.Reckzeh@airbus.com
[2] Senior research engineer, high-lift devices group, Heinz.Hansen@airbus.com

scaling effects strongly depend on the Reynolds number as the characteristic parameter between subscale and flight conditions [2, 3, 4]. These scaling effects can introduce an element of risk to the aircraft programme, particularly for large wings, which are designed for high subsonic Mach numbers. This could possibly require expensive design modifications to be made during flight tests. On the other hand the high lift design procedure includes extensive parametric studies at the project design stage, where time is an important factor. Optimisation and refinement are required and wind tunnel tests at reasonably high Reynolds numbers can result in a drastic increase in development costs, if undertaken solely in the wind tunnel. Therefore the combination of experimental and suitable computational investigations is increasingly necessary. The current situation of the aforementioned areas of interest can be summarised as follows:

- Many of the high lift flow phenomena, especially for 3D configurations and at high Re numbers are not fully understood [2, 8].
- Most of the high lift testing until to date has been done at low or moderate Re numbers (Re<7 Mio.).
- Significant adverse scale effects can sometimes be found in flight-testing, but no detailed measurements are available for the detailed understanding of these effects.
- Only a very limited number of tests are available in Europe at flight Re numbers and free flight conditions.
- Scale effects on stall data depend ultimately on some details of the local 3D geometry and the spanwise stall behaviour can be strongly affected by the local Re number.

The need for advanced complex CFD-methods in combination to high-Re testing

Rapid viscous-inviscid interaction codes are a standard industrial tool in the development of high lift systems. However, the extremely complex high lift flows and the need for improved prediction accuracy have led to increased effort of introducing Reynolds averaged Navier-Stokes (RANS) codes as a tool for high lift application. For complex flows with strong separation - even in 2D- an improvement of accuracy for industrial applications in terms of prediction of maximum lift, stall behaviour and for take-off configurations the accurate drag is needed in addition [5]. Common research projects have provided 2D RANS codes for industrial applications. Verification has been accomplished for 3D pilot applications and is currently extended [6]. The challenge of grid generation could be tackled by using unstructured grid approaches, but reliable prediction of all involved flow phenomena requires considerable future effort. The computational results have to be improved for cases with strong separated flows. Previous research activities [2, 7] have shown the importance of the accurate prediction of transition on the high lift elements. The accuracy of computations will have to be driven forward by improved modelling of the main flow phenomena and therefore basic research experiments with detailed flow measurements -especially in 3D- are needed to provide the code developer with the necessary information.

Contributions of the EUROLIFT R&T programme

The role of EUROLIFT [9] can be understood as a qualification phase for the approach to test high-lift design solutions derived from design assessment with complex CFD methods. The main tasks of EUROLIFT were:

- Preparation of an experimental database with detailed flow field information providing numerical code developers with a comprehensive set of information with step-by-step increase of geometric and flow complexity.
- Assessments of state-of-the-art European high lift codes with a clear path to industrial application by using this experimental database.
- Improvement of numerical tools in fields with clear shortcomings and derivation of common European guidelines for the further improvement and development of high lift tools with respect to time effective pre-optimisation and the high accuracy required for industrial applications.
- Extensive use of the unique capabilities of the European cryogenic Transonic Windtunnel (ETW) for high Reynolds number testing at low speed/high lift to bridge the gap between sub-scale testing and flight conditions.
- Application of the improved knowledge for testing of an advanced high lift system up to flight Reynolds numbers and demonstration of the improvement potential.

As far as possible existing results and experiences of different national and European research activities have been used. The CFD tools developed on a national basis have been brought into the EUROLIFT community. A common set of well-defined experiments has given the unique possibility to compare most of the existing high lift CFD tools within Europe. This will lead to a standardised European guideline for fast grid generation and accurate transition and turbulence modelling tools. The integration and application of these tools in the industrial high lift design process has been significantly accelerated.

With the availability of the ETW test facility it is possible to achieve flight Reynolds numbers. Within EUROLIFT this tunnel has been used for the first time for low speed testing of half span models in high lift configuration up to flight conditions. All partners within EUROLIFT could benefit from the assessment and exploitation of this unique European testing facility. Each partner was involved in this process and gains the common knowledge about the advantages of such a tunnel.

EUROLIFT ETW-High-Re verfication procedure of an advanced high-lift system

Within EUROLIFT the European high Re test facility ETW and its unique possibility to use both - pressurized and cryogenic conditions- in combination with cryogenic half model test technique, it was possible for the first time to perform 3D low speed high lift measurements up to flight Re numbers in a wind tunnel [14].

Therefore the approach taken with the KH3Y validation model was testing of selected configurations and high lift element settings over the complete range of Re numbers starting with low Re numbers typical for smaller wind tunnels up to real flight Re numbers. Before these tests with the validation model could be performed it must be assured that this new low speed high lift testing with a newly designed half model balance could be performed in the ETW with a high quality standard. Therefore a pre-testing for the verification of ETW for high lift, high Re testing with half span models was performed. This was an important milestone for the following tests in ETW. For this verification process an existing cryogenic model of Airbus-D (Fig.1) was used as an example of a modern transport aircraft. This model has been extensively tested in the LSWT in Bremen and in the cryogenic non-pressurised tunnel in Cologne (KKK). Based on this information the results from the ETW could be cross checked in the appropriate Ma/Re limits under cryogenic conditions [13, 15]. By use of global flight test results of this aircraft (max. lift, stall behaviour) an additional comparison at flight Re numbers between the new ETW results and flight was possible. This comparison demonstrated a good agreement of the ETW results with other test facilities and the existing flight test results. A detailed overview about these tests has been given in [15]. After this verification phase corresponding to the low Re number validation test in the LSWT a high Re number ETW test for selected configurations has been performed and could provide a database for the assessment of CFD codes up to flight Re numbers. Several configurations have been tested in the ETW up to flight Re numbers: a clean configuration with and without a modified leading edge (behaviour of leading edge stall with Re), a take-off and landing configuration, and a landing configuration with different flap settings. Due to the cryogenic conditions no detailed flow field measurements could be performed but for selected points the boundary layer transition has been observed by I.R detection technique. For some interesting areas and conditions in addition to this the deformation of the flap and the wing bending as a function of the changeable stagnation pressure was measured by a mini CCD camera system.

In a last step the validation model KH3Y (Fig.2) was used in a more realistic complex aircraft configuration as an example of an typical industrial application. Use was made of a so-called multifunctional flap system (Fig.3). With this system an extended fowler flap will replace the aileron. The 'full-span' fowler is equipped with a small splitflap ("Gurney-flap" or "mini-TED") or a camber tabs over the total fowler flap span. These tab elements can be deflected differentially in span-wise direction and used for increased maximum lift, optimising the lift distribution in take-off, roll- and glide path control, as well as for the improvement of cruise performance. This new high lift system was developed and extensively tested by Airbus-D with a large full span model. Corresponding to the KH3Y model geometry at low Re numbers (Re=3 Mio.) within the national German high-lift technology programme HAK [12] was build by Airbus-D within EUROLIFT. In high lift configuration at high deflection angles of the camber tab the effectiveness of the tab is limited by separation at low Re numbers. Tested in the last phase of EUROLIFT tests in ETW the main objective therefore was the assessment of such a system over the complete range of Re numbers from sub-scale testing up to flight

Re numbers by use of the ETW test facility. These measurements helped to improve the understanding of scaling effects and to evaluate design criteria for such a system as a function of Re number. As an example of the significant impact of the Re number on the obtained performance the development of $C_{L,max}$ vs Re is shown in Fig.4. The development of maximum increment lift over Reynolds-number is given for advanced different configurations (extended flap, splitflap and differentiated tab deflections) relative to the reference single slotted flap & aileron layout. The benefit from the advanced flap systems is significantly higher at high Re conditions and also the relation between the layout changes. This example indicates very clearly that a design selection at low Re conditions could have let to a non-optimised solution at flight conditions.

Experience from the A380 design work

The targeted approach with establishing a high-Re 'benchmark' very early in the design process could not yet be applied for A380 [16]. Nevertheless a large amount of tests at 'medium-to-high-Re conditions' was already conducted during the development work. Windtunnels as DNW-KKK, Onera F1 and Qinetiq Q5m provided already Reynolds-numbers up to Re=12 Mio. These tests revealed significant impact of the Reynolds-number on an optimum solution for various design features as the

- Maximum lift & drag performance of various leading edge layouts (slat, droop-nose device, sealed slat, beret-basque-pylon)
- Spanwise combination of slat angles and maximum slat angles
- Slat-setting (lift optimised for landing, drag optimised for take-off)
- Flap-setting (lift optimised for landing, drag optimised for take-off)
- Extension and profile of the unprotected inner wing leading edge
- Winglet efficiency on take off drag

Nevertheless, due to the qualification work for the ETW application for low-speed testing in EUROLIFT which was conducted in parallel to the A380 development process, an ETW check-out campaign for performance risk-mitigation could be finally included in the A380 development work. In this approach a cryogenic halfmodel with the specific wing twist of the landing configuration was manufactured and tested in ETW. It gave very valuable contributions concerning the maximum lift level and the wing separation behaviour under flight conditions, as well as insight in the impact of aeroelastic distortion effects on the high-lift performance with the help of measurements of the deformed wing shape under different windtunnel pressure conditions.

Requirements to high-Re windtunnel facilities in industrial application

From the extensive experience in windtunnel testing for design verification at Airbus following top-line requirements for high-Re facilities can be formulated.

- Flexibility
    - Fast configuration changes, low testing time per configuration

    - Suited for significantly different configurations (e.g. A380, A400M)
    - Engine simulation (high importance esp. for propeller-configurations)
- Quality of results & experimental techniques
    - Good repeatability (short-term & long term)
    - Reliable online results in standardized format
    - Good flow & separation visualisation (tufts, oil, accenaphtene, PSP)
    - Transition visualisation (Infrared, TSP)
    - Deformation measurement to provide information on tested model wing shapes under load (to rule-out "pseudo Re-effects")
- Compatibility to other windtunnels in the 'verification chain'
    - Identical models to be used
    - Identical data procedures & correction methods
- Reasonable relation between cost & effort relative to amount of results & quality
    - For several tasks many variations at low/med Re may be more reasonable than little at high Re-conditions to provide the required amount of data under given time & budget constraints.

Especially the last point can be considered of key importance as the cost of high-Re testing severely conflicts with the usual budget requirements of industrial development programmes.

Conclusions

It is obvious that for further improvement of high-lift configurations' performance by application of advanced system solutions or also simply by better exploitation of state of the art systems by reduction of performance margins to cover the uncertainties in tunnel to flight scaling the experimental verification at flight conditions has to play a central role. The ETW is the only facility where these conditions can be reached. After recent R&T work -especially in EUROLIFT- the ETW can be considered now as qualified for low speed testing and a first application in an aircraft programme has been taken place with an A380 high-lift campaign. Nevertheless, further work is necessary to establish the ETW as a 'robust & known' part in the Airbus windtunnel verification chain. Recent R&T programmes as IHK/HICON, EUROLIFT 2 and FLIRET shall serve this purpose.

In combination to early high-Re testing to provide benchmarks for a new configurations' performance capabilities advanced CFD-tools are required to conduct the detailed design work. It has to be pronounced that these codes have to become more mature, robust and user-friendly than today's CFD-suites, even if impressive developments have been taken place in the last years which led to the establishment of 2D and 3D RANS-codes as integrated analysis tools in the design process and already first cases where design decisions were solely taken based on CFD-results.

The future approach has to include a proper mix of high-Re benchmark testing, CFD-based design assessment and low/medium-Re testing to capture larger amounts of configuration variations and also testing techniques, which are not (yet) applicable at high-Re in ETW (e.g. power simulation).

References

[1] Flaig, A.; R. Hilbig; High-Lift Design for large civil Aircraft; High-Lift System Aerodynamics, CP-515; AGARD, Sept.1993, pp. 31/1-12.

[2] Haines, A.B; Scale Effects on Aircraft and Weapon Aerodynamic; AGARD AG-323, July 1994

[3] Termes P., *et al.* ; Reynolds and Mach Number effects and the 2D-3D correlation based on measurements and computed results for the Garteur take-off configuration; CEAS Forum on High-Lift and Separation Control, University of Bath (UK), March 1995

[4] Fiddes, S.P., Kirby, D.A., Woodward, D.S., Peckham, D.H.; Investigations into the Effects of Scale and Compressibility on Lift and Drag in the RAE 5m Pressurised Low-Speed Wind Tunnel; Aeronautical Journal, Vol. 89, March 1985

[5] Lindblad, I.A.A., de Cock, K.M.J.; CFD prediction of Maximum Lift of a 2D High Lift Configuration; AIAA-99-3180;

[6] Becker, K., Kroll, N., Rossow, C.C., Thiele, F; MEGAFLOW - A numerical Flow Simulation System; ICAS Paper 98-2.7.4, ICAS Congress 1998; Melbourne, September 1998

[7] Arnal, D., Casalis, G., Reneaux, J., Cousteix, J. ; Laminar-Turbulent Transition in Subsonic Boundary Layers : Research and Applications in France; AIAA Paper 97-1905

[8] Delery, J.M., Separation in Two and Three-Dimensional Flows: Still a Subject of Mystery ?; VKI, The Jean J. Ginoux Lecture, 1998

[9] Thiede, P.; EUROLIFT – Advanced High Lift Aerodynamics for Transport Aircraft; AIR&SPACE EUROPE, Vol. 3 No 3+4, 2001

[10] Dargel, G., Schnieder, H.; GARTEUR High Lift Action Group (AG 08) Final Report; GARTEUR TP 043, Nov.1989

[11] Thibert, J.J.; The Garteur High Lift Research Programme; High-Lift System Aerodynamics, CP-515, AGARD, Sept.1993

[12] Hansen, H.; Überblick über das Technologieprogramm Hochauftriebskonzepte (HAK); DGLR Jahrestagung, Bremen 1998

[13] Quest, J.; ETW – High Quality Test Performance in cryogenic environment; AIAA-00-2206

[14] Wright, M.C.N.; Development of the Half Model Testing; Capability at ETW; ICAS 2000

[15] Quest, J.; Hansen, H.; Mesuro, G.G.; First Measurements on an Airbus High Lift; Configuration at ETW up to Flight Reynolds Number; AIAA-2002-0423

[16] Reckzeh, D; Aerodynamc Design of the High-Lift Wing for a Megaliner Aircraft; Aerospace Science & Technology; Oct 2002

Figure 1: K3DY Airbus model

Figure 2: KH3Y Eurolift model

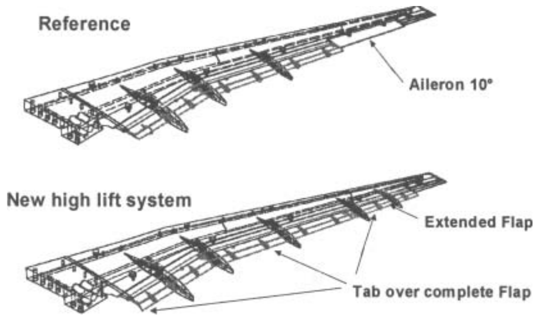

Figure 3: New Flap/Tab system on model KH3Y for test in ETW
Reference: Single Slotted Flaps and drooped aileron
New high-lift system: Extended Single Slotted Flaps with Tabs

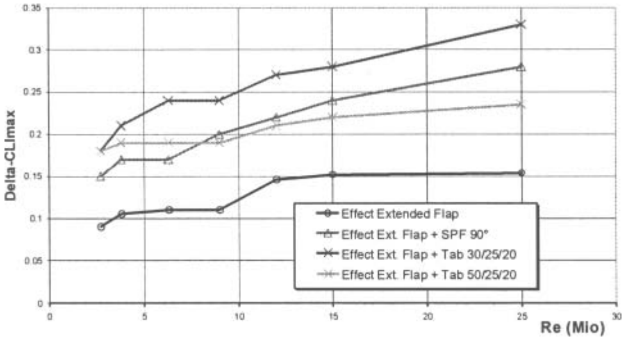

Figure 4: ETW-Results for maximum lift dependency vs. Reynolds-number of different flap system layouts relative to reference single slotted flap (SPF=splitflap, Tab=spanwise differentiated tabs on single slotted flap) (Reference for Reynolds-number: mean wing chord)

Aerodynamic Analysis of Flows with Low Mach- and Reynolds-Number under Consideration and Forecast of Transition on the Example of a Glider

S. Melber-Wilkending[1], G. Schrauf[2], and M. Rakowitz[1]

[1] German Aerospace Center (DLR), Institute of Aerodynamics and Flow Technology
Lilienthalplatz 7, D-38108 Braunschweig, Germany
[2] AIRBUS Deutschland GmbH, D-28183 Bremen, Germany
EMail: Stefan.Melber@DLR.de, Geza.Schrauf@AIRBUS.com, Mark Rakowitz@DLR.de

Summary

For the simulation of low Reynolds- and Mach number flows, a boundary layer code in combination with a database method is coupled with an unstructured RANS code. The technique is validated in 2D using a glider airfoil and in 3D for a glider, showing good agreement with experiments and a well-validated 2D design code.

1 Introduction

The aim of the study at hand is to extend the range of applicability of the DLR TAU-Code [1] for flows at low Reynolds- and Mach numbers. This flow regime is characterised by significant portions of laminar and transitional flow. Therefore, the numerical simulation of this flow regime must include the modelling of transition. The modified TAU-Code can then be used for a wide range of simulations in which laminar flow and transition are of importance, e.g. general aviation, wind turbines and the simulation of models in the wind tunnel. Afterwards TAU will also be used for design tasks like the numerical optimisation of a glider winglet and the wing-root fairing optimisation of the new SB15 glider. This paper describes the development of a process-chain for this purpose, using the boundary layer code COCO [3] and a data base method [4] in combination with TAU. Validation results are presented for a 2D glider airfoil case and in 3D for the simulation of complete speed polars of the glider "Std. Cirrus".

2 Geometry / Grids

For validation in 2D the HQ17 [5] glider airfoil by Horstmann and Quast is used. Four grids are generated with the hybrid unstructured grid generator Centaur [6] with a thickness of the boundary layer (35 layers) which is adapted to four Reynolds-numbers ($Re = 0.7, 1.0, 1.5, 2.0 \cdot 10^6$). The hybrid 2D grid for $Re = 1.5 \cdot 10^6$ is depicted in fig. 1 (left). A 15m "Std. Cirrus" glider by Schempp-Hirth is used for

3D validation. The geometry used here is equivalent to the first version of this glider (1969) with a wing twist of $-0.75°$. The computational geometry is build using CATIA V.5 based on original drawings, the airfoil coordinates and measurements of an original "Std. Cirrus" glider. The geometry includes the wing with wing tip, the fuselage and the wing-root fairing. The empennage is not included. As only symmetrical flow cases are investigated, the half-geometry is meshed using Centaur. The hybrid mesh contains $5.5 \cdot 10^6$ points overall with 20 prism layers for the resolution of the boundary layer. The volume between the outer prism surface and the farfield is filled with tetrahedra. The geometry and grid in the wing-root area are shown in fig. 1 (right).

3 Flow Solution Method

3.1 DLR TAU-Code

The solution of the Reynolds-averaged Navier-Stokes equations (RANS) is carried out using the hybrid unstructured DLR TAU-code [1]. For the closure of the Reynolds-averaged equations the k-ω-SST turbulence model of Menter [7] is used, which combines robustness with the applicability for partly detached flows. Due to the low Mach-numbers and the resulting stiffness of the RANS equations, low Mach-number preconditioning is used [8]. Finally, the central JST-scheme [9] in combination with 80% matrix dissipation [10] assures numerical flow solutions with low numerical dissipation.

3.2 Transition Prediction Method

The concept of the presented transition prediction method is to couple a RANS flow solver, a boundary layer code and a database method for Tollmien-Schlichting waves to predict transition. The main advantage of this method is the usage of "industrial grid densities" for the RANS flow solver, which means a typical grid density (e.g. $\approx 6 \cdot 10^6$ points for a glider) is adequate to get the parameters of the laminar boundary layer from the boundary layer code. To calculate these parameters directly in the RANS flow solver an extremely fine grid (e.g. $\approx 30 \cdot 10^6$ points for a glider) would be needed to achive sufficient resolution of the boundary layer. For complex 3D-configurations this demand would lead to an extensive number of grid points. For this reason the shown coupling of a RANS flow solver and a boundary layer code allows the usage of transition prediction for complex configurations with only small additional costs. The transition prediction method (boundary layer code and data base) is used as an external program. This allows a flexible coupling with any structured or unstructured RANS flow solver (e.g. FLOWer, TAU at DLR) and independent development of the transition prediction method.

In this paragraph the details of the complete process-chain are described: First a RANS flow solver is used to compute a surface pressure distribution for a given configuration based on a fully turbulent flow. Cutting this pressure distribution in planes with a tool of Wilhelm [2], a 2D-flow can be extracted. This method supposes a plane aligned with the streamlines at the upper end of the boundary layer. This situation is found on typical stretched wings whereas e.g. on the wing/fuselage junction this assumption is invalid and the shown method cannot be used there. Based on the pressure distribution in cuts through the flow field, a 2D coordinate system is derived, the pressure distribution is transformed in this coordinate system and splitted at the stagnation point. Two filters for the reduction of local noise and local point clustering provoked by the cutting of unstructured grids are applied on the pressure distribution. Then a boundary layer code - described below - simulates the laminar boundary layer starting from the stagnation points in these cuts.

Two kinds of transition types must be treated in such a 2D-cut: First, at a point of the pressure distribution where laminar boundary layer is inhibited, e.g. on laminar separation bubbles. At such a point the boundary layer code cannot converge to a solution and transition must occur. The second kind of transition is driven by Tollmien-Schlichting waves. This type of transition is detected with a database method (see below) which uses the boundary layer parameters from the boundary layer code and gives an N_{TS}-factor. If $N_{TS} > N_{crit}$, transition will occur. The point of transition is the point which is closer to the stagnation point in a 2D-cut. Afterwards, the transition point is transformed back into the coordinate system of the flow solver. After some iterations the transition points are converged to the final location.

COCO Boundary Layer Code The boundary layer code COCO [3] computes the velocity and temperature profiles of a compressible laminar boundary layer along a swept and conical wing. It uses a new transformation that maps a thickening boundary layer onto a rectangular region, which facilitates the computation of streamwise derivatives needed for non-local stability calculations. The equations are expressed in terms of four coefficients: the acceleration of the inviscid flow driving the boundary layer, the conicity of the wing, the viscosity, and the boundary layer suction.

The equations are discretized with a fourth-order compact scheme ("Mehrstellenverfahren") and the resulting system is solved by a Newton method. One of the features of COCO is its consistent computation of all streamwise derivatives. The user can choose between several possibilities and the chosen streamwise discretisation is then used to solve the differential equation for the velocities at the boundary layer edge, to compute the coefficient of acceleration, to solve the parabolic boundary layer equations, and to compute the streamwise derivatives needed for non-local stability computations. As with every direct method, the boundary layer equations become singular at separation and the Newton iteration fails to converge. This behavior can be used as an indicator for separation.

Data Base Method for Tollmien-Schlichting Waves For a given station in a two-dimensional, incompressible boundary layer, one can present the local spatial amplification rates with the help of level lines in an (F,Re)-diagram, where F is the (non-dimensional) reduced frequency $F = 2\pi f\ \nu/u_e^2$ and Re the Reynolds number $Re = u_e \delta_1/\nu$. If we consider a fixed reduced frequency F, the negative value of the spatial amplification rate σ becomes a function of the Reynolds number Re alone. This function can be approximated by two half-parabolas ($i = 0 : Re_0 \leq Re \leq Re_M; i = 1 : Re_M \leq Re \leq Re_1$) with zeros at Re_0 and Re_1, and the maximum σ_M at Re_M:

$$\frac{\sigma}{\sigma_M} = 1 - \left[\frac{Re - Re_M}{Re_i - Re_M} \right]^2 , \ i = 1,2 \tag{1}$$

Arnal [11] showed that the four parameters defining the two half-parabolas can be expressed as functions in terms of the reduced frequency F, the shape parameter H_{12}, and the local Mach number M_e at the boundary layer edge. Thus, having performed a boundary layer calculation, we can, for a given frequency, compute the approximate local amplification rates for a Tollmien-Schlichting wave with this frequency and integrate these rates to obtain the N_{TS}-factor. A version has been implemented into the boundary layer code COCO [4].

4 Results

4.1 Airfoil HQ17

In this section the validation of the presented transition prediction method based on the airfoil HQ17 is presented. For comparison, measurements in two different low-speed wind tunnels [5] and a simulation with XFoil [12], a well validated analysis and design program for low-Re number flows, are used. Polars are simulated for four Reynolds numbers: $Re = 0.7, 1.0, 1.5, 2.0 \cdot 10^6$.

As an example in figure 2 for $Re = 1.5 \cdot 10^6$ a comparison is shown. On the right side the transition positions on the airfoil with XFoil and COCO are compared. As XFoil predicts a transition position on the end of the laminar separation bubble, whereas COCO predicts the begin of the bubble, the results of XFoil are modified with the bubble length. On the bottom side XFoil gives a transition position of about 5% behind that of COCO, which leads to a decreased drag, especially at higher angles of attack. The measurements and the TAU-results show a good agreement, whereas the XFoil results at higher angles of attack have an increased lift.

4.2 Complete Glider Configuration

As an example of the numerical simulation in combination with transition prediction of a flow with low Reynolds- and Mach-number, a glider configuration is considered. The aim is to calculate the speed-polar of a "Std. Cirrus" for the following free stream conditions: $H_{flight} = 1000\ m, T_\infty = 281.65\ K, p_\infty = 89875\ Pa$ and $\rho_\infty = 1.112\ kg/m^3$. To get a speed-polar, for every given speed of the flight envelope a corresponding lift coefficient is needed, which can be calculated from the assumption of aerodynamic lift equals weight of the aircraft:

$$c_A = \frac{2\ g\ m_{Glider}}{\rho\ V_\infty^2\ A_{Glider}} \tag{2}$$

and with the parameters of the glider $m_{Glider} = 316.7\ kg$, $A_{Glider} = 10.04\ m^2$ and $l_\mu = 0.67\ m$ the lift coefficients can be calculated. The angle of attack is calculated iteratively by the flow solver by matching the simulated lift coefficient with the given one. These parameters and the Reynolds-number can be found in table 1.

In figure 3 the streamlines and the simulated transition lines on the upper- and lower surface of the wing for $V_\infty = 100\ km/h$ and $V_\infty = 220\ km/h$ are depicted. At $V_\infty = 100\ km/h$ on the upper side a laminar separation bubble can be found on most of the wing. The transition is located nearly on the thickest point of the wing. At $V_\infty = 220\ km/h$ the transition line on the upper side has moved downstream because of the reduced angle of attack ($\alpha = -5.06^o$ instead of $\alpha = 0.288^o$, compare table 1) and leads to a reduced boundary layer load. The laminar separation bubble has disappeared at this speed. On the lower side the transition line has moved upstream compared to $V_\infty = 100\ km/h$. Especially on the outer wing this leads to a transition location near the leading edge of the wing.

In figure 4 (left) the flow field in the area of the wing/fuselage junction at a speed of $V_\infty = 90\ km/h$ is depicted. On the surface, streamlines are shown which demonstrate the laminar separation bubble on the upper wing surface. The transition occurs above this laminar separation bubble. Furthermore, on the trailing edge of the wing close to the fuselage a flow separation can be found which is noticeable in flight on a real "Std. Cirrus" as a rumbling noise at lower flow speeds. The horseshoe vortex around the wing on the fuselage can also be found in the figure. In the vorticity cuts this vortex is shown on the upper andon the lower side of the wing. Finally, a fuselage-vortex on the upper side of the fuselage (red) due to the stagnation line there is depicted.

The speed-polar of the Std. Cirrus is depicted in figure 4 (right). Because in the simulation the horizontal and vertical tail is not present, the viscous-, profile- and induced drag of the tail is estimated and added to the simulated speed polars. In comparison with flight measurements from the Idaflieg [13] the fully turbulent simulation has an increased drag, and sink-speed respectively. At $V \leq 90\ km/h$ in the simulation a flow separation on the upper wing takes place which is responsible for the drop-down of the speed polar. The main reason for this effect seems to be an over-prediction of flow separation due to the turbulence model. The same effect can be found for the simulation with transition. At $V > 90\ km/h$ the simulated speed polar has a reduced sink velocity compared to the measurement. Possible reasons can be a too small separation bubble drag because of a too coarse grid in this area and the missing induction of the horizontal tail on the wing, which would lead to an increased induced drag of the wing.

5 Conclusion / Further Work

In the presented paper a combination of a boundary layer code, a data base method and a RANS flow-solver is shown to be appropriate for the simulation of flows at low Mach- and Reynolds number under consideration of the laminar/turbulent transition. For the validation of this process-chain the airfoil HQ17 is used, which gives results in agreement with measurements in low Reynolds-number wind-tunnels and an analysis and design system for low Reynolds-number airfoils. The method is also applied for a glider configuration which shows good agreement with measurements.

The presented process-chain will be used for the optimisation of the wing / fuselage junction of a new glider (Akaflieg Braunschweig, SB15) and as a basis for the numerical optimisation of winglets for gliders. Furthermore, trimmed polars with complete gliders can be calculated (including the tail), the results of wind-tunnel measurements can be improved using transition in corresponding numerical simulations. Also, e.g. the efficiency of wind turbines can be improved using the laminar/turbulent transition for design.

Acknowledgments: The authors would like to thank R. Wilhelm (DLR) for the pressure distribution extraction tool and J. Himisch (Technical University of Braunschweig) for the assistance in the post-processing of the numerical simulations.

References

[1] Kroll, N.; Rossow, C.-C.; Becker, K.; Thiele, F.: *MEGAFLOW - A Numerical Flow Simulation System.* 21st ICAS congress, 1998, Melbourne, ICAS-98-2.7.4, 1998.

[2] Wilhelm, R.: *Ein inverses Verfahren zum aerodynamischen Entwurf von Triebwerksgondeln.* Forschungbericht 2004-30, DLR Braunschweig, 2004.

[3] Schrauf, G.: *COCO – A Program to Compute Velocity and Temperature Profiles for local and Nonlocal Stability Analysis of Compressible, Conical Boundary Layers with Suction.* ZARM Technik Report, November 1998.

[4] Schrauf, G.: *Implementation of Data Base Methods into a Boundary Layer Code.* EUROLIFT Technical Report TR 2.6-1, January 2003.

[5] Althaus, D.: *Windkanalmessungen am Profil HQ17/14.38 und Vergleich mit Messungen der University of Technology Delft sowie DFVLR.* Institut für Aerodynamik und Gasdynamik, Universität Stuttgart, Juli 1984.

[6] Kallinderis, Y.: *Hybrid Grids and Their Applications.* Handbook of Grid Generation, CRC Press, Boca Raton / London / New York / Washington, D.C., 1999.

[7] Menter, F.R.: *Zonal Two Equation k-ω Turbulence Models for Aerodynamic Flows.* AIAA-Paper 93-2906, 1993.

[8] Melber, S.; Heinrich, R.: *Implementierung einer Praekonditionierungstechnik für Strömungen kleiner Mach-Zahlen im DLR TAU-Code und Anwendung im Hochauftrieb.* 13th AG STAB/DGLR Symposium München, 12.-14. November 2002.

[9] Jameson,A. ; Schmidt, W.; Turkel, E.: *Numerical Solutions of the Euler Equations by Finite-Volume Methods using Runge-Kutta Time-Stepping Schemes.* AIAA Paper 81-1259, 1981.

[10] Turkel, E.: *Improving the Accuracy of Central Difference Schemes.* 11th International Conference on Numerical Methods in Fluid Dynamics, Lecture Notes in Physics, Vol. 323, pp. 586-591, 1988.

[11] Arnal, D.: *Transition Prediction in Transonic Flow.* IUTAM Symposium Transonicum III, DFVLR-AVA Göttingen, May 1988.

[12] Drela, M.: *XFOIL: An Analysis and Design System for Low Reynolds Number Airfoils.* Conference on Low Reynolds Number Airfoil Aerodynamics, University of Notre Dame, June 1989.

[13] *Vorläufige Ergebnisse der Flugleistungs- und Flugeigenschaftsvermessungen 1974 in Aalen-Elchingen.* DFVLR / Idafleg, 1974.

Table 1 Input for speed-polar, Std. Cirrus.

$V_\infty[km/h]$	70	75	80	90	100	120	150	220
$V_\infty[m/s]$	19.4	20.8	22.2	25.0	27.8	33.3	41.7	61.1
$c_A[-]$	1.472	1.282	1.127	0.890	0.721	0.501	0.321	0.149
$Re[10^6]$	0.824	0.883	0.941	1.06	1.18	1.41	1.77	2.59
$\alpha[°]$	14.5	8.68	7.58	2.16	0.288	-1.80	-3.54	-5.06

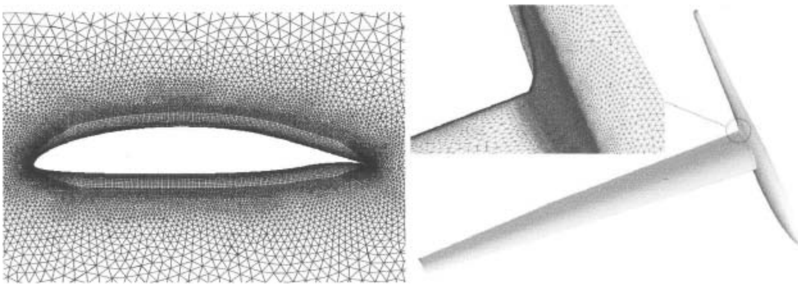

Figure 1 left: Airfoil HQ17, $Re = 1.5 \cdot 10^6$, right: Std. Cirrus and surface grid at wing/fuselage junction.

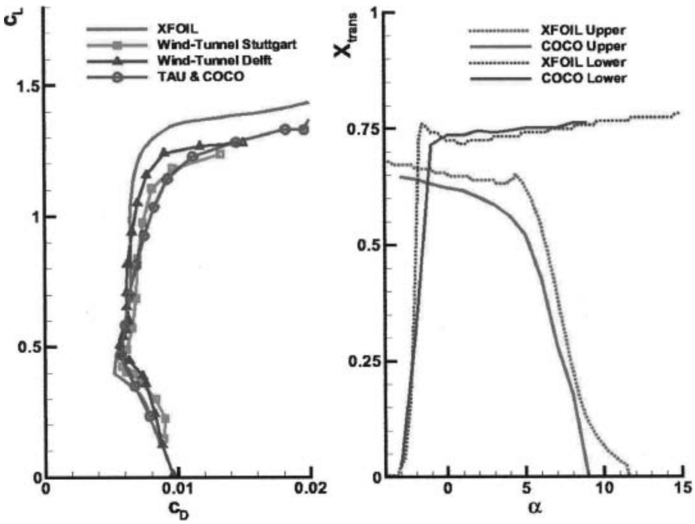

Figure 2 Airfoil HQ17, $Re = 1.5 \cdot 10^6$, comparison of measurement and simulation, polar and transition position.

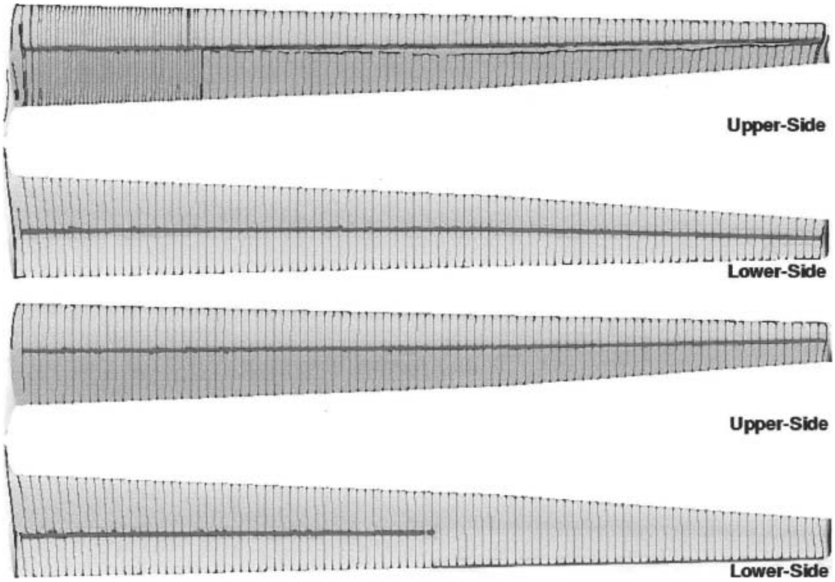

Figure 3 Streamlines on the wing surface, transition line (red) on upper and lower side, velocity: $V = 100\ km/h$ (both top), $V = 220\ km/h$ (both bottom), Std. Cirrus.

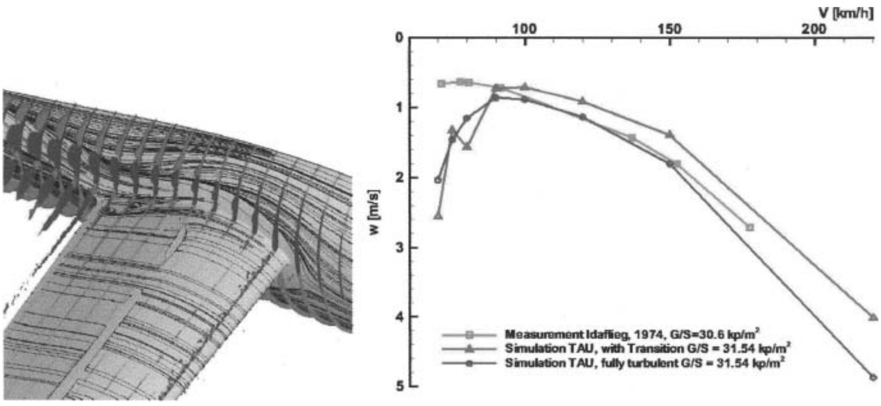

Figure 4 right: Flow field of Std. Cirrus in the area of wing/fuselage junction, $V_\infty = 90\ km/h$, surface stream-lines, vorticity in cuts, left:Speed-Polar of Std. Cirrus.

Analysis of PIV Flow Field Measurements behind the ALVAST-Model in High-Lift Configuration

H. Vollmers, W. Puffert-Meissner, and A. Schröder

DLR Institute of Aerodynamics and Flow Technology
Bunsenstr.10, 37079 Göttingen, Germany
Andreas.Schroeder@dlr.de

Summary

Flow field measurements with a stereoscopic PIV-system were performed behind the DLR-ALVAST half-model in a high lift configuration with and without an ultra-high bypass ratio engine simulator in the DNW-NWB Low-Speed Wind Tunnel Braunschweig at Mach numbers of 0.18 and 0.22 ($\approx$ 62 m/s and $\approx$ 76 m/s). In the plane behind the engine, the vortex position and strength strongly depended on thrust and angle of attack.

1 Introduction

An improved understanding of the viscous and induced drag associated with engine installation was the main objective of the DLR-NLR pre-competitive research co-operation project named Low Speed Propulsion Airframe Integration. One prior test programme with the turbine powered CRUF simulator was carried out in the DNW-LST. As important supplement to these tests, flow field measurements using Particle Image Velocimetry (PIV) were carried out with emphasis on the spatial development of the fan jet flow [2].

During these wind tunnel investigations, flow field measurements in the wake of the ALVAST [3] half model equipped with the high lift wing were carried out with and without CRUF (Counter Rotating Ultra-high bypass Fan) simulator in the DNW-NWB wind tunnel. The flow field measurements behind the wing and the simulator were made using a five-hole rake and a stereoscopic PIV-system. Balance measurements and surface pressure measurements on the wing were performed as well [1]. Results from the PIV measurements will be reported and discussed with respect to variations of the velocity fields and their topology.

2 Test Setup

For the present tests the closed test section running at atmospheric pressure was used with the under floor half-model balance. The longitudinal slots of the test section were kept closed except for small openings for introducing the laser light to illuminate a light sheet for PIV.

The DLR-ALVAST model is a generic wind tunnel model with a span of 3.428 meters, representing a modern transport aircraft similar to an Airbus A320 by scale 1:10. The modular concept of the model allows its individual assembly for different wind tunnel tests as full-span or half-span model either with cruise wing(s) or high lift wing(s). The high lift wings are equipped with inboard/outboard ailerons and with slats and flaps adjustable in a take-off and a landing position. Engine simulators can be installed on both full and half model configurations.

For the tests in DNW-NWB a half-model configuration was assembled with the high lift wing with slats and flaps adjusted in take-off position. The wing is equipped with pressure measurement instrumentation distributed along nine spanwise sections with a total of 617 pressure tappings.

The CRUF simulator unit has a two-stage counter rotating fan of 10" (254 mm) diameter with 8 blades per stage. This fan is driven by a four stage air turbine powered by pressurized air. Drive air was supplied through the balance by a pipe system equipped with three flexible air bridges, so that balance forces due to through-flowing high pressure air was minimized. A gear-box distributes the turbine power on the two counter rotating fan shafts without any RPM reduction. In the fan duct static pressure orifices and rakes with total pressure and temperature probes are installed.

2.1 PIV Setup and Data Processing

For seeding Di-2-Ethylhexyl-Sebacat (DEHS) was used as liquid. The generated particles are small droplets with a mean size of about 1 micro-meter. The seeding was introduced in the settling chamber. One particle generator was sufficient for seeding directly through its hose without a rake. The particles were illuminated by two Nd:YAG lasers of type Quantel Brilliant B. These light sources were pulsed with two independent oscillators with a repetition rate of 10 Hz. The pulse energy at $\lambda = 532$ nm was 2 x 320 mJ. As the PCO cameras utilised for thist test are limited to about 3 Hz, double images have been recorded each 0.3 sec.

To obtain three components of the velocity in the light sheet a stereo set-up was used. Two cameras with 1024 x 1280 pixels and a dynamic range of 12 bits were installed outside the tunnel and thus outside of the tunnel flow viewing through the outlets of the removed breezer flaps into the tunnel.

The measurement areas had dimension of about 20 cm x 20 cm, which results into $\approx$ 50 pixel per centimetre. The distance to the cameras was about 3 to 3.3 m. The opening angle between both cameras was $\approx 50^\circ$. Lenses of 135 mm focus length with maximum aperture of 2.8 were used. The time delay was chosen around 10 μsec. Remotely focusing of the lenses was achieved with a focusing system at the Scheimpflug adapters required for recording of stereo images.

In order to relate the position of both images and the coordinate system of the tunnel, calibration images of a planar table in the plane of the light sheet with a regular grid were taken. From these images the coefficients of the transformation have been evaluated, that maps from pixel values of both cameras into the plane of