Numerical Aerodynamics at DLR

Cord-Christian ROSSOW, Norbert KROLL, Dieter SCHWAMBORN
(*DLR, Deutsches Zentrum für Luft- und Raumfahrt e. V. in the Helmholtz-Association, Institute of Aerodynamics and Flow Technology, 38108 Braunschweig, 37073 Göttingen, Germany*)

Abstract: Some years ago the national CFD project MEGAFLOW was initiated in Germany to combine many of the CFD development activities from DLR, universities and aircraft industry. Its goal was the development and validation of a dependable and efficient numerical tool for the aerodynamic simulation of complete aircraft which met the requirements of industrial implementations. The MEGAFLOW software system includes the block-structured Navier-Stokes code FLOWer and the unstructured Navier-Stokes code TAU. Both codes have reached a high level of maturity and they are intensively used by DLR and the German aerospace industry in the design process of new aircraft. Recently, the follow-on project MEGADESIGN and MEGAOPT were set up which focus on the development and enhancement of efficient numerical methods for shape design and optimization. This article highlights recent improvements of the software and its capability to predict viscous flows for complex industrial aircraft applications.

Key words: numerical aerodynamics; Navier-Stokes solver; MEGAFLOW; DLR

数值空气动力学在 DLR(德国宇航研究院). Cord-Christian Rossow, Norbert Kroll, Dieter Schwamborn. 中国航空学报(英文版), 2006, 19(2): 134-150.

摘　要: 德国在若干年前启动的国家 CFD 项目 MEGAFLOW 集中了 DLR、大学以及航空工业界的许多 CFD 研究开发工作,目的是开发并验证能满足工业实践所要求的可靠并有效的数值工具,以对全机进行空气动力模拟。MEGAFLOW 软件系统包括多区结构网格中求解 N-S 方程的软件 FLOWer 和在非结构网格中的软件 TAU。两软件均已达到很高的成熟度并为 DLR 和德国航空工业广泛地应用于新飞机的设计过程。最近又启动了其后续项目 MEGADESIGN 和 MegaOpt,旨在开发和增强外形设计及优化的有效数值方法。着重介绍了软件最近的改进及其计算粘性流动绕复杂飞机外形的能力。

关键词: 数值空气动力学; N-S 解算器; MEGAFLOW; DLR

文章编号: 1000-9361(2006)02-0134-17　　**中图分类号:** V211　　**文献标识码:** A

Aerospace industry is increasingly relying on advanced numerical flow simulation tools in the early aircraft design phase. Today, computational fluid dynamics has matured to a point where it is widely accepted as an essential, complementary analysis tool to wind tunnel experiments and flight tests. Navier-Stokes methods have developed from specialized research techniques to practical engineering tools being used for a vast number of industrial problems on a routine basis[1]. Nevertheless, there is still a great need for improvement of numerical methods, because standards for simulation accuracy and efficiency are constantly rising in industrial applications. Moreover, it is crucial to reduce the response time for complex simulations, although the relevant geometries and underlying physical flow models are becoming increasingly complicated.

In order to meet the requirements of German aircraft industry, the national project MEGAFLOW was initiated some years ago under the leadership of DLR[2,3]. The main goal was to focus and direct development activities carried out in industry, DLR and universities towards industrial needs. The close collaboration between the partners led to the development and validation of a common aerodynamic simulation system providing both a structured and an unstructured prediction capability

Received date: 2005-10-23; Revision received date: 2006-01-20

for complex applications.

In the first phase of the project the main emphasis was put on the improvement and enhancement of the block-structured grid generator MEGACADS and the Navier-Stokes solver FLOWer. In a second phase the activities were focused on the development of the unstructured/hybrid Navier-Stokes solver TAU. In addition to the MEGAFLOW initiative, considerable development and validation activities were carried out in several DLR internal and European projects which contributed to the enhancement of the flow solvers.

Recently, based on the MEGAFLOW network the national project MEGADESIGN (2004-2007) as well as the complimentary DLR project MegaOpt were set up[4]. Their main objective is to enhance and establish numerical shape optimization tools within industrial aircraft design processes.

The present article describes the main features of the MEGAFLOW software and demonstrates its capability on the basis of several industrial relevant applications[5]. Finally, the perspective and future requirements of CFD for industrial applications are shortly outlined.

1 MEGAFLOW Software

1.1 Grid generation

For the generation of block-structured grids the interactive system MEGACADS has been developed. Specific features of the tool are the parametric construction of multi-block grids with arbitrary grid topology, generation of high-quality grids through advanced elliptic and parabolic grid generation techniques, construction of overlapping grids and batch functionality for efficient integration in an automatic optimization loop for aerodynamic shape design[6]. The limitation of MEGACADS is the non automatic definition of the block topology which may result in a time consuming grid generation activity.

In contrast to the block-structured approach, no major development activities have been devoted to the generation of unstructured meshes within the MEGAFLOW project. A strategic cooperation, however, has been established with the company CentaurSoft which provides the hybrid grid generation package Centaur[7].

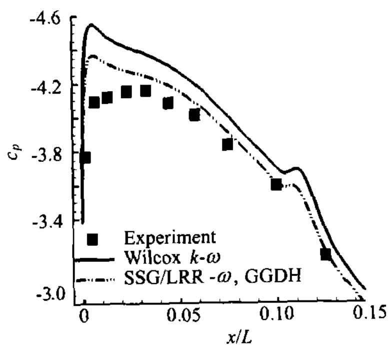
1.2 Flow solvers

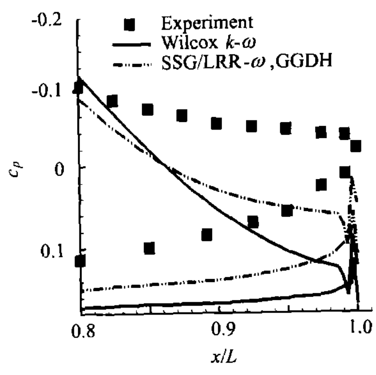
The main components of the MEGAFLOW software are the block-structured flow solver FLOWer and the unstructured hybrid flow solver TAU. Both codes solve the compressible three-dimensional Reynolds averaged Navier-Stokes equations for rigid bodies in arbitrary motion. The motion is taken into account by transformation of the governing equations. For the simulation of aero-elastic phenomena both codes have been extended to allow geometry and mesh deformation. In the following sections the specific features of the Navier-Stokes solvers are briefly described.

(1) Block-structured Navier-Stokes code FLOWer

The FLOWer-Code is based on a finite-volume formulation on block-structured meshes using either the cell vertex or the cell-centered approach. For the approximation of the convective fluxes a central discretization scheme combined with scalar or matrix artificial viscosity and several upwind discretization schemes are available[8]. Integration in time is performed using explicit multistage time-stepping schemes. For steady calculations convergence is accelerated by implicit residual smoothing, local time stepping and multigrid. Preconditioning is used for low speed flows. For time accurate calculations an implicit time integration according to the dual time stepping approach is employed. The software is highly optimized for vector and parallel computer[9].

A variety of turbulence models is implemented in FLOWer, ranging from simple algebraic eddy viscosity models over one-and two-equation models up to differential Reynolds stress models. The Wilcox k-ω model is the standard model in FLOWer which is used for all types of applications, while for transonic flow the linearized algebraic stress model LEA[10] and the nonlinear EARSM of Wallin[11] have shown to improve the prediction of shock locations. Furthermore, the SST model of Menter[12] is available for a better prediction of separating flows. All two-equation models can be combined with Kok's modification[13] for improved prediction of vortical flows. For supersonic flows different compressibility corrections are available. Recently, within the European project FLOMANI-A Reynolds stress models based on the Wilcox stress-ω model[14] and the so-called SSG/LRR-ω model, a combination of the Wilcox stress-ω and the Speziale-Sarkar-Gatski model[15], have been implemented into FLOWer[16]. Particularly the SSG/LRR-ω model has been applied to a wide variety of test cases, ranging from simple airfoils to complex aircraft configurations and from transonic to high-lift conditions. Generally improved predictions have been obtained, while the numerical behavior of the Reynolds stress models appeared to be as robust as that of two-equation models. Fig. 1 shows the predicted pressure and the skin friction distribution obtained with the Wilcox k-ω and with the SSG/LRR-ω model for the Aerospatiale A airfoil demonstrating the improvement by Reynolds stress modeling.





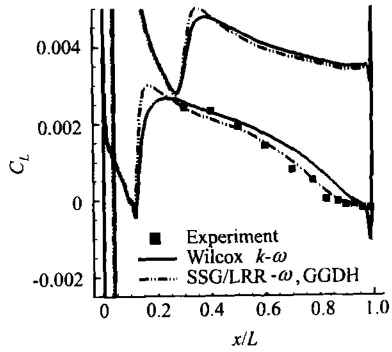


Fig. 1 Pressure distribution (near leading and trailing edge), skin friction distribution for Aerospatiale A airfoil ($Ma_\infty = 0.15$, $\alpha = 13.3°$, $Re = 2 \times 10^6$) FLOWer with Reynolds stress turbulence model

Besides the modeling accuracy for turbulent flows, the numerical robustness of the respective transport equation turbulence models for complex applications has been a major issue. In FLOWer numerical stability has been enhanced by an implicit treatment of the turbulence equations and different limiting mechanisms that can be activated by the user. The convergence behavior of the FLOWer-Code for a rather complex application is demonstrated in Fig. 2. Results of a viscous computation for a helicopter fuselage are shown[17]. The rotor is modeled through a uniform actuator disc. For this low Mach number case the preconditioning technique has been employed. The fully implicit integration of the turbulence equations also ensures efficient calculations on highly stretched cells as they appear in high Reynolds number flows[18].

FLOWer is able to perform transition prediction on airfoils and wings using a module consisting of a laminar boundary layer code and an e^N-database method based on linear stability theory[19]. Fig. 3 shows the predicted and measured force polars and transition locations of a subsonic laminar airfoil.

An important feature of FLOWer is the Chimera technique, which considerably enhances the flexibility of the block-structured approach[20,21]. This technique mainly developed within the German / French helicopter project

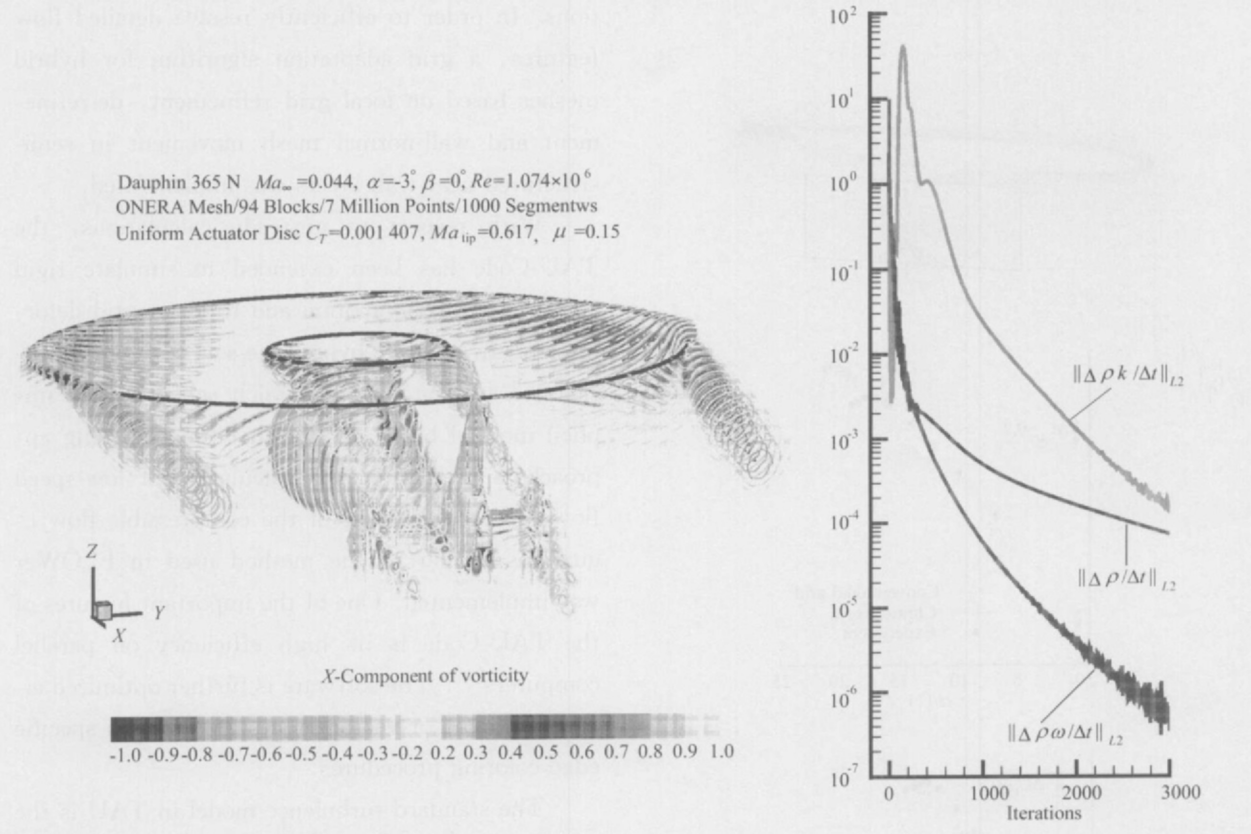
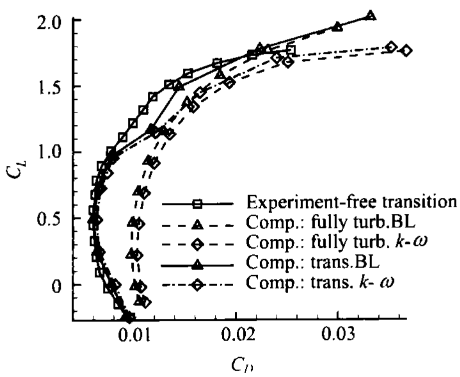


Fig. 2 Viscous calculation for Dauphin helicopter fuselage at $Ma_\infty = 0.044$, convergence behavior of mass and k-ω turbulence equations

CHANCE[22] enables the generation of a grid around a complex configuration by decomposing the geometry into less complex components. Separate component grids are generated which overlap each other and which are embedded in a Cartesian background grid. In combination with flexible meshes, the Chimera technique enables an efficient way to simulate bodies in relative motion. The potential of the Chimera technique is demonstrated in Fig. 4 for the viscous calculation of a 3D high-lift configuration.



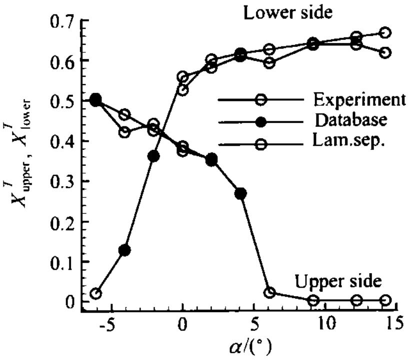


Fig. 3 Transition prediction with e^N-database method for laminar Sommers airfoil at $Ma_\infty = 0.1$ and $Re = 4\times10^6$, force polars calculated fully turbulent and with transition, computed and measured transition locations

(2) Hybrid Navier-Stokes code TAU

The Navier-Stokes solver TAU makes use of the advantages of unstructured grids[23,24] and employs a dual-grid approach. The mesh may consist of a combination of prismatic, pyramidal, tetrahedral and hexahedral cells and therefore combines

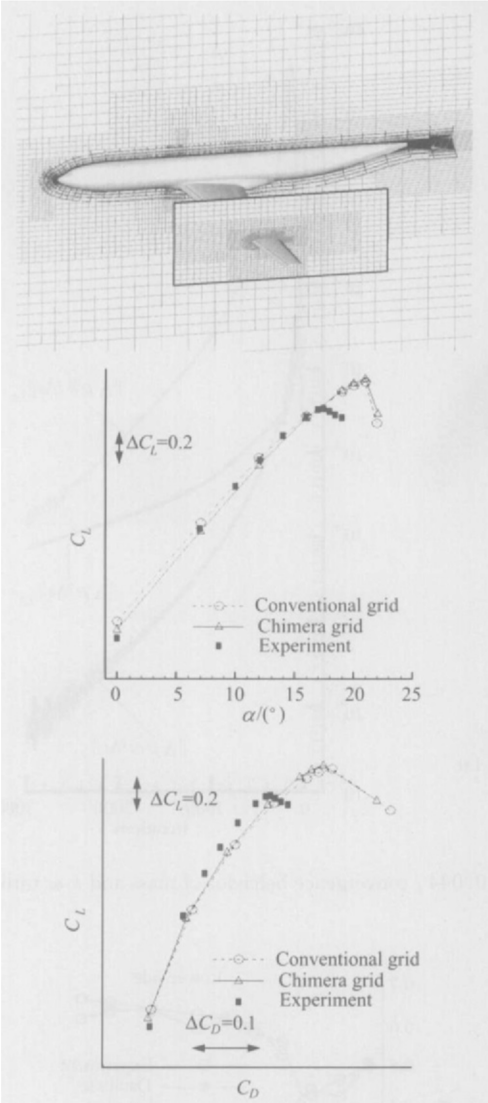


Fig. 4 Viscous computation about a 3D high-lift configuration using the Chimera technique of the block-structured FLOWer-Code, $Ma_\infty = 0.174$, $\alpha = 7°$

the advantages of regular grids for the accurate resolution of viscous shear layers in the vicinity of walls with the flexibility of grid generation techniques for unstructured meshes. The use of a dual mesh makes the solver independent of the type of cells that the initial grid is composed of. Various spatial discretization schemes were implemented. The basic hybrid TAU-Code uses an explicit Runge-Kutta multistage scheme in combination with an explicit residual smoothing. In order to accelerate convergence, a multigrid procedure was developed based on the agglomeration of the control volumes of the dual grid for coarse grid computations. In order to efficiently resolve detailed flow features, a grid adaptation algorithm for hybrid meshes based on local grid refinement, de-refinement and wall-normal mesh movement in semi-structured near-wall layers was implemented.

With respect to unsteady calculations, the TAU-Code has been extended to simulate rigid bodies in arbitrary motion and to allow grid deformation. In order to bypass the severe time-step restriction associated with explicit schemes, the implicit method based on the dual time stepping approach is used. For the calculation of low-speed flows, preconditioning of the compressible flow equations similar to the method used in FLOWer was implemented. One of the important features of the TAU-Code is its high efficiency on parallel computers[9]. The software is further optimized either for cache or vector processors through specific edge coloring procedures.

The standard turbulence model in TAU is the Spalart-Allmaras model with Edwardsmodification, yielding highly satisfactory results for a wide range of applications while being numerically robust. Besides this model, a number of different k-ω models with and without compressibility corrections are available. Also nonlinear Explicit Algebraic Reynolds Stress Models (EARSM) and the linearized LEA model[10] have been integrated. Several rotation corrections for vortex dominated flows are available for the different models. Finally, there are options to perform Detached Eddy Simulations (DES) based on the Spalart-Allmaras model[25] and so-called Extra-Large Eddy Simulations (XLES)[26].

The explicit character of the solution method severely restricts the CFL number which in turn often leads to slow convergence, especially in the case of large scale applications. In order to improve the performance and robustness of the TAU-Code, an approximately factored implicit scheme has been recently implemented[27]. The Lower-Upper Symmetric Gauss-Seidel (LU-SGS) scheme has been selected as a replacement for the Runge-Kutta scheme. In contrast to fully implicit schemes, this method has low memory requirements, low operation counts and can be parallelized with relative ease. Compared to the explicit Runge-Kutta method, the LU-SGS scheme is stable with almost no time step restrictions. An example of the performance improvement achieved is given in Fig. 5, where two convergence histories for viscous calculations of a delta wing are shown. In terms of iterations LU-SGS can be seen to converge approximately twice as fast as the Runge-Kutta scheme. Furthermore, one iteration of LU-SGS costs roughly 80% of one Runge-Kutta step.

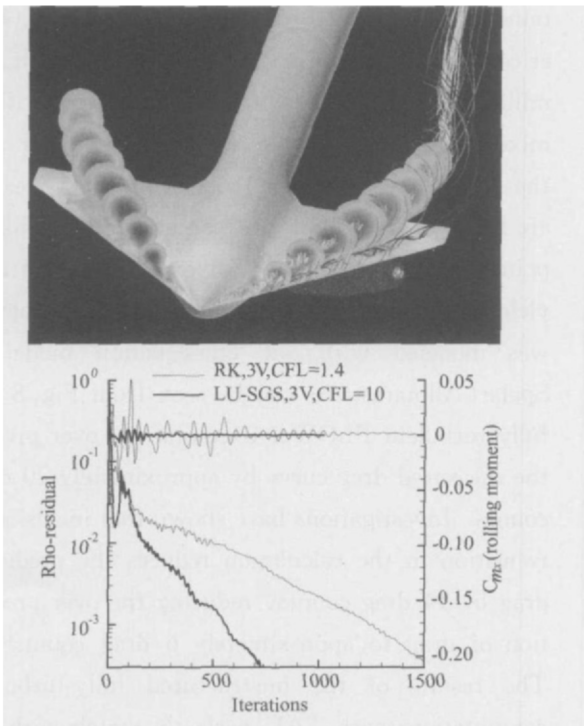


Fig. 5 Convergence of TAU for viscous flow around a delta wing $Ma = 0.5$, $\alpha = 9°$. Comparison of Runge-Kutta scheme (RK) and the implicit LU-SGS scheme

As the Chimera technique has been recognized as an important feature to efficiently simulate maneuvering aircraft, it has been also integrated into the TAU-Code[28]. In the context of hybrid meshes the overlapping grid technique allows an efficient handling of complex configurations with movable control surfaces. In Fig. 6, results of a viscous Chimera calculation for a delta wing with trailing edge flaps are shown[29].

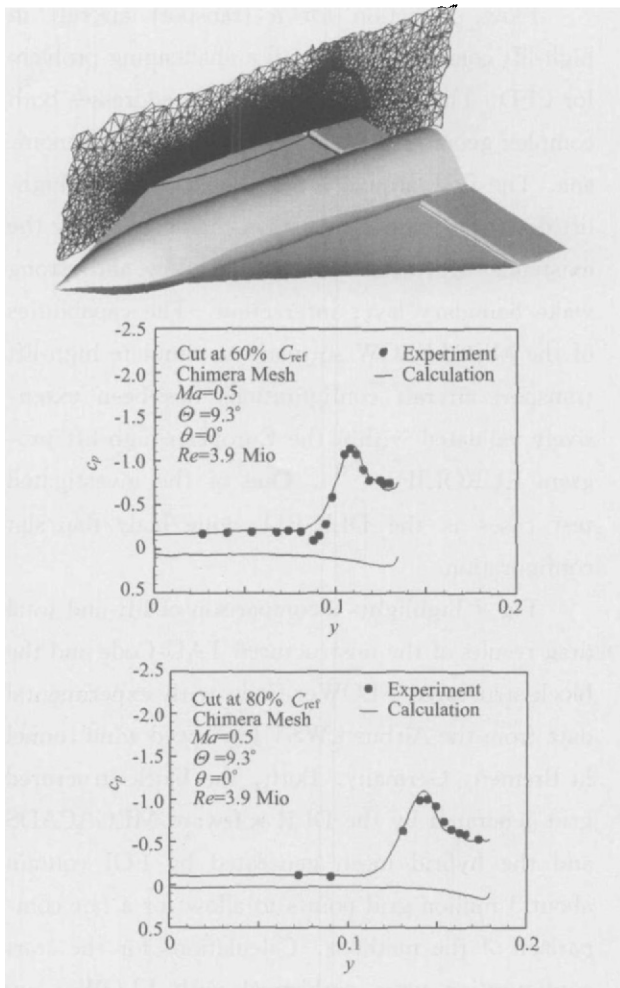


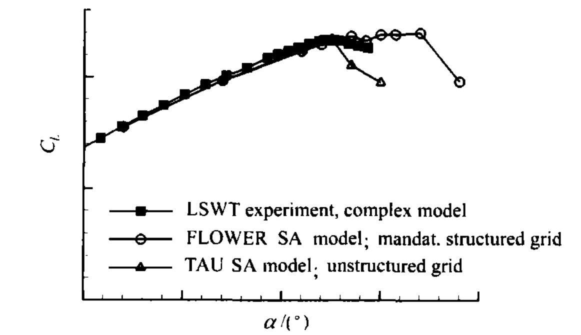
Fig. 6 Viscous computation of a delta wing with trailing edge flap using TAU with Chimera, surface pressure distributions for flap deflection angle $\theta = 0°$ at 60% and 80% chord

2 Software Validation

Software validation is a central and critical issue when providing reliable CFD tools for industrial applications. Among others, the verification and validation exercises should address consistency of the numerical methods, accuracy assessment for different critical application cases and sensitivity studies with respect to numerical and physical parameters. Best practice documentation is an essential part of the work. Over the last few years the MEGAFLOW software has been validated within various national and international projects for a wide range of configurations and flow conditions[5,30]. This section shows sample results for a subsonic and transonic validation test case.

Flow prediction for a transport aircraft in high-lift configuration is still a challenging problem for CFD. The numerical simulation addresses both complex geometries and complex physical phenomena. The flow around a wing with deployed high-lift devices at high incidence is characterized by the existence of areas with separated flow and strong wake/boundary layer interaction. The capabilities of the MEGAFLOW software to simulate high-lift transport aircraft configurations has been extensively validated within the European high-lift program EUROLIFT-I[31]. One of the investigated test cases is the DLR-F11 wing/body/flap/slat configuration.

Fig. 7 highlights a comparison of lift and total drag results of the unstructured TAU-Code and the block-structured FLOWer-Code with experimental data from the Airbus LWST low speed wind tunnel in Bremen, Germany. Both, the block-structured grid generated by the DLR software MEGACADS and the hybrid mesh generated by FOI contain about 3 million grid points to allow for a fair comparison of the methods. Calculations for the start configuration were performed with FLOWer and TAU using the Spalart-Allmaras turbulence model with Edwards modification. In both cases preconditioning was used.



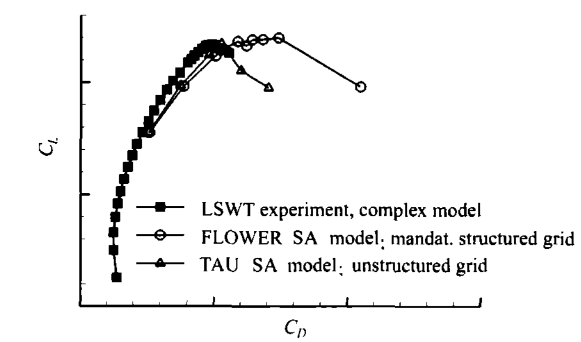


Fig. 7 Viscous computations for DLR-F11 high-lift configuration at $Ma_\infty = 0.18$, $Re = 1.4 \times 10^6$

In the framework of the 1st AIAA CFD Drag Prediction Workshop[32], the accuracy of the MEGAFLOW software was assessed to predict aerodynamic forces and moments for the DLR-F4 wing-body configuration[33]. In Fig. 8 lift coefficient as function of drag and angle of attack for Case 2 calculated with FLOWer and TAU are presented. These results were obtained using grids generated in-house at DLR. On request all calculations were performed fully turbulent. The FLOWer computations were carried out on a grid with 3.5 million points using central discretization with a mixed scalar and matrix dissipation operator and the k/ω-LEA turbulence model. The TAU results are based on an initial grid containing 1.7 million points which was adapted for each angle of attack yielding grids with 2.4 million points. Turbulence was modeled with the one-equation model of Spalart-Allmaras. As can be seen from Fig. 8 the fully turbulent FLOWer computations over predict the measured drag curve by approximately 20 drag counts. Investigations have shown that inclusion of transition in the calculation reduces the predicted drag by 14 drag counts, reducing the over prediction of drag to approximately 6 drag counts[33]. The results of the unstructured fully-turbulent computations with TAU perfectly match with the experimental data. However, as for the structured computations, hybrid calculations with transition setting will reduce the predicted level of drag, in this case by approximately 10 drag-counts. Fig. 8 also shows the comparison of predicted and measured lift coefficient as a function of angle of attack. The values calculated by FLOWer agree very well with the experiment, whereas the results obtained with TAU over predict the lift almost in the whole range of angle of attack. For the pitching moment the results obtained with FLOWer agree very well with experimental data. This is due to the fact that the surface pressure distribution predicted with the FLOWer-Code is in good agreement with the experiment. In case of the hybrid TAU-Code there are some discrepancies between the predicted and measured surface pressures resulting in a significant over prediction of the pitching moment. Further investigations have shown that the improved results obtained with the FLOWer-Code are mainly attributed to a lower level of numerical dissipation (grid resolution and matrix dissipation) combined with the advanced two-equation k/ω-LEA turbulence model[33].

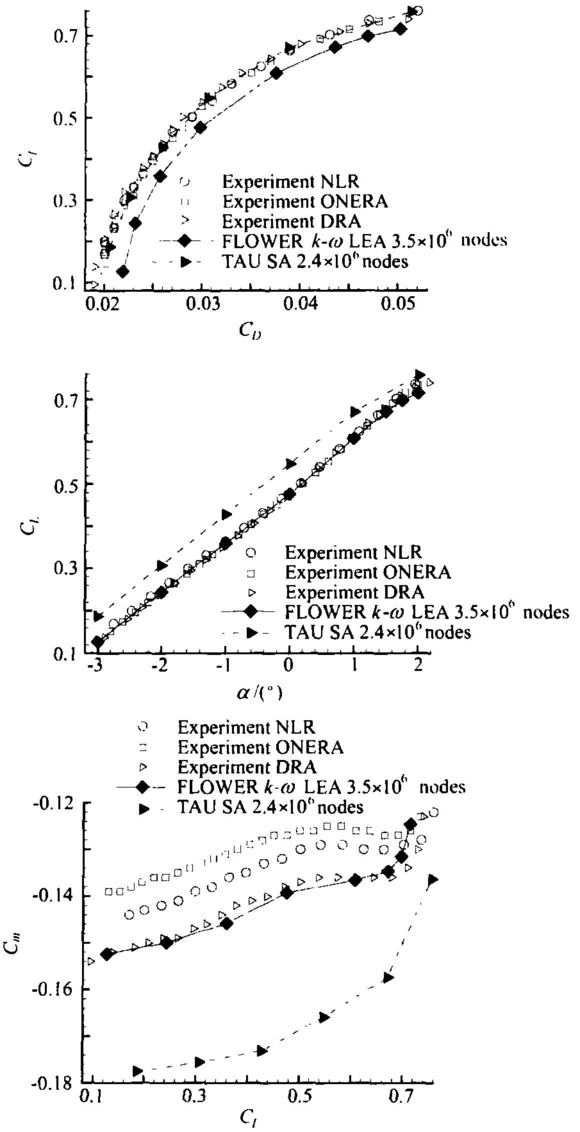


Fig.8 Viscous calculations for DLR-F4 wing/body configuration, AIAA DPW-I, $Ma_\infty = 0.75$, $Re = 3 \times 10^6$

Within the 2nd AIAA Drag Prediction Workshop the hybrid TAU-Code was further assessed with respect to performance calculations for a wing/body/pylon/nacelle configuration[34,35]. For this exercise the Spalart-Allmaras one-equation turbulence model was used. The drag polar is predicted in good agreement with the experimental data while the lift is constantly over predicted (see Fig. 9). A detailed analysis of the flow features reveals that in principle all areas of flow separations on the investigated DLR-F6 configuration are identified, however, compared with experiments the sizes of those areas are slightly under predicted (wing upper side) or over predicted (wing lower side). Fig. 10 compares measured and predicted flow features near the pylon inboard side at the wing lower side. This difference results in systematic deviations of the pressure distributions and pitching moments.

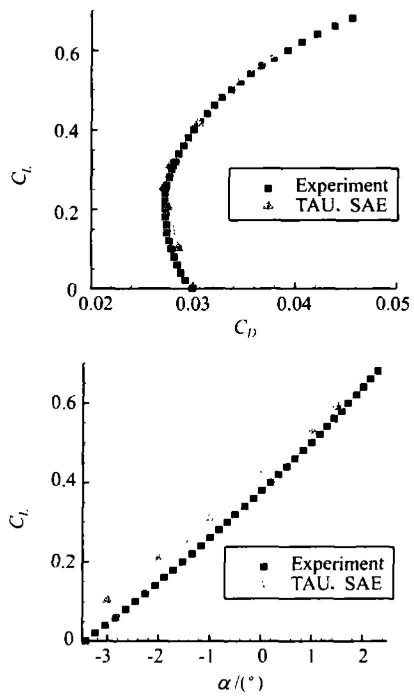
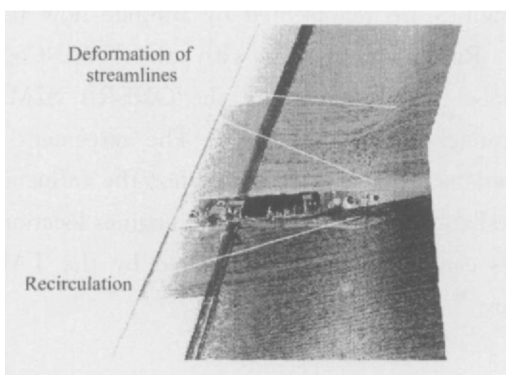


Fig.9 TAU results for DLR-F6 wing/body/pylon/ nacelle configuration, AIAA DPW-II, $Ma_\infty = 0.75$, $Re = 3 \times 10^6$



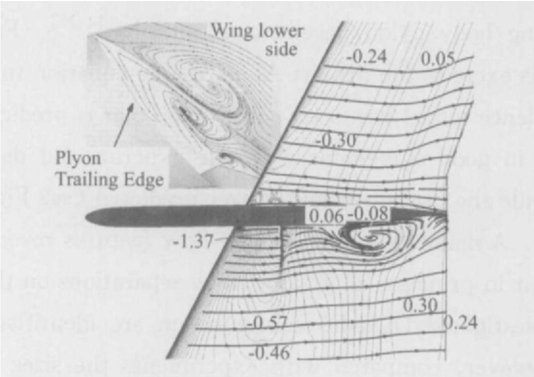


Fig. 10 Oil flow patterns (experiments) and streamlines (TAU), DLR-F6 wing/body/pylon/nacelle configuration, wing lower and pylon inboard side, $Ma_{\infty}=0.75$, $C_L=0.5$

3 Industrial Applications

Some typical large scale applications listed below demonstrate the capability of the MEGAFLOW software to support aircraft and helicopter design.

3.1 Civil transport aircraft at cruise flight conditions

One key issue during the design of a civil aircraft is the efficient engine-airframe integration. Modern very high bypass ratio engines and the corresponding close coupling of engine and airframe may lead to substantial loss in lift and increased installation drag. At DLR, numerical and experimental studies have been devoted to estimate installation drag with respect to variations of engine concepts and the installation positions[36,37]. For numerical investigations in this field both the block-structured FLOWer-Code and the hybrid TAU-Code have been used. In Fig. 11 the lift as a function of the installation drag is plotted for three different positions of the CFM56 long duct nacelle. The engines are represented by through-flow nacelles. Results predicted with the TAU-Code (symbols) and measured in the ONERA S2MA wind tunnel (lines) are shown. The agreement is very satisfactory demonstrating that the influence on installation drag due to varying engines locations or sizes can be accurately predicted by the TAU software[38].

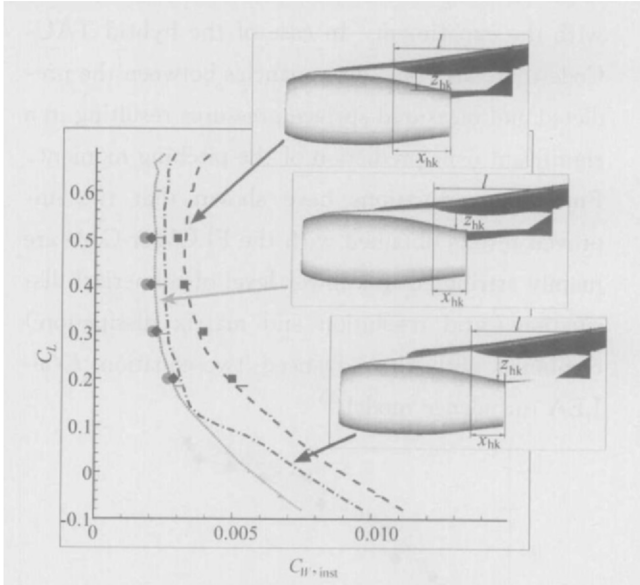


Fig. 11 Engine-airframe interference drag (TAU), lift as a function of installation drag for three different position of CFM56 engine, $Ma_{\infty}=0.75$, $Re=3\times10^6$, symbols: calculation, lines: experiment

3.2 Civil transport aircraft at high-lift flight conditions

Based on thorough development and validation efforts of the hybrid unstructured approach employing both the Centaur grid generation software and the Navier-Stokes-Code TAU, complex high-lift flows become more and more accessible. As an example the flow around the DLR ALVAST model in high lift configuration equipped with two different engine concepts, the Very High Bypass Ratio (VHBR) and the Ultra High Bypass Ratio (UHBR) engine has been computed[39]. The numerical simulations are focused on complex flow phenomena arising from the engine installation at high-lift conditions. Special attention was paid to a possible reduction of the maximum lift angle resulting from dominant three-dimensional effects due to engine installation. Fig. 12 displays the surface pressure coefficient of the ALVAST high-lift configuration with installed VHBR and UHBR engine at an angle of attack of $\alpha=12°$ in take-off conditions. The computations were performed on a hybrid grid generated with Centaur. In Fig. 13 the vortex shedding from the inboard side of the nacelle is shown. The vortex originates from the rolling-up of the shear layer and crosses the slat and the wing upper side. Using the computational data as input this vortex system could be identified with PIV visualization in a recent wind tunnel campaign. Fig. 13 also shows the impact of the two different engine concepts on the span wise lift distribution. For the VHBR concept the lift loss on the wing due to engine mounting is roughly compensated by the lift generated by the nacelle itself. For the UHBR concept the wing lift loss is slightly stronger than for the VHBR. Nevertheless, it is overcompensated by the higher lift carried by the large nacelle.

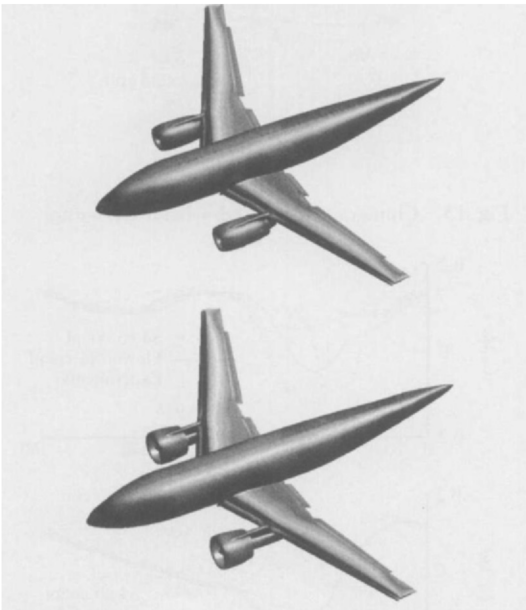


Fig. 12 Viscous simulation of the ALVAST high-lift configuration with VHBR (top) and UHBR (bottom) engine using TAU, surface pressure distribution, $Ma_\infty = 0.22$, $\alpha = 12°$, $Re = 2 \times 10^6$

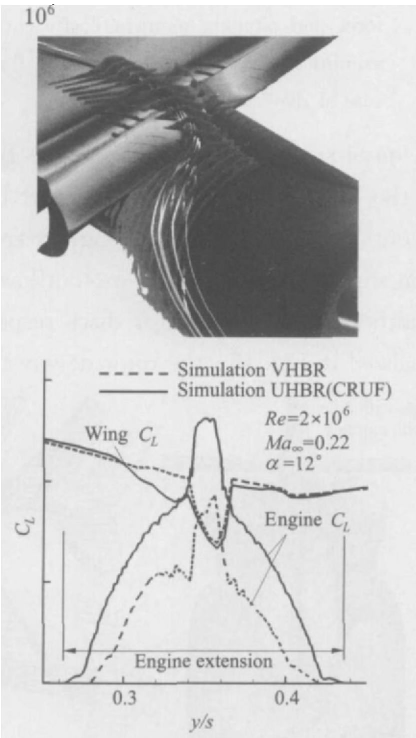


Fig. 13 Engine interference for ALVAST high-lift configuration with VHBR and UHBR engine $Ma_\infty = 0.22$, $\alpha = 12°$, $Re = 2 \times 10^6$, top: nacelle vortex, bottom: lift distribution of wing and nacelle

One key aspect of the development of a new transport aircraft is the design of a sophisticated and optimal high-lift system for take-off and landing conditions. A possibility to increase maximum lift is the usage of small delta wing like plates on the engine nacelles, the so-called nacelle strakes. These strakes generate vortices which run above the wing for high angles of attack. These vortices influence the wing and slat pressure distributions and shift the flow separations to higher angles of attack. At cruise flight conditions the strakes should not produce any significant additional drag. In order to quantitatively predict the lift increment due to the strakes, care must be taken generating and adapting the grid with and without strakes. The idea has been to use the final adapted grid of the configuration with nacelle strakes and to fill the strakes with tetrahedral elements so that a nearly identical grid for the configuration with and without strakes can be build. The initial grid generation has been performed with Centaur. The TAU grid adaptation has been used. The three times adapted grid contains approximately 16.7 million points. The filling of the strake volume has been performed using customized tools based on MEGACADS[6] and the NETGEN[40] software. The solutions have been calculated using the TAU-Code for various flow conditions. Fig. 14 demonstrates the resolution of the strake vortex and an iso-vorticity plane for $\alpha = 10°$. It has been shown that for this configuration a lift increase of $\Delta C_L \approx 0.1$ can be found both from the numerical calculations and the experiments although the absolute maximum lift values differ[41].

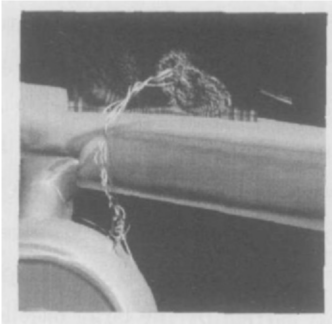


Fig. 14 High-lift configuration with nacelle strake, calculated streamlines and iso-vorticity cut planes. $Ma_\infty = 0.18$, $\alpha = 10°$, $Re = 3 \times 10^6$

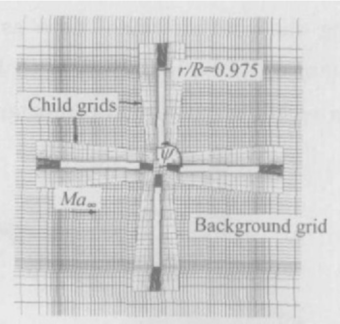


Fig. 15 Chimera grid around 4-bladed 7A-rotor

3.3 Helicopter

At DLR large effort is devoted to the enhancement of the MEGAFLOW software for helicopter applications. The development and validation activities are carried out in the German/French project CHANCE[22]. They include performance prediction of the isolated rotor in hover and forward flight as well as the quasi-steady and time-accurate simulation of a helicopter including engines and main and tail rotor.

The aerodynamic assessment of helicopter main rotors requires a computational procedure with fluid-structure coupling including trim. The results which are presented here were obtained with a weak coupling between the RANS solver FLOWer and the comprehensive rotor simulation code S4 in which the blade structure is modeled as a beam[42]. The test case is the four-bladed 7A-rotor with rectangular blades in high-speed forward flight ($Ma_\omega R = 0.64$, $Ma_\infty = 0.256$, advance ratio $\mu = 0.4$). Fig. 15 presents the grid system used while Fig. 16 compares the measured with the predicted data. The overall agreement of the coupled solution (FLOWer/S4 coupling) with the experimental data is acceptable although the negative peak in normal force around 120° azimuth is not well computed. This phenomenon is subject of ongoing research. The results of the simplified blade element aerodynamic module of S4 are presented by dashed lines in Fig. 16. It is obvious that this simplified aerodynamic model is not able to capture the time dependent blade load history.

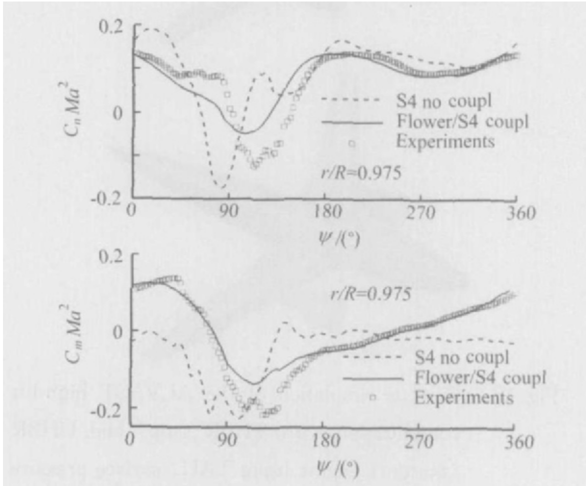


Fig. 16 Comparison of predicted and measured normal force and pitching moment coefficients versus azimuth for a high-speed forward flight test case of the 7A rotor

A quasi-steady computation of the flow-field around the Eurocopter EC-145 helicopter has been carried out[17,43]. The effect of engines and rotors has been simulated by means of in-/outflow boundary conditions and by actuator discs respectively. As visualized in Fig. 17, the rotor downwash re-

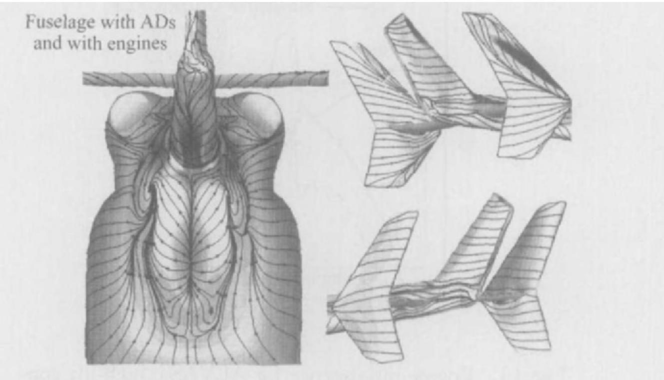


Fig. 17 c_p-distribution and friction lines on the EC145 fuselage, visualization of separation areas on the boot and vertical stabilizers sults in an asymmetrical flow pattern on the fuselage surface. The figure shows separation lines and singular points on the boot and tail boom. Moreover, the right vertical stabilizer experiences a much higher loading as the left one.

4 Multidisciplinary Simulations

The aerodynamic performance of large transport aircraft operating at transonic conditions is highly dependent on the deformation of their wings under aerodynamic loads. Hence accurate performance predictions require fluid/structure coupling in order to determine the aerodynamics of the configuration in aero-elastic equilibrium. Consequently, at DLR major effort is currently devoted to couple the flow solvers FLOWer and TAU with numerical methods simulating the structure. The activities include the development of efficient and robust grid deformation tools, accurate interpolation tools for transferring data between the fluid grid and the structure grid as well as the implementation of suitable interfaces between the flow solvers and the structural solvers. Concerning structure, both high-fidelity models (ANSYS, NASTRAN software) and simplified models (beam model) are considered.

The importance of fluid/structure coupling is demonstrated in Fig. 18. Within the European project HiReTT Navier-Stokes calculations were performed for a wing-body configuration of a modern high speed transport type aircraft. FLOWer was used with the k/ω turbulence model. Two types of calculations were carried out. On the one hand the aerodynamic behavior of the jig-shape was predicted. On the other hand the aero-elastic equilibrium was determined by a fluid/structure coupling. For this calculation the coupling procedure of the University of Aachen was used[44]. It is based on FLOWer for the fluid and a beam model for the structure. From Fig. 18 it is obvious that good agreement with experimental data obtained in the European Transonic Wind tunnel can only be achieved with the fluid/structure coupling.

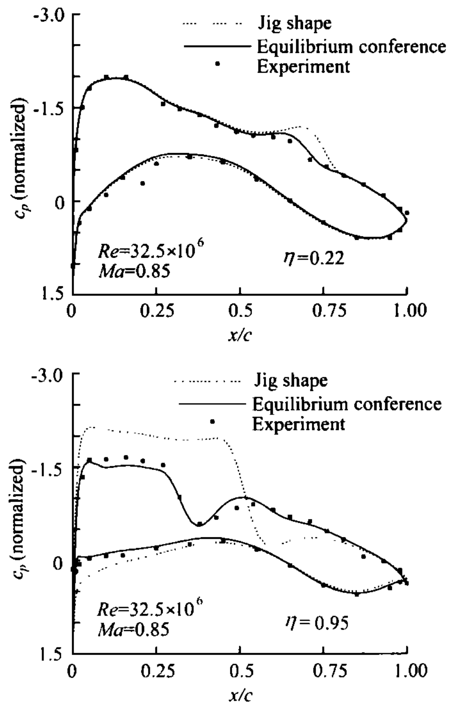


Fig. 18 c_p-distribution for different span wise sections for a wing/body configuration, numerical results obtained for pre-deformed geometry (dashed line) and with fluid/structure coupling (full line)

5 Numerical Optimization

For aerodynamic shape optimization, FLOWer and TAU offer an inverse design mode which is based on the inverse formulation of the small perturbation method according to Takanashi[45]. The method has been extended to transonic flows and is capable of designing airfoils, wings and nacelles in inviscid and viscous flows[46]. In the context of regional aircraft development various wing designs for transonic flow were performed at DLR with the inverse mode of FLOWer. As design target suitable surface pressure distributions were specified subject to geometrical constraints and a given lift coefficient. Fig. 19 shows the comparison of drag rise between an early baseline wing and an improved wing as a function of Mach number. The reduction of drag in the higher Mach number range is clearly visible. The constraint with respect to the lift coefficient was satisfied. The inverse design methodology coupled to the hybrid TAU-Code was also applied to the design of wing-mounted engine nacelles[47].

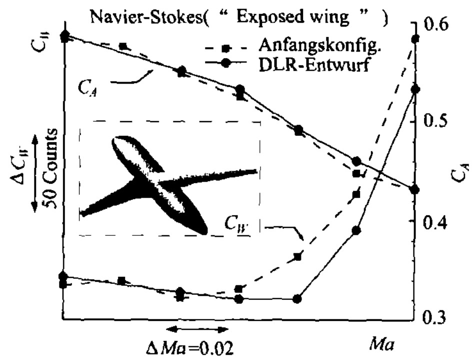


Fig. 19 Inverse wing design using FLOWer, drag rise and lift as function of Mach number for baseline configuration and optimized configuration

The inverse design method is very efficient; however it is restricted to a prescription of a target pressure distribution. A more general approach is the numerical optimization in which the shape, described by a set of design parameters, is determined by minimizing a suitable cost function subject to some constraints. At DLR high-lift system optimization is of major interest. Hence, the MEGAFLOW software has been coupled to various optimization strategies. As a demonstration results of a drag optimization for a 3-element airfoil in take-off configuration are presented in Fig. 20[48]. A limit in pitching moment has been prescribed as secondary constraint. In total 12 design variables

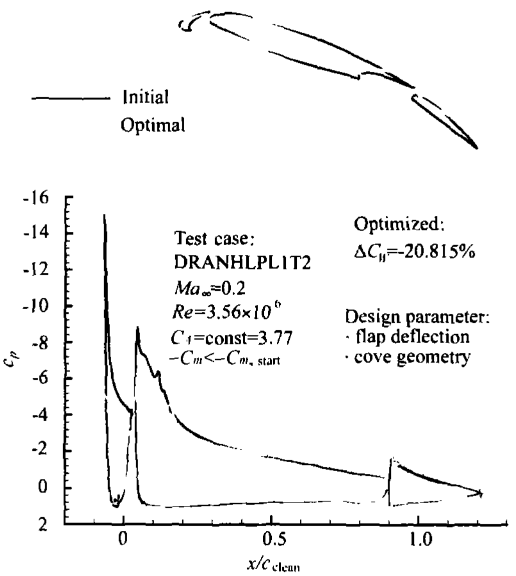


Fig. 20 Setting optimization of a 3-element airfoil using FLOWer

are taken into account. These are slat and flap gap, overlap and deflection. In addition, the slat and flap cut-out contours are parameterized by three variables each. The optimization method is based on a deterministic SUBPLEX strategy. The Navier-Stokes FLOWer-Code is used to predict the flow field. The optimization affects the element chord, setting and deflection angle as well as the angle of attack. The optimization results in a decrease in total drag of 21%, while the maximum lift is slightly improved by 2%.

Because detailed aerodynamic shape optimizations still suffer from high computational costs, efficient optimization strategies are required. Regarding the deterministic methods, the adjoint approach is seen as a promising alternative to the classical finite difference approach, since the computational cost does not depend on the number of design parameters[49]. Accordingly, within the MEGAFLOW project an adjoint solver following the continuous adjoint formulation has been developed and widely validated for the block-structured flow solver FLOWer[50]. The adjoint solver can deal with the boundary conditions for drag, lift and pitching-moment sensitivities. The adjoint option of FLOWer has been validated for several 2D as well as 3D optimization problems controlled by the adjoint Euler equations. Within the ongoing MEGADESIGN project the robustness and efficiency of the adjoint solver will be further improved, especially for the Navier-Stokes equations. The adjoint solver implemented in FLOWer is currently transferred to the unstructured solver TAU.

To demonstrate the capability of the adjoint approach to handle many design parameters with low cost, the optimization of a supersonic transport wing/body configuration has been carried out[51]. The baseline geometry is based on the EUROSUP geometry (see Fig. 21), which is a supersonic commercial aircraft of 252 seats capacity, designed for a range of 5 500 nautical miles with supersonic cruise at Mach number $Ma_\infty = 2.0$[52]. The optimization goal is to minimize the drag at a fixed lift coefficient of $C_L = 0.12$. The fuselage incidence is

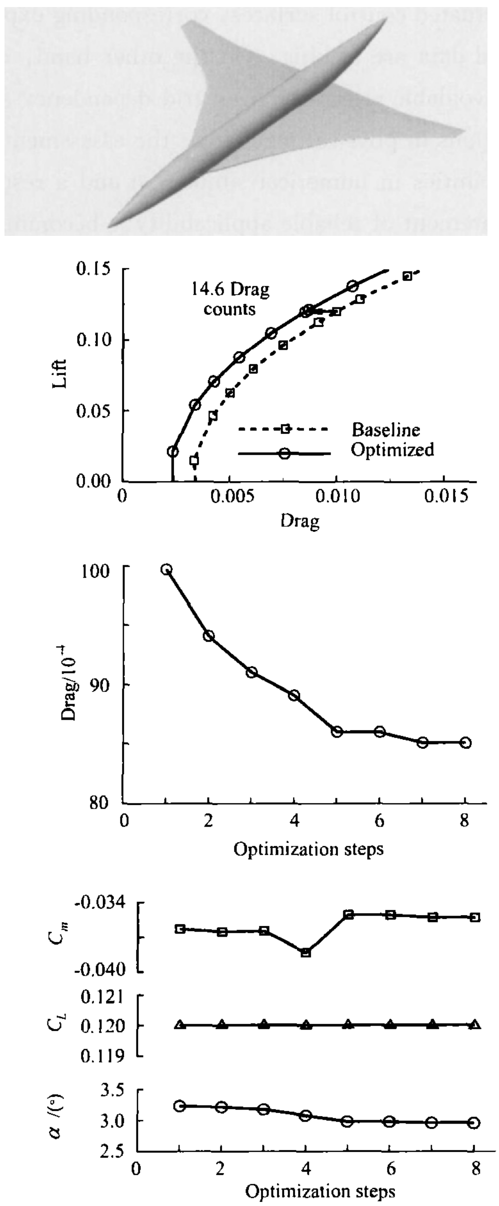


Fig.21 Shape optimization of supersonic transport aircraft at $Ma_\infty = 2.0$ (drag minimization at constant lift)

allowed to change in order to maintain the lift coefficient but it should not be greater than 4 degrees to the onset flow. In order to explore the full potential of the adjoint technique, no specific restrictions are set to define the parameterization. 74 design variables were used to change the twist, the thickness and the camber line at specific wing sections and 10 more design variables allowed changing the radial distribution of the fuselage. A minimum allowable value of the fuselage radius and a minimum wing thickness law were imposed in order to prevent unrealistic aircraft. After geometrical modifications, the intersection of wing and fuselage is recalculated automatically by the DLR in-house grid generator MegaCads for each new configuration. At $Ma_\infty = 2.0$, the main aerodynamic effects are well predicted using the Euler equations. Therefore, the aerodynamic states are computed by FLOWer running in Euler mode. In the present optimization problem, the unique aerodynamic constraint is the lift, which is handled directly by FLOWer and the geometrical constraints are automatically fulfilled during the parameterization. Fig.21 shows the evolution of the drag coefficient during the optimization, where an optimization step includes the evaluation of the gradient and the line search. About 8 optimization steps were necessary to achieve the optimum, which represents 54 aerodynamic computations and 8 adjoint flow evaluations. This approach is more than 11 times faster than using brute force optimization based on finite differences. The optimum configuration has 14.6 less drag counts than the baseline geometry.

6 Conclusion and Perspective

The main objective of the MEGAFLOW initiative was the development of a dependable, effective and quality controlled software package for the aerodynamic simulation of complete aircraft. Due to its high level of maturity, the MEGAFLOW software system is being used extensively throughout Germany for solving complex aerodynamic problems-especially in industrial development processes. However, since industry is still demanding more accurate and faster simulation tools, further development is required despite the high level of numerical flow simulation established today. Four major fields of further research activities may be identified.

The first field is the enhancement of numerical methods by new algorithms and solution strategies. Here, accuracy, robustness, and efficiency have to be addressed, while recognizing that these are contradicting requirements. In the design process of the aerospace industry with its severe time constraints, the difficult set-up of highly accurate computations can not be tolerated. However, to establish numerical simulation during design, where decisions involving extreme economical risks have to be made, accuracy and reliability are crucial, which is why expensive wind tunnel testing is still indispensable. Furthermore, the efficiency of numerical methods has to be substantially improved. Relying solely on the progress of computational hardware is not an option, since over the last two decades the size of the problems to be simulated increased in parallel to or even faster than advancements in computer technology.

Second, the physical modeling of fluid flow needs further to be addressed. Despite long-time efforts, the current status of modeling of turbulence and transition is still inadequate for the highly complex flows to be simulated in aircraft design. Due to the immense computational effort required, the Direct Numerical Simulation (DNS) or even Large Eddy Simulation (LES) of fluid flow will not be a practical alternative even for the next four or five decades. Therefore, reliable modeling of turbulence and transition will become decisive to bring numerical simulation as a routinely used tool into the aeronautical design process.

Third, the architecture of the simulation software is becoming more and more a strategic issue. On the one hand the software architecture must thoroughly exploit computational capabilities like parallelism, which requires a certain degree of dedication to a certain computational environment; on the other hand the software should be portable to different hardware arrangements. Furthermore, the software must be flexible with respect to coupling with other disciplines and integration into optimization strategies to allow the definition of an interdisciplinary simulation and optimization environment.

The last field to be addressed is validation. This requires on the one hand the thorough definition of suitable experiments by using most advanced measuring techniques. Especially for the envisaged simulation of unsteady flows with moving bodies and actuated control surfaces, corresponding experimental data are lacking. On the other hand, due to unavoidable effects such as grid dependency and limitations in physical modeling, the assessment of uncertainties in numerical simulation and a resulting statement of reliable applicability is becoming a major matter of future concern.

Acknowledgements

The authors would like to thank their colleagues of the DLR Institute of Aerodynamics and Flow Technology for providing material presented in this article. Thanks also to C. Braun from the University of Aachen, who provided the numerical results shown in Fig. 18. Furthermore, the partial funding of the MEGAFLOW and MEGADESIGN project through the German Government in the framework of the aeronautical research program is gratefully acknowledged.

References

[1] Vos J B, Rizzi A W, Darracq D, *et al*. Navier-Stokes solvers in European aircraft industry[J]. Progress in Aerospace Sciences, 2002, 38: 601 – 697.

[2] Kroll N, Rossow C C, Becker K, *et al*. The MEGAFLOW project [J]. Aerospace Science Technology, 2000, 4: 223 – 237.

[3] Kroll N, Rossow C C, Schwamborn D, *et al*. MEGAFLOW-a numerical flow simulation tool for transport aircraft design [R]. ICAS Paper 2002-1105, 2002.

[4] Kroll N, Gauger N R, Brezillon J, *et al*. Ongoing activities in shape optimization within the German project MEGADESIGN[A]. ECCOMAS 2004[C]. Jyväskylä Finland, 2004.

[5] Kroll N, Fassbender J K. MEGAFLOW-Numerical flow simulation for aircraft design[A]. Notes on Numerical Fluid Mechanics and Multidisciplinary Design[C]. Springer, 2005. 89.

[6] Brodersen O, Ronzheimer A, Ziegler R, *et al*. Aerodynamic applications using MegaCads[A]. In Cross M ed. 6th International Conference on Numerical Grid Generation on Computational Field Simulations[C]. ISGG. 1998. 793 – 802

[7] CentaurSoft, URL: http//www.centaursoft.com [cited Sept 2005].

[8] Kroll N, Radespiel R, Rossow C C. Accurate and efficient flow solvers for 3D-applications on structured meshes[R]. AGARD R-807, 1995, 41 – 459.

[9] Aumann P, Barnewitz H, Schwarten H, *et al*. MEGAFLOW parallel complete aircraft CFD[J]. Parallel Computing, 2001, 27: 415 – 440

[10] Rung T, Lübcke H, Franke M, *et al*. Assessment of explicit algebraic stress models in transonic flows [A]. Proceedings of the 4th Symposium on Engineering Turbulence Modeling and

Measurements[C]. France, 1999. 659 – 668.

[11] Wallin S, Johansson A V. An explicit algebraic reynolds stress model for incompressible and compressible turbulent flows[J]. Journal Fluid Mechanics, 2000,403: 89 – 132.

[12] Menter F R. Two-equation eddy-viscosity turbulence models for engineering applications[J]. AIAA Journal, 1994, 32: 1598 – 1605.

[13] Kok J C, Brandsma F J. Turbulence model based vortical flow computations for a sharp edged delta wing in transonic flow using the full Navier-Stokes equations[R]. NLR-CR-2000-342, 2000.

[14] Wilcox D C. Turbulence modeling for CFD[M]. La Cañada: CA DCW Industries, 1998.

[15] Speziale C G, Sarkar S, Gatski T B. Modeling the pressure-strain correlation of turbulence an invariant dynamical systems approach[J]. Journal of Fluid Mechanics, 1991, 227:245 – 272.

[16] Eisfeld B. Numerical simulation of aerodynamic problems with a Reynolds stress turbulence model[A]. STAB 2004, Notes on Numerical Fluid Mechanics and Multidisciplinary Design (NNFM)[C]. Springer Verlag, 2005.

[17] LeChuiton F. Actuator disc modeling for helicopter rotors [J]. Aerospace Science Technology, 2004,8: 285 – 297.

[18] Fassbender J K. Improved robustness for numerical simulation of turbulent flows around civil transport aircraft at flight [R]. Reynolds Numbers DLR-FB 2003-09, 2003.

[19] Krumbein A. Coupling of the DLR Navier-Stokes solver FLOWer with an e^N-database method for laminar-turbulent transition prediction on airfoils[J]. Notes on Numerical Fluid Mechanics, 2002, 77: 92 – 99.

[20] Heinrich R, Kalitzin N. Numerical simulation of three-dimensional flows using the chimera technique[J]. Notes on Numerical Fluid Mechanics, 1999, 72:15 – 23

[21] Schwarz T. Development of a wall treatment for Navier-Stokes computations using the overset grid technique[A]. 26th European Rotorcraft Forum[C]. 2000. Paper 45.

[22] Sidès J, Pahlke K, Costes M. Numerical simulation of flows around helicopters at DLR and ONERA[J]. Aerospace Science and Technology, 2001,5:35 – 53.

[23] Galle M. Ein verfahren zur numerischen simulation kompressibler, reibungsbehafteter Strömungen auf hybriden Netzen [R]. DLR-FB 99-04, 1999.

[24] Gerhold T, Friedrich O, Evans J, *et al*. Calculation of complex three-dimensional configurations employing the DLR-TAU Code[R]. AIAA Paper 97-0167, 1997.

[25] Strelets M. Detached eddy simulation of massively separated flows[R]. AIAA Paper 2001-0879, 2001.

[26] Kok J C, Dol H S, Oskam B, *et al*. Extra-large eddy simulation of massively separated flows[R]. AIAA Paper 2004-0264, 2004.

[27] Dwight R. A comparison of implicit algorithms for the Navier-Stokes equations on unstructured grids[A]. Proceedings of the ICCFD Conference[C]. Toronto Canada, 2004.

[28] Madrane A, Raichle A, Stürmer A. Parallel implementation of a dynamic overset unstructured grid approach[A]. ECCOMAS 2004[C]. Jyväskylä, Finland, 2004.

[29] Schütte A, Einarsson G, Madrane A, *et al*. Numerical simulation of maneuvering aircraft by CFD and flight mechanic coupling[A]. RTO Symposium[C]. Paris, 2002.

[30] Rudnik R, Melber S, Ronzheimer A, *et al*. Three-dimensional Navier-Stokes simulations for transport aircraft high lift configurations[J]. AIAA Journal of Aircraft, 2001,38(5): 895 – 903.

[31] Rudnik R. Towards CFD validation for 3D high lift flows-EUROLIFT[A]. ECCOMAS 2001[C]. Swansea, United Kingdom, 2001.

[32] Levy D, Zickuhr T, Vassberg J, *et al*. Data summary from the first AIAA computational fluid dynamics drag prediction workshop[J]. AIAA Journal of Aircraft, 2003,40(5): 875 – 882.

[33] Rakowitz M, Sutcliffe M, Eisfeld B, *et al*. Structured and unstructured computations on the DLR-F4 wing-body configuration[R]. AIAA Paper 2002-0837, 2002.

[34] Laflin K, Klausmeyer S, Zickuhr T, *et al*. Summary of data from the second AIAA CFD drag prediction workshop[R]. AIAA Paper 2004-0555, 2004.

[35] Brodersen O, Rakowitz M, Amant S, *et al*. Airbus ONERA and DLR results from the 2nd AIAA drag prediction workshop[R]. AIAA Paper 2004-0391, 2004.

[36] Brodersen O, Stürmer A. Drag prediction of engine-airframe interference effects using unstructured Navier-Stokes Calculations[R]. AIAA Paper 2001-2414, 2001.

[37] Rudnik R, Rossow C C, Geyr H V. Numerical simulation of engine/airframe integration for high-bypass engines [J]. Aerospace Science and Technology, 2002, 6:31 – 42.

[38] Brodersen O. Drag prediction of engine-airframe interference effects using unstructured Navier-Stokes calculations[J]. AIAA Journal of Aircraft, 2002, 39(6): 927 – 935.

[39] Melber S. 3D RANS simulations for high-lift transport aircraft configurations with engines[R]. DLR-IB 124-2002/27, 2002.

[40] NETGEN, URL: http//www. hpfem. jku. at/netgen [cited 2004].

[41] Brodersen O, Wild J. DLR-IB 124-2004-18[Z]. Braunschweig, 2004.

[42] Pahlke K, van der Wall B. Chimera simulations of multibladed rotors in high-speed forward flight with weak fluid-structure-coupling [A]. 29th European Rotorcraft Forum [C]. Friedrichshafen, Germany, 2003.

[43] Le Chuiton F. Chimera simulation of a complete helicopter with rotors as actuator discs[A]. STAB 2004, Notes on Numerical Fluid Mechanics and Multidisciplinary Design (NNFM)[C]. Springer Verlag, 2005.

[44] Braun C, Bouche A, Ballmann J. Numerical study of the influence of dynamic pressure and deflected ailerons on the deformation of a high speed wing model[A]. In: Krause E, Jaeger W, Resch M, Eds. High Performance Computing in Science and Engineering 2004[C]. Springer, 2004.

[45] Takanashi S. Iterative three-dimensional transonic wing design using integral equations[J]. AIAA Journal of Aircraft, 1985, 22(8): 655 – 660.

[46] Bartelheimer W. An improved integral equation method for the design of transonic airfoils and wings[R]. AIAA Paper 95-1688, 1995.

[47] Wilhelm R. An inverse design method for designing isolated and wing-mounted engine nacelles[R]. AIAA Paper 2002-

0104, 2002.

[48] Wild J. Validation of numerical optimization of high-lift multi-element airfoils based on Navier-Stokes-equations[R]. AIAA Paper 2002-2939, 2002.

[49] Jameson A, Martinelli L, Pierce N. Optimum aerodynamic design using the Navier-Stokes equations[J]. Theoretical Computational Fluid Dynamics, 1998,10:213-237.

[50] Gauger N R. Das adjungiertenverfahren in der aerodynamischen formoptimierung[R]. DLR-FB 2003-05, 2003.

[51] Brezillon J, Gauger N R. 2D and 3D aerodynamic shape optimization using adjoint approach[J]. Aerospace Science and Technology, 2004,8:715-727.

[52] Lovell D A. Aerodynamic research to support a second generation supersonic transport aircraft [A]. The EUROSUP Project ECCOMAS[C]. 1998.

Biographies:

Cord-Christian ROSSOW born in 1957, received his M.S. (Diplomingenieur) from Technische Universitaet Braunschweig in 1984. He joined DLR in 1984, and received his doctoral degree in 1988 from Technische Universitaet Braunschweig. In 2003, he became director of the DLR Institute of Aerodynamics and Flow Technology and full professor at Technische Universitaet Braunschweig. Tel: (0049) 531 295 2400, E-mail: cord.rossow@dlr.de

Norbert KROLL born in 1954, received his Diploma in Mathematics from Bonn University in Germany in 1981 and his doctoral degree in aeronautical engineering from the Technical University in Braunschweig in 1989. He joined the DLR Institute of Aerodynamics and Flow Technology in 1982. Since 1990 he is head of the Numerical Methods Branch. He was project leader of several national projects on the development and validation of numerical methods for aircraft design. Since 2003 he is professor at the Applied University of Braunschweig/Wolfenbüttel. Tel: (0049) 531 295 2440, E-mail: norbert.kroll@dlr.de

Dieter SCHWAMBORN born in 1952, received his Diploma in Aerospace Engineering from the Technische Hochschule Aachen in 1977. Afterwards he was awarded a scholarship from DLR in Köln which lead to his doctoral degree from the Technische Hochschule Aachen in 1981. In 1980 he joined the DLR in Goettingen, where he is heading the Numerical Methods Branch since 1990, first in the Institute of Fluid Mechanics and now in the Institute of Aerodynamics and Flow Technology. He has published several scientific papers in various periodicals. Tel: (0049) 551 709 2271, E-mail: dieter.schwamborn@dlr.de