ONGOING ACTIVITIES IN FLOW SIMULATION AND SHAPE OPTIMIZATION WITHIN THE GERMAN MEGADESIGN PROJECT

N. Kroll*, K. Becker, H. Rieger***, F. Thiele******

***German Aerospace Center (DLR), D-38108 Braunschweig, Germany**
e-mail: norbert.kroll@dlr.de

****Airbus, D-28183 Bremen, Germany**
e-mail: klaus.becker@airbus.com

*****EADS Deutschland GmbH, Military Aircraft, D-81663 München, Germany**
e-mail: herbert.rieger@eads.com

******Technical University of Berlin,**
Institute of Fluid Mechanics and Technical Acoustics, D-10623 Berlin, Germany
e-mail: frank.thiele@cfd.tu-berlin.de

Abstract

Within the framework of the third German aviation research program (2003-2007), the CFD project MEGADESIGN was initiated. The main goal of the project is the improvement and enhancement of the simulation and optimization capabilities of the German MEGAFLOW software. The project deals with several key issues regarding physical modeling, solver efficiency, fluid/structure coupling and advanced strategies for shape optimization. In order to meet the requirements of industrial implementations, a co-operative effort has been set up involving German aircraft industry, DLR, several universities and small enterprises specialized in numerical simulation and optimization. The paper outlines the main activities and achievements of the four-year project started mid 2003.

1 Introduction

Over the last decade, high level Computational Fluid Dynamics (CFD) has become a mature technology for the development of new products in aeronautical industry. Aerodynamic design engineers have progressively taken advantage of the possibilities offered by the numerical solution of the Reynolds averaged Navier-Stokes (RANS) equations. Significant improvements in physical modeling and solution algorithms as well as the enormous increase of computer power enable high-fidelity numerical simulations in all stages of aircraft development.

In Germany, the national CFD project MEGAFLOW furthered the development and availability of RANS solvers for the prediction of complex flow problems significantly. MEGAFLOW was initiated by the first aviation research program of the Federal Government in 1995 under the leadership of the DLR ([1], [2], [3]). A network from aircraft industry, DLR and several universities was created with the goal to focus and direct development activities for numerical flow simulation towards a common aerodynamic simulation system providing both a block-structured (FLOWer-Code) and a hybrid (TAU-Code) parallel flow prediction capability. Today, both codes have reached a high level of maturity and reliability. They are routinely used at DLR and German aeronautic industry for a wide range of aerodynamic applications. For many universities the MEGAFLOW software represents a platform for the improvement of physical models and for the investigation of complex flow problems. The network was established as an efficient group of very closely co-operating partners with supplementing expertises and experience. Focusing on common software, the process of transferring latest research and technology results into production codes used at industry has been considerably accelerated.

Despite the progress made in CFD, future demands of aircraft industry with respect to more environmentally friendly, safer and more economical aircraft require further improvement of simulation capabilities. The need to achieve reliable results at a high level of accuracy for complex configurations within short turn-around time places severe constrains on the application of CFD for aerodynamic data production and the integration of RANS methods into multidisciplinary simulation and optimization procedures. Consequently, enhanced CFD capabilities for reducing design cycle time and cost are indispensable for the industry.

In order to meet future requirements of German aircraft industry, a MEGAFLOW follow-on project was set up within the third aviation program of the Federal Government mid 2003. Based on the MEGAFLOW software the main objectives of the four-years project MEGADESIGN are to ensure the prediction accuracy with a guaranteed error bandwidth for certain aircraft configurations at design conditions, to reduce the simulation turn-around time for large-scale applications significantly, to improve the reliability of the flow solvers for full aircraft configurations in the complete flight regime, to extend the flow solvers to allow for multidisciplinary simulations and to establish numerical shape optimization as a vital tool within the aircraft design process. Partners of the MEGADESIGN consortium are DLR (Institute of Aerodynamics and Flow Technology), Airbus, EADS Military Air Systems, Synaps Ingenieur-Gesellschaft mbH, FastOpt, HPCC Space GmbH, RWTH Aachen University (Department of Mechanics), Berlin Technical University (Institute of Fluid Mechanics and Technical Acoustics), Braunschweig Technical University (Institute for Fluid Mechanics), Darmstadt University of Technology (Institute of Fluid Mechanics and Aerodynamics), Trier University (Department of Mathematics). The project is coordinated by DLR.

The present paper gives on overview of the main activities and results achieved so far. Recent improvements and enhancements of the flow solvers are described, followed by new developments with respect to aerodynamic shape optimization. Improved numerical simulation capabilities are demonstrated by several industrial applications.

2 MEGAFLOW software

The components of the MEGAFLOW software mainly developed at DLR are the block-structured flow solver FLOWer [4], [5] and the unstructured hybrid flow solver TAU [6],[7],[8]. Both codes solve the compressible three-dimensional Reynolds averaged Navier-Stokes equations for rigid bodies in arbitrary motion. The equations are solved by a finite-volume method with second order upwind or central space discretization with scalar or matrix artificial dissipation. In FLOWer cell centered and cell vertex formulations are provided, whereas TAU uses a vertex centered dual mesh formulation. The discrete equations are integrated explicitly by multistage Runge-Kutta schemes, using local time stepping and multigrid acceleration. In FLOWer the explicit scheme is used in combination with implicit residual smoothing, whereas in TAU the implicit LU-SGS scheme is additionally available. For time accurate computations the implicit dual time stepping method is employed. Preconditioning is used for low speed flow simulations. Various turbulence models are available, ranging from eddy viscosity to full differential Reynolds stress models including options for DES. For transition prediction on airfoils and wings FLOWer is coupled to a laminar boundary layer code and an e^N-database method. The Chimera technique enhances the flexibility of FLOWer and TAU with respect to complex geometries or independently moving bodies. For the simulation of aeroelastic phenomena both codes have been extended to allow geometry and mesh deformation. A key feature of TAU is the grid adaptation capability for hybrid meshes based on local grid refinement and wall-normal mesh movement in semi-structured near-wall layers, allowing to efficiently resolve detailed flow features. Both codes, FLOWer and TAU, have been efficiently parallelized and ported to a variety of platforms. According to strategic considerations, MEGADESIGN concentrates on further development of hybrid RANS technology based on TAU. FLOWer is primarily used as exploration platform.

2.2 Improvement of Physical Modeling

With the support of MEGADESIGN and the EU-project FLOMANIA [9] DLR developed the SSG/LRR-ω Reynolds stress model [10],[11]. This model combines the Speziale-Sarkar-Gatski (SSG) model [12] in the far field with the Launder-Reece-Rodi (LRR) model [13] close to solid walls, where according to the Wilcox stress-ω model [14] the wall-reflection terms are omitted in the LRR-part. The length scale of the SSG/LRR-ω model is supplied by Menter's BSL ω-equation [15], which accordingly combines an ε-equation in the far field with the ω-equation of Wilcox [16] at the wall by blending the respective model coefficients. The implementation of the SSG/LRR-ω model is based on detailed stability analysis and exploits an explicit integration scheme previously applied successfully to two-equation eddy viscosity models [17]. The implementation appears rather robust and efficient, allowing the computation of complex flow fields around complex aircraft configurations in cruise and high lift conditions [18]. Fig.1 shows results for the DLR F6 wing/body/nacelle/pylon-configuration and the A310 high-lift 3-element airfoil (EUROLIFT II). The predictions of aerodynamic forces with the SSG/LRR-ω model appear in better agreement with measurements than with classical eddy viscosity models. The SSG/LRR-ω model has been transferred to DLR's unstructured TAU code, where first results confirm the promising experiences made with FLOWer.

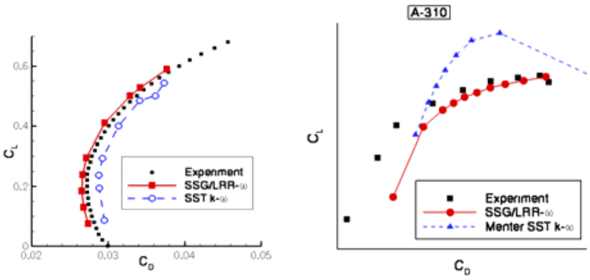


Fig.1: Polar for DLR-F6 wing/body/nacelle/pylon-configuration (left) and for A310 high-lift 3-element airfoil (right), SST-kω model results in left picture provided by ANSYS CFX (2nd DWP).

Based on the modular implementation framework and the proven numerics for Reynolds stress models in FLOWer, Darmstadt University implements an improved model [19] into the code. The idea behind is to more correctly predict the anisotropy of the Reynolds stresses close to solid walls, which is not fully accounted for by the SSG/LRR-ω model due to the omission of wall-reflection terms. Besides including wall-reflection terms and an anisotropic dissipation term, the model coefficients are expressed in terms of carefully calibrated functions of tensor invariants of turbulence. The length scale of the model is supplied by the ε-equation.

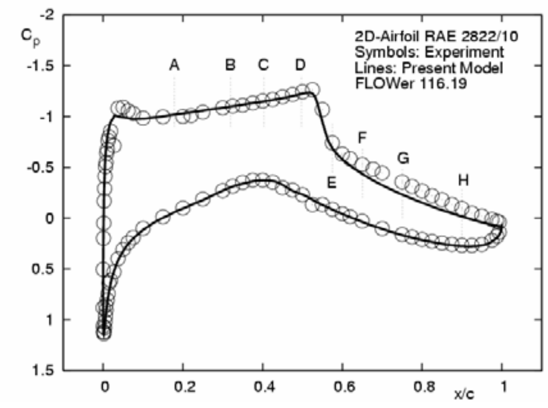


Fig. 2: Pressure distribution for RAE 2822, Case 9, improved prediction by new Reynolds stress model.

First results for the transonic flow around the RAE 2822 airfoil [20] show a very good agreement of the predicted shock position (Fig. 2) and the mean axial velocity profiles (Fig. 3) with the experiments for the so-called cases 9 and 10.

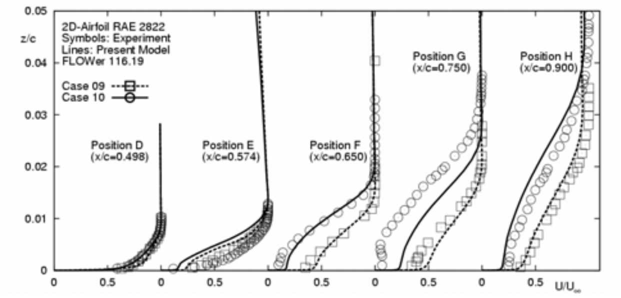


Fig. 3: Mean axial velocity profile evolution over the upper RAE 2822 airfoil surface for Cases 9 and 10 calculated by the new Reynolds stress model .

A transition prediction module based on the hybrid TAU-code was developed at the Institute for Fluid Mechanics at Braunschweig Technical University, which is capable of predicting transition for flow around complex geometries [21]. A major issue of this advanced tool is its applicability for parallel computations. Strategies had to be developed to assemble non-local flow data and to perform transition prediction on decomposed grids without gathering the whole solution domain. Various approaches were implemented for transition prediction, but the focus is put on linear stability theory based on the e^N-method. Boundary layer data is extracted along inviscid streamlines either directly from the RANS solution or is determined from the boundary layer code COCO [22]. The stability solver LILO [23] computes amplification rates that are integrated along the streamlines and analyzed to give the new transition position.

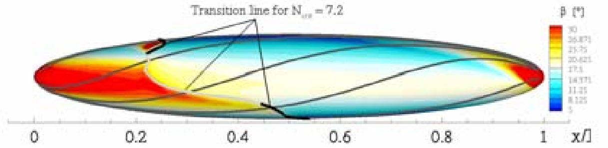


Fig. 4: Inviscid streamlines and transition line on a prolate spheroid at *α=10.0°, M=0.14, Re=7.2x10*^{6}, black transition line: TS transition, light grey transition line: CF-transition, β angle between wall streamline and inviscid streamline

The capabilities of the new prediction module have been demonstrated in various test, e.g. for the flow around an inclined prolate spheroid. A grid with 128 grid points normal to the surface was used, to sufficiently resolve the cross flow velocity profiles and thus to reliably compute both, the cross flow (CS) N-factors and the Tollmien/Schlichting (TS) N-factors. In Fig. 4 the computed transition line is depicted, where a clear distinction between the two transition scenarios is made. The numerical transition line corresponds well with the experimental one (not shown here).

As a test case for more complex geometries, the transition prediction module was applied to the flow around a generic transport aircraft at a moderate angle of attack. A hybrid grid with 12 million grid points was used which was partitioned into 8 domains for parallel computation. Transition was predicted on each surface of the model (body, wing, horizontal and vertical tail plane). Due to the coarse grid resolution (32-48 points normal to the surface in the structured part of the grid), only TS transition was considered. The resultant transition lines are depicted in Fig. 5. As a next step the transition module will be applied to the 3D DLR-F11 high-lift configuration.

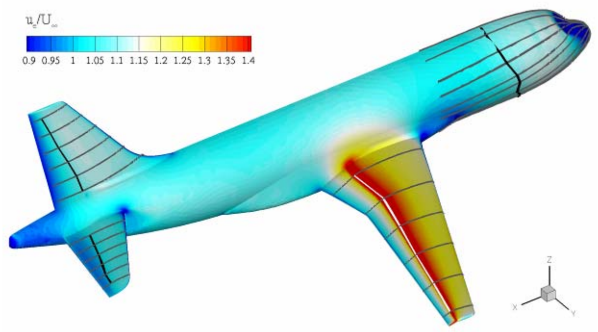


Fig. 5: Inviscid streamlines and transition lines on a generic transport aircraft at α=-4.0°, i_H=4.0°, M=0.2, Re =2.3x10^6, black transition line: TS-transition, white transition line: laminar separation.

2.2 Efficiency improvement

Shape optimization for industrial problems requires large resources. Thus it is mandatory to improve the most costly elements of the optimization chain, in particular the flow solver and the grid generation process. Within MEGADESIGN several activities are devoted to improving the performance of the flow simulation process.

For accuracy reasons usually so-called low-Reynolds number boundary conditions are applied to the turbulence equations, when assessing the aerodynamics of aircraft configurations. This formulation requires the integration of the turbulence equations through the entire boundary layer down to the wall. The first point must therefore be placed at around $y^+=1$ placing severe constraints on grid generation and resulting in computational meshes with a large number of points. On the other hand a so-called high-Reynolds number formulation of the boundary conditions is available, allowing to bridge the buffer and viscous sub-layer by the assumption of turbulent equilibrium. This approach requires the nearest point to the wall to be placed in the logarithmic part of the boundary layer, at a normalized distance of y^+ greater than 20 offering a significant reduction of mesh points in the wall normal direction compared to the high-Reynolds number formulation.

Both wall boundary conditions have very differing requirements concerning the mesh resolution in the wall-normal distance, any violation of which leads to rapid degeneration in the quality of the numerical solution. Thus the automation of the grid generation, required within the shape optimization process, is hampered. Within MEGADESIGN the Berlin Technical University implemented a hybrid-adaptive wall boundary condition [24] for the block-structured FLOWer code providing a single boundary condition for all types of structured grids. Its particular formulation uses a Dirichlet condition of ω in the first wall node, a modified source term in the transport equation for the turbulent kinetic energy, stress transformation for the turbulent viscosity and an asymptotic transition between the viscous and inertial sub-layers. A series expansion of y^+ and a blending for the wall shear stress as a function of y^+ was introduced. For the turbulent length scale a Wolfstein-like assumption was made. The hybrid boundary condition was tested and validated for several 2D and 3D cases including cruise and high-lift conditions. The capability of the hybrid-adaptive boundary condition is demonstrated for the RAE2882 airfoil in Fig. 6. Results for Case 9 conditions computed on both types of meshes suitable for high-Re and low-Re boundary conditions using the different boundary conditions are shown. It can be clearly seen that in both cases where the inappropriate boundary condition is used, a dramatic under-prediction of the skin friction coefficient ensues. The hybrid boundary condition gives very similar results compared to the appropriate boundary condition on each mesh. The poorer prediction of the prediction of the shock locations for both the high-Re and hybrid adaptive conditions on the high-Re mesh may be due to the strong non-equilibrium turbulence present in the boundary layer, which invalidates the physical hypothesis upon which the high-Re boundary condition is based. For routine use further validation is needed.

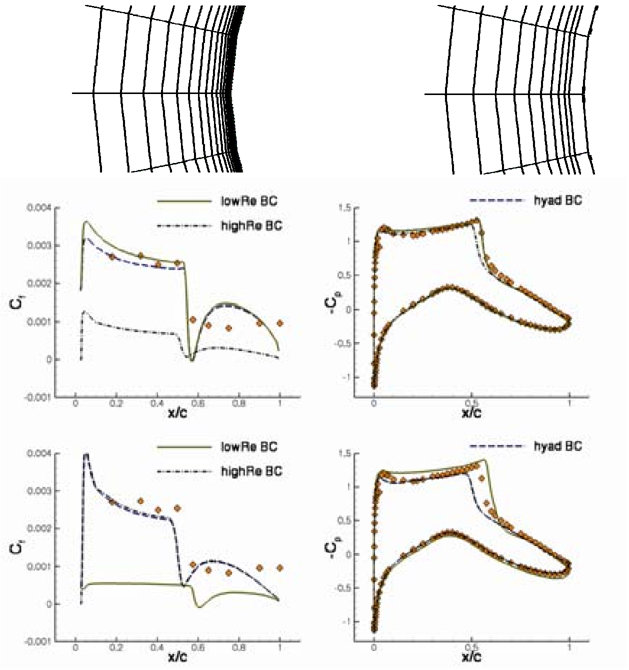


Fig. 6: Comparison of results obtained by low-Re, high-Re and hybrid-adaptive boundary condition for the RAE2822 airfoil (case 9) on a mesh suitable for low-Re boundary condition (upper figure) and on a mesh suitable for high-Re boundary conditions (lower figure).

Within a DLR-internal project, recently a universal wall function has been implemented into the hybrid RANS solver TAU showing very promising results [25].

One of the most critical issues of numerical shape optimization for aircraft design is the efficiency of flow solver algorithms, in particular for more complex applications on unstructured meshes. In order to improve the performance and robustness of the unstructured TAU-Code, implicit methods following different strategies are studied at DLR [26]. Due to its enormous memory requirements the exact Newton method is prohibitive for practical engineering cases. Various approximations of the Jacobians in combination with different solvers for the corresponding linear equations are investigated aiming at a good compromise between memory requirements and performance. In Fig. 7 two implicit methods are compared with the baseline Runge-Kutta time-stepping scheme combined with multigrid for the transonic RAE2822 aerofoil with Spalart-Almaras turbulence model. Using LU-SGS in combination with multigrid, which has similar memory requirements to Runge-Kutta, a speed up of 40% in CPU time is obtained. By allowing sufficient memory to store a complete first-order approximation to the Jacobian and solve the resulting linear system with a Krylov method the so-called FOKI scheme may be constructed, which improves on LU-SGS by a factor of 3-4 in CPU time. The mean-flow and turbulence equations are decoupled in FOKI allowing some flexibility as to amount of work assigned to each system, resulting in small improvements as shown.

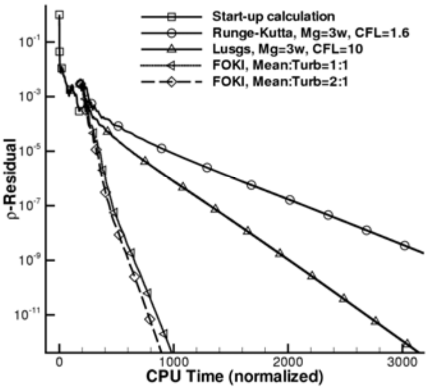


Fig. 7: Convergence of different solution algorithms for RAE2822 airfoil in viscous transonic flow.

3 Numerical Optimization

Numerical shape optimization will play a strategic role for future aircraft design. However, detailed aerodynamic shape optimization based on high-fidelity flow solvers requires extremely high computational resources when employing straightforward optimization strategies to industrially relevant problems. Within MEGADESIGN several research activities address the development of innovative strategies for efficient aerodynamic and multidisciplinary optimization.

3.1 Gradient based optimization strategies

Derivatives of the cost function with respect to design variables can be efficiently obtained by the solution of the adjoint flow equations, enabling the prediction of sensitivities independent of the number of design variables. In the past a continuous adjoint solver was developed for the block-structured FLOWer-Code [27]. Various applications to 2D and 3D flows employing the Euler adjoint solver demonstrated the high potential of this approach for efficient detailed shape optimization [28]. Within MEGADESIGN DLR addresses the development of an adjoint solver for the unstructured TAU-Code. For inviscid flows both, the continuous and discrete adjoint formulation for the Euler equations dealing with drag, lift and pitching moment sensitivities were implemented. For various 2D and 3D flows the adjoint sensitivities were successfully validated with corresponding gradients computed by approximate finite differences methods. For viscous turbulent flows effort is concentrated on the discrete adjoint approach, since for the continuous formulation the implementation of suitable boundary conditions and the integration of turbulence models are rather critical.

The formulation of the discrete adjoint equation requires differentiating the corresponding flow solver, including the discrete boundary conditions and turbulence models. For the DLR TAU-Code this has been carried out by hand for most of the solver options by successively applying the chain rule [26], [29]. Given an explicitly stored Jacobian a Krylov subspace method such as GMRES with an appropriate preconditioner can be used to efficiently solve the corresponding linear adjoint equations. In this case the calculation of the adjoint solution requires only 10% of the time needed for a non-linear flow prediction. Given cheap gradients, second-order optimization strategies that relay on many gradient evaluations, such as Quasi Newton Trust Region, become more attractive since they may reduce the overall cost of the optimization process due to improved convergence rates compared to classical first-order gradient based strategies.

Fig. 8 demonstrates the capability of the discrete adjoint approach for drag optimization of an airfoil at constant lift in viscous transonic flow [29]. The design conditions are M_∞=0.73, C_l=0.8055, Re=6.5x10^6 as Mach number, lift and Reynolds number, respectively. 20 design parameters were used changing the airfoil camberline while keeping the maximum thickness. The desired lift is maintained adjusting the angle of attack. The flow calculations were carried out with the TAU-Code on hybrid grids consisting of quadrilaterals and triangles. For turbulence modeling the one-equation Spallart-Almaras Edwards model was used. Optimization was carried out with three strategies of increasing efficiency: steepest decent, conjugate gradient and Quasi-Newton Trust Region (QNTR).

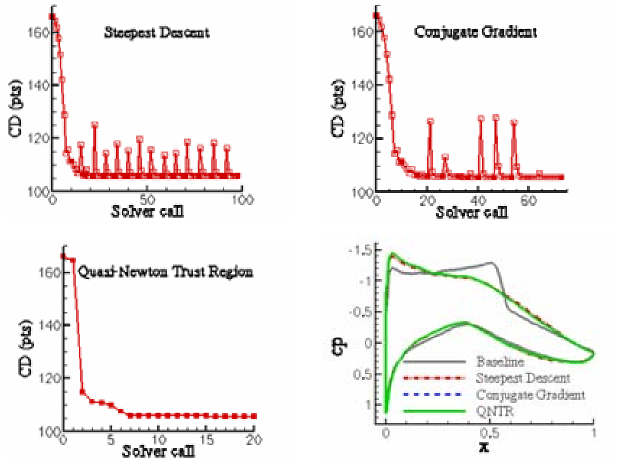


Fig. 8: Airfoil optimization for drag reduction at constant lift in viscous transonic flow using different optimization strategies, sensitivities computed by discrete adjoint solver.

Fig 8 shows drag reduction as function of TAU solver calls for the three different strategies. The optimum is obtained with QNTR with 5 times fewer flow calculations compared to steepest decent. However, QNTR requires more gradient evaluations which can be efficiently predicted by the discrete adjoint solver. Depending on the optimization strategy used, the same optimization with finite differences would be 3 to 8 times slower. The comparison of the pressure distribution in Fig. 8 shows that the final design provides shockfree flow. The pressure distribution is also characterized by an increased suction peak and increased rear loading needed to meet the target lift.

The capability of the adjoint method was also demonstrated for high-lift application. Fig. 9 shows results for the optimization of the flap setting of a multi-element airfoil at take-off condition (M_∞=0.1715, Re=6.5x10^6) [29].

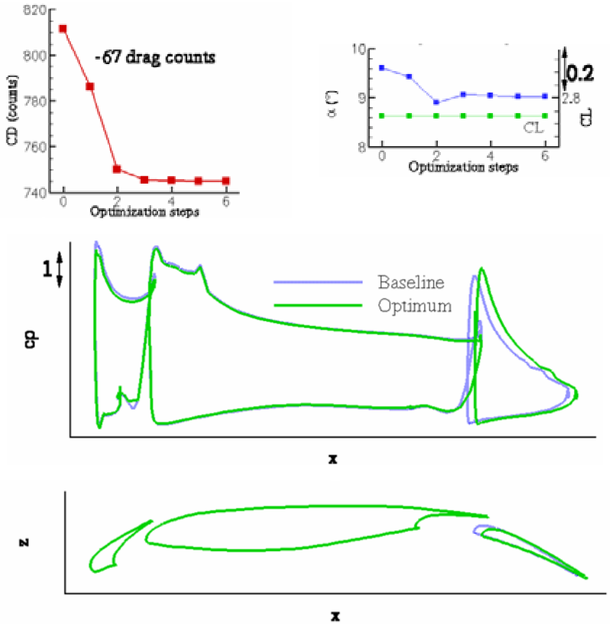


Fig. 9: High-lift flap design based on discrete adjoint, history of the optimization process and pressure distribution for initial and optimized geometry.

The initial geometry and aerodynamic conditions of the configuration were defined in the EU project EUROLIFT II. In this case the gap and the deflection angle of the flap are the design variables, while the goal function is drag reduction at constant lift. The conjugate gradient optimization strategy is used. The upper pictures of Fig. 9 show the evolution of drag, lift and angle of incidence. The geometries and pressure distributions of the initial and optimized configurations are shown in the lower pictures.

The optimization required 6 steps to converge and resulted in a 67 counts lower drag at equal lift.

Explicit storage of the exact Jacobian on the one hand allows an efficient and accurate adjoint solution, but on the other hand it limits its application to 2D problems due to the high memory requirements. Therefore, different possibilities for approximate differentiation of the flow solver were identified which allow a drastic reduction of the memory requirement while ensuring sufficient accuracy for the gradient prediction used in the optimization process [30]. Current activities are devoted to the development of robust linear solvers which offer a good trade-off between efficiency and memory requirements. This will allow taking advantage of the discrete adjoint method for large 3D optimization problems.

Another possibility to generate sensitivities needed for optimization is the use of automatic differentiation (AD) tools. Within MEGADESIGN the SME FastOpt used their AD tool TAF [31] in cooperation with DLR to differentiate the FLOWer-Code. TAF is a so-called source-to-source translation tool for programmes written in FORTRAN 77-95. From the source code representing a function TAF generates a second, well readable source code representing the derivative. After initial preparations, the entire FLOWer code encompassing about 166,000 lines (without comments) was processed by TAF in a fully automated procedure, with the lift declared as dependent and the angle of attack as independent variables. The derivative code thus provides the sensitivity of lift w.r.t. angle of attack. AD was applied both in forward mode (tangent linear model) and in reverse mode (adjoint model). Two versions of the adjoint have been generated, one for steady flows, based on the assumption of a converged primal solve ([32]) and as such comparable to the adjoints produced by the aforementioned approaches. The second version of the adjoint is for general use, suitable, e.g., for time dependent problems. All generated derivative codes were successfully verified against finite differences in Navier-Stokes setup with five of FLOWer's turbulence models, including the two-equation models such as Wilcox-kω. To provide a primal solution and the gradient, the steady flow version of the adjoint requires three times the memory of the primal and, depending on the platform, a factor of 6-10 in CPU time. Ongoing activities focus on automated generation of yet faster adjoint code and on generation of derivatives w.r.t. the coordinates of the mesh points, as is necessary for shape design.

Trier University is developing fast one-shot methods for aerodynamic shape optimization. The term "one-shot" refers to the fact that, in contrast to textbook optimization approaches, the simulation problem is solved in parallel to the optimization problem. Classical gradient based techniques consist of three basic steps: (1) determination of the flow field for a given geometry, (2) computation of gradients with respect to the geometry – favorably by the adjoint method – and (3) update of the geometry according to the gradient information. These steps are repeated until convergence. At least the first and third step typically involve iterations by themselves, which ultimately lead to large computational effort for the optimization task compared to a single flow solution. Factors of 30 to 40 are a good lower bound. In contrast to that, the one-shot method basically performs only single iterations of the iterative techniques employed in the first and third step. The number of outer iterations is increased, but the overall numerical effort is reduced to an equivalent of 4 flow solutions for drag minimization only [33] and to an equivalent of 7 to 10, if additional state constraints like lift and/or pitching moment are to be preserved as well [34]. The methodology has been implemented so far for 2D and 3D cases within the pseudo-time-stepping framework of the Euler formulation of FLOWer. Comparison with classical steepest descent methods yielded the same numerical results in a much less time. Mathematical investigations show that the convergence of the method observed so far in numerical experiments can be theoretically justified based on a reduced SQP point of view [36].

Fig. 10 illustrates the capability of the one-shot method for airfoil drag reduction at constant lift in inviscid transonic flow [34]. The left-top figure shows the residuals of the flow problem and the adjoint problems for drag and lift during the iterations. After some initial iterations defining a starting point close to feasibility, the optimization is started. The iterations are stopped, if no further progress is noticeable. Afterwards, the state solver is used to compute an appropriate flow solution which possesses an accuracy that is compatible with a textbook steepest descent approach. The right-top and the left-bottom picture show the history of the drag and lift coefficient. The right-bottom figure show pressure profiles before and after optimization (right).

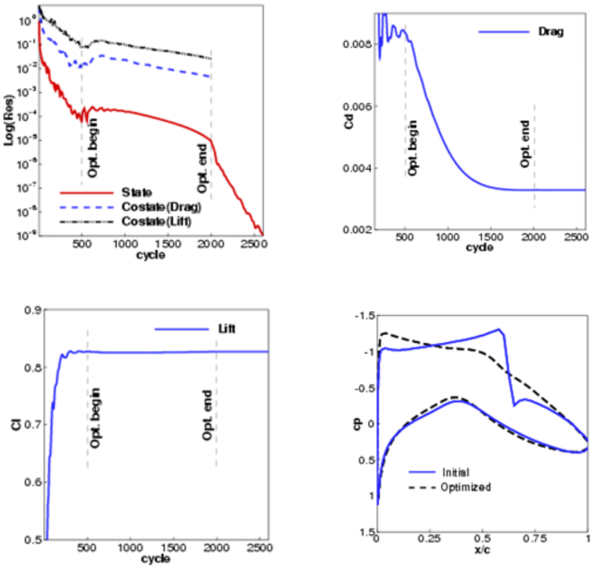


Fig. 10: Drag reduction of airfoil at constant lift with one-shot method, inviscid transonic flow.

The one-shot methodology is not limited to inviscid flow problems and is currently enhanced also to viscous flow modelled by the Navier-Stokes equations.

3.2 Approximation of design space

Within the MEGADESIGN project a major effort of SYNAPS is the development of a sophisticated approximation of the solution spaces for numerical optimization based upon Artificial Neural Networks (ANN). The idea for the application of ANN is to find a way for the prediction of promising sets of design variables within an optimization process (candidate selection). This way it will be possible to avoid the costly calculation of useless high-fidelity solutions and thus reduce the amount of time significantly. ANNs consist of layers, where the first one is fed with the input and the last one represents the result. In between there are so called hidden layers which do the nonlinear transformation depending on the cross-linking of neurons.

In the beginning it was unclear, what kind of topology such an ANN should have. The prediction capability, however, strongly depends on the selected topology. Therefore, the first step was to investigate the correlation between the number of design variables and the number of neurons of the network. It was possible, by running a large number of optimizations and applying all possible network topologies, to find a distinct dependency.

Based upon this information, neural networks can be set up automatically depending on the number of design variables. The concurrently developed universal ANN software was integrated into the SynapsPointer® Pro software [37] and tested with different applications. The approach proved to be reliable and the result is a reduction of about two third in the number of necessary high-fidelity calculations.

3.4 Multidisciplinary optimization using reduced structural models

Within the scope of MEGADESIGN, the Mechanics Department (LFM) at RWTH Aachen University contributes to the development and application of numerical aero-structural dynamics methods based on reduced structural models. Two major program modules are developed and supplied: the Beam Generator (BG) and the Aeroelastic Coupling Module (ACM). These tools are to be integrated into a multidisciplinary optimization (MDO) software package enabling the code for combined aircraft shape and structural optimization.

The BG is used to automatically create reduced structural models by representing the wing box as a multi-axial Timoshenko beam. The module enfolds the Beam Preprocessor (BP) and the FE-program FEAFA (Finite Element Analysis for Aeroelasticity). The wing box geometry is configured by the BP from sections through the aerodynamic surface mesh, with spar positions and sheet thicknesses being supplied by the optimizer in each MDO step. The BP calculates the equivalent structural beam properties with analytical methods using an idealization as thin-walled cell sections. The equivalent Timoshenko beam is characterized by the cross-sectional coordinates of the centers of mass, shear and bending. This data is relayed to FEAFA which assembles the mass and stiffness matrices for use by the ACM. Furthermore, the BP allows the determination of the element-wise and total structural weight, the cross-sectional area for the definition of fuel loads, and data permitting the estimation of the section-wise maximum equivalent stress. The latter can be used by the optimizer to evaluate the yield stress boundary condition once the aerodynamic loads are known. The BG has been validated extensively with different thin-walled structural configurations, comparing the results using FEAFA to those calculated by the commercial FE software MSC/MARC with shell models.

The concept of the ACM enables aeroelastic calculations using any kind of existing CFD solver. The operations performed by the ACM involve the consistent and conservative projection of aerodynamic loads onto the structural model, the calculation of the resulting structural deformations and the ensuing deformation of the aerodynamic surface mesh. Any of the numerous available commercial FE codes for structural analysis can easily be combined with the ACM.

As an additional module of the coupled aeroelastic method an efficient and robust grid deformation algorithm based on a fictitious framework of beams matches the CFD mesh with the deformed aerodynamic surface provided by the ACM.

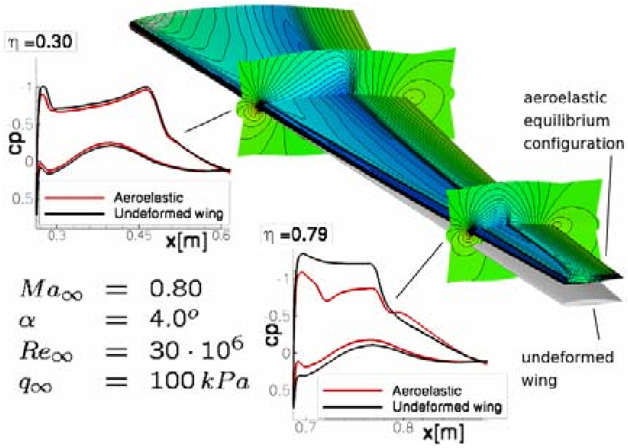


Fig. 11: Aeroelastic equilibrium with corresponding pressure isolines for the HIRENASD wing computed with FLOWer-ACM.

The ACM has been successfully used together with the block-structured flow solver FLOWer in various projects [38], [39] and is regarded as a proven tool. Fig. 11 shows exemplary results for the HIRENASD wing [40], which is defined as a test case within MEGADESIGN. The validation of the ACM coupled with the unstructured TAU-Code is currently underway.

The capability to automatically generate reduced structural models while varying structural model parameters and using them for aero-elastic calculations has already been proven. With both the ACM and the BG operative, the focus now lies upon using these elements within multidisciplinary optimization.

4. Industrial Simulation Process Chains

Main focus of aircraft industry's activities is directed to build-up industrial processes to handle reliable aerodynamic and multi-disciplinary simulation and optimization problems taking into account new developments achieved within the MEGADESIGN project. In the following some results from Airbus and EADS-MAS are shown that demonstrate the improved simulation capabilities.

4.1 Civil aircraft applications

Within the MEGADESIGN project, the overall strategy at Airbus is to establish a "Hybrid RANS CFD chain architecture" that is defined for fully parallel operation on distributed hardware at a very high level of automatization. This architecture is mainly based on the DLR TAU-Code and includes all accompanying steps such as geometry generation and modification, mesh generation and deformation, flow solution, post-processing and monitoring as well as interdisciplinary optimization. By integrating the different modules on a user-friendly plug-in basis into a single CFD chain, advantage can be taken by increasing efficiency of the overall design process, but also accuracy can be improved by easily exchanging certain modules and monitoring/comparing the results.

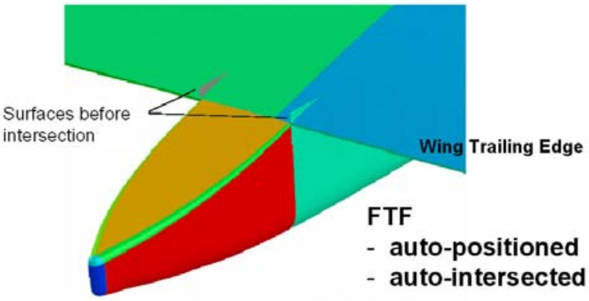


Fig. 12: Sample of automated geometry generation and modification.

A sample of positioning a flap-track-fairing (FTF) underneath a wing is given in Fig 12. After changing the shape and/or position of the FTF the intersections between wing and FTF are redefined automatically. Then the resulting CAD surfaces/intersections are cleaned and made waterproof. Features like these are absolutely necessary when aiming for automated shape optimization.

Fig. 13 demonstrates a full process chain application for comparing different hybrid /unstructured meshes for a complex flow problem (wing/body/pylon/nacelle/flap-track-fairings and large winglet at cruise condition). Plugged-in boundary layer profile evaluation [41] and far-field drag-decomposition [42], [43] modules provide deep insight in the solutions, revealing good agreement even with respect to the boundary layer development but small differences in spurious drag as a mesh quality measure (Fig 14).

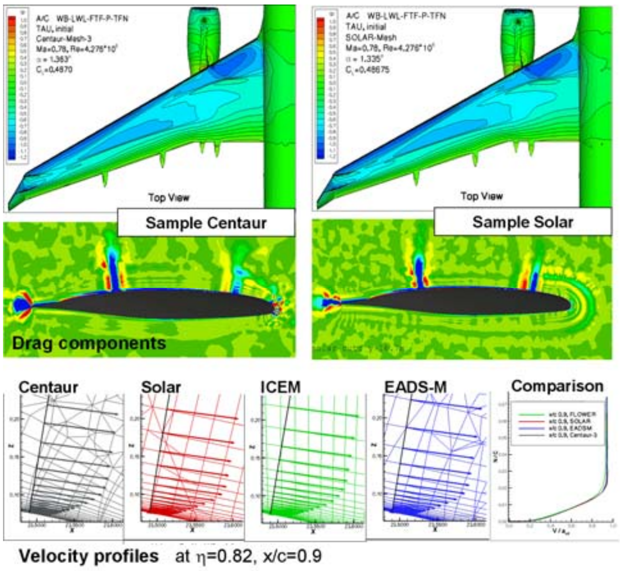


Fig 13: Demonstration of full process chain application including flow analysis: pressure maps, drag components and boundary layer evaluation.

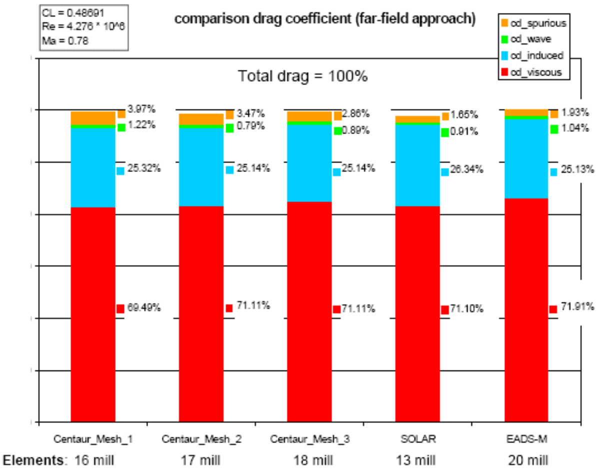
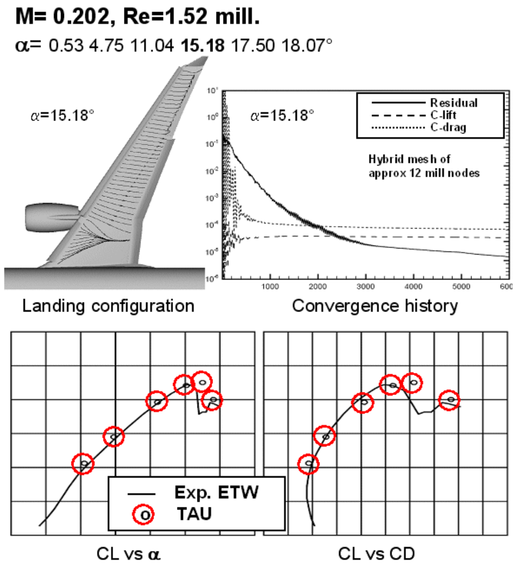


Fig 14: Drag decomposition of flow solutions within different meshes

In addition, the overall high efficiency of the process chain had been demonstrated by “overnight” computation of a lift-drag polar up to the maximum lift regime for a typical transport aircraft in landing configuration (wing/body/slats/flaps/pylon/engine). By using 288 processor units in parallel, this goal had been reached within 12.9 hours. A short summary of results is given in Fig 15. The correlation between experimental data and computed results is rather good.



Landing configuration

Convergence history

CL vs α

CL vs CD

Fig 15: Overview on results obtained from massively parallel “overnight” polar computations.

A significant step towards increased accuracy is seen in the incorporation of wing aeroelasticity. Fig 16 shows the achievements by a coupled CFD-CSM procedure i.e. coupling TAU flow solutions and NASTRAN structural modelling for a wing/body-configuration.

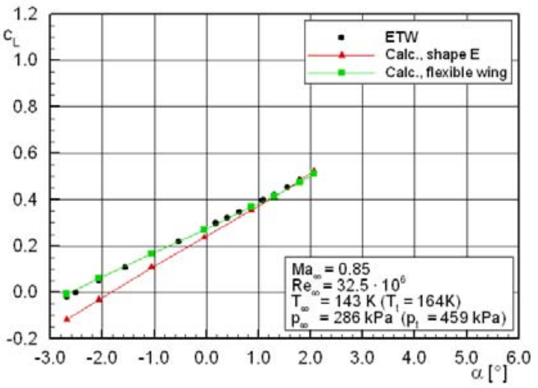


Fig 16: Improvement by fluid-structure coupling TAU – NASTRAN, wing/body-configuration

Next steps will cover the complete and efficient integration of the computational elements demonstrated above into the aforementioned single computational chain, aiming at interdisciplinary and full multi-point optimization.

4.2 Military aircraft applications

For EADS-MAS a key element is the tight integration of all necessary processes to generate geometry models and computational meshes, to perform accurate flow analysis and to extract engineering data as part of an optimization loop.

The approach chosen is centred around the optimization software *modeFRONTIER* [44] which can handle a great variety of optimization problems focusing on genetic and evolutionary algorithms. To enable parameterized shape modifications which can immediately be used in the CAD-based design process, EADS-MAS has chosen CatiaV5 deserving as a geometry engine. By design parameter tables an automatic geometry generation process in batch mode is enabled. In Fig. 17 the process is illustrated for a generic multidisciplinary optimization work flow. In the *modeFRONTIER* system the optimization problem is set up by defining the objectives, the constraints and the work flow for obtaining the analysis results. Using advanced CFD simulation as an analysis method, shape optimization has to be embedded into a chain of process elements (PE's). Those are parametric geometry generation (PE-1), mesh generation (PE-2), CFD solution (PE-3), analysis of results and visualisation (PE-3) as well as determination of structural deformation and stresses (PE-4). For a coupled aero-structural analysis the results have to be computed iteratively between (PE-3) and (PE-4).

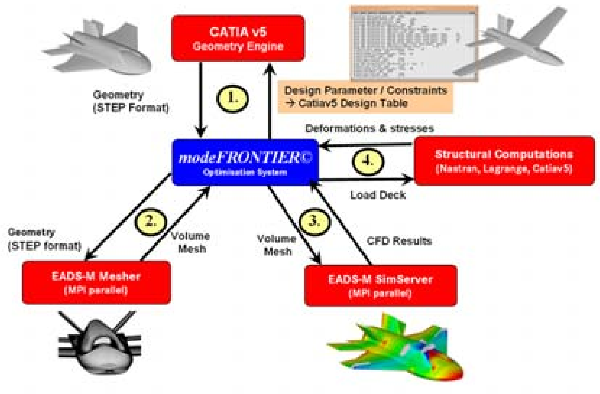


Fig. 17: Parametric CATIAV5-based optimization process centered around modeFRONTIER.

In order to substantially reduce simulation turn-around-time, the parallel implementation of the RANS solver alone will not overcome completely all the inconveniences and problems linked to increasing problem sizes and necessary data exchange processes. An efficient aerodynamic simulation environment therefore should be designed from the beginning as a parallel and memory distributed system. At EADS-MAS a systematic development is underway to realize such a high performance simulation environment for aerodynamic and multidisciplinary applications. This environment, called SimServer [45], heavily relies on an object-oriented programming model and MPI as the distributed communication system paradigm. The intended goal of the SimServer effort is to run complex multidisciplinary simulation problems on cost-efficient cluster computer systems built up from COTS hardware components. The targeted mid-term application was formulated to be the simulation of a fighter aircraft in manoeuvring flight operating all control surfaces under aero-elastic deformation and being able to react according to the integrated flight control system model. Pre-requisite for a corresponding simulation system as outlined in Fig. 18 is a logical model of the flight vehicle which enables the controlled motion of the vehicle and its controls as well as any aero-elastic deformation arising from aerodynamic and dynamic loads.

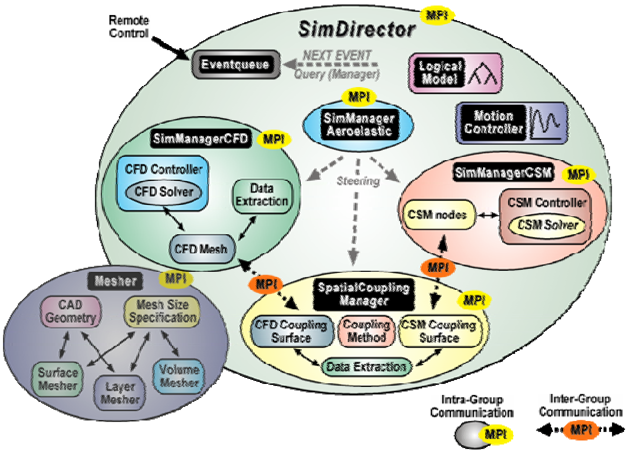


Fig. 18: EADS-MAS SimServer for distributed parallel multidisciplinary simulations.

The SimDirector as the control instance of the simulation environment steers several sub-controllers to control the CFD and the CSM solvers, to manage the spatial coupling processes, to initiate any meshing operations and to extract from the results all necessary information for the optimization process or to perform visualisation tasks as appropriate. As main CFD solver the DLR TAU-Code developed is integrated into the SimServer.

An important element of the SimServer is the automatic mesh generator MESHER [46] of EADS-MAS. CAD-based geometrical definition of the bodies to be meshed is realized by the STEP and IGES format. Some basic CAD cleaning is integrated into the MESHER. For a high-quality surface mesh generation the MESHER generates surface triangulations directly on the NURBS definition of the various trimmed surface patches and distributes triangle sizes according to the local curvatures of the underlying surfaces. Within the MEGADESIGN project the implementation of the layer mesh generation was generalized and improved with respect to robustness. From the SimServer point of view the mesh generator as well as the re-meshing component is fully integrated into the system [47].

As part of the MEGADESIGN main milestones EADS-MAS performed a planform optimization for a generic UAV configuration using all process elements developed so far. The demonstration was set up as a multi-objective optimization partly using design information from various disciplines. Those are currently based on conceptual design assumptions. However the concept will be refined in the very near future step by step in substituting simplified analysis by advanced methods to develop a real multi-disciplined optimization process. For simplicity the generic mission profile consists of three mission phases: the cruise-in and cruise-out mission part as well as the loiter phase as the main focus of the overall mission. To simplify the problem further it was assumed that cruise-in and cruise-out conditions are identical and characterized by transonic flight for a certain distance. The loiter phase should be operated under subsonic conditions at a medium flight level. The objective of the optimization is to maximize the missions loiter time. Optimization variables are the wing aspect ratio, the wing sweep angle, and the inner and outer wing taper ratios within certain limits. It is assumed that the optimization concerns a single engine configuration where the mission weight is kept constant during cruise and loiter phase. To account for realistic cruise and loiter flight conditions a trimmed state is necessary. The corresponding angle-of-attack and angle of incidence of the horizontal tail plane is derived from linear aerodynamic assumptions whereas the drag data are extracted from corresponding RANS simulations. To allow for a natural restriction in wing size the corresponding wing weight is calculated from approximate weight formulas derived from first order bending moment analysis of a simple wing box. The resulting cruise and loiter times are determined from fuel consumption estimates of a corresponding high bypass engine.

The optimization problem posed is characterized by a single objective function (maximization of loiter time) which can be handled by any gradient-based optimization method. As a very promising and robust technique the Single Objective De-randomized Evolution Strategy (DES) [48] was chosen, as that technique has proven to perform well for optimization problems showing local extrema. Moreover the DES method has some advantages over genetic approaches as the computational efforts in terms of analysis steps to get to the optimum are considerable less. For the present optimization problem per population generation 12 individuals were chosen and the optimization was run over 10 generations. A variance of 75% was allowed for the variation of optimization parameters. On a compute cluster with dedicated 48 processor nodes the problem was run for more than one hundred analysis steps during four days. The maximal loiter time of approximate 4.7h is achieved for a wing aspect ratio of approximate 4.7, a wing sweep angle of around 20° and taper ratios for the inner and outer wing of around 0.4 and 0.94. Fig. 19 shows planforms of the determined optimal configuration and of extreme non-optimal configurations which demonstrate the design space being explored during the optimization run. It is worth mentioning that all the visualizations were executed automatically as a specific SimServer action.

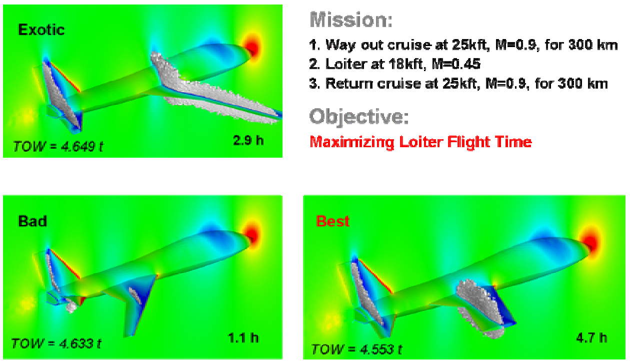


Fig. 19: Visualization of pressure distribution with shock regions for exotic, bad and best configuration, planform optimization for generic UAV with respect to maximizing loiter time.

The results convincingly demonstrate an automatic optimization loop in which various programs were compiled into a heterogeneous process. The solved optimization problem is inherently a multi-disciplinary problem which will be refined further during the runtime of the MEGADESIGN project. Future steps involve the integration of adjoint solvers in the *modeFRONTIER* environment in order to make best benefit from the combination of various optimization strategies. Other activities will include the parameterization of the baseline structures and the set-up of a truly multi-disciplinary and multi-objective optimization process.

5. Conclusions

MEGADESIGN is the central German activity in the area of CFD-related development for aerospace applications and involves DLR, aircraft industry, small enterprises and several universities. It is a continuation of the German MEGAFLOW initiative and R&T network, in which the flow solvers FLOWer and TAU have been developed initially. The project deals with key issues regarding the improvement and enhancement of simulation and optimization capabilities for industrial applications. Although the project is still ongoing, significant improvements with respect to physical modeling, numerical algorithms, fluid/structure coupling, optimization strategies and automation of the simulation process chain were achieved. The new capabilities were demonstrated for several industrial applications.

Acknowledgement

The authors would like to thank all colleagues of the participating institutions of the MEGADESIGN project who carried out the developments and obtained the results shown in this paper. They are too numerous to be mentioned by name. Furthermore, the partial funding of the MEGADESIGN project through the German Government in the framework of the third air transport research program is gratefully acknowledged.

References

[1] Kroll, N., Rossow, C.C., Becker, K., Thiele, F.: *The MEGAFLOW project*, Aerosp. Sci. Technol. Vol. 4, pp. 223-237, 2000.

[2] Kroll, N., Rossow, C.C., Schwamborn, D., Becker, K., Heller, G.: *MEGAFLOW – A Numerical Flow Simulation Tool for Transport Aircraft Design*, ICAS-2002-1105.20, 2002.

[3] Kroll, N., Fassbender, J.K.: (Eds). *MEGAFLOW – Numerical Flow Simulation for Aircraft Design*; Notes on Numerical Fluid Mechanics and Multidisciplinary Design, Volume 89, Springer, 2005.

[4] Raddatz, J., Fassbender, J.K.: *Block Structured Navier-Stokes Solver FLOWer*, In: Kroll, N., Fassbender, J. (Eds) *MEGAFLOW – Numerical Flow Simulation Tool for Transport Aircraft Design*, Notes on Multidisciplinary Design, Vol. 89, Springer, 2005.

[5] Kroll, N., Radepiel, R., Rossow, C.C.: *Accurate and Efficient Flow Solvers for 3D Applications on Structured Meshes*, AGARD R-807, 4.1-4.59, 1995.

[6] Galle, M.: *Ein Verfahren zur numerischen Simulation kompressibler, reibungsbehafteter Strömungen auf hybriden Netzen*, DLR-FB 99-04, 1999.

[7] Gerhold, Th.: *Overview of the Hybrid RANS TAU-Code*, In: Kroll, N., Fassbender, J. (Eds) *MEGAFLOW – Numerical Flow Simulation Tool for Transport Aircraft Design*, Notes on Multidisciplinary Design, Vol. 89, Springer, 2005.

[8] Schwamborn, D., Gerhold, Th., Heinrich, R.: *The DLR TAU-Code: Recent Applications in Research and Industry*, ECCOMAS CFD 2006, The Netherlands, 2006.

[9] Haase, W., Aupoix, B., Bunge, U., Schwamborn, D.: *FLOMANIA – A European Initiative on Flow Physics Modelling*, Notes on Numerical Fluid Mechanics and Multidisciplinary Design, Vol. 94, Springer, to appear 2006.

[10] Eisfeld, B.: *Implementation of Reynolds stress models into the DLR-FLOWer code*, DLR-Institutsbericht, DLR-IB 124-2004/31, 2004.

[11] Eisfeld, B., Brodersen, O.: *Advanced Turbulence Modelling and Stress Analysis for the DLR-F6 Configuration,* AIAA-Paper 2005-4727, 2005.

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

[13] Launder, B.E., Reece, G.J., Rodi, W.: *Progress in the development of a Reynolds-stress turbulence closure*, Journal of Fluid Mechanics, Vol. 68 No. 3, pp. 537 – 566, 1975.

[14] Wilcox, D.C.: *Turbulence Modeling for CFD*, DCW Industries, 2nd ed., 1998.

[15] Menter, F.R.: *Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications*, *AIAA Journal*, Vol. 38, pp. 1598 – 1605, 1994.

[16] Wilcox, D.C.: *Reassessment of the Scale-Determining Equation for Advanced Turbulence Models*, AIAA Journal, Vol. 26, pp. 1299 – 1310, 1988.

[17] Fassbender, J. K.: *Improved Robustness for Numerical Simulation of Turbulent Flows around Civil Transport Aircraft at Flight Reynolds Numbers*, DLR-Forschungsbericht, DLR-FB 2003-09, 2004.

[18] Eisfeld, B.: *Computation of Complex Compressible Aerodynamic Flows with a Reynolds Stress Turbulence Model*, Int. Conference on Boundary and Interior Layers, BAIL, Göttingen, Germany, 2006.

[19] Hanjalic, K., Jakirlic, S.: *Contribution towards the Second-Moment Closure Modelling of Separating Turbulent Flows*, Computers and Fluids, Vol. 22 No. 2 , pp. 137 – 156, 1998.

[20] Jester-Zürker, R., Jakirlic, S., Eisfeld, B.: *Near-Wall, Reynolds-Stress Model Calculations of Transonic Flow Configurations Relevant to Aircraft Aerodynamics*, to be presented at CMFF'06 Conference, Budapest, September 6-9, 2006.

[21] Krimmelbein, N.; Radespiel, R.; Nebel, C.: Numerical Aspects of Transition Prediction for Three-Dimensional Configurations, AIAA-2005-4764, 2005.

[22] 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, 1998.

[23] Schrauf, G.: *LILO 2.0 - User's guide and tutorial*. GSSC Technical Report 6, 2004.

[24] Rung, T., Lübcke, H., Thiele, F.: *Universal wall-boundary conditions for turbulence-transport models,* Zeitschrift für angewandte Mathematik und Mechanik, Vol. 81, pp. 1756-1758, 2000.

[25] Knopp, T., Alrutz, Th., Schwamborn, D.: *A grid and flow adaptive wall-function method for RANS turbulence modelling*, accepted by Elsevier for Journal of Computational Physics, 2006.

[26] Dwight, R.: *Efficiency improvements of RANS-based analysis and optimization using implicit and adjoint methods on unstructured grids*, PHD thesis submitted to the University of Manchester, 2006.

[27] Gauger, N. R.: *Das Adjungiertenverfahren in der aerodynamischen Formoptimierung*, DLR report No. DLR-FB-2003-05 (ISSN 1434-8454), 2003.

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

[29] Brezillon, J., Dwight, R.: *Discrete adjoint of the Navier-Stokes equations for aerodynamic shape optimization*, EUROGEN 2005 –Sixth Conference on Evolutionary and Deterministic Methods for Design, Optimization and Control with Applications to Industrial and Societal Problems, Germany, 2005.

[30] Dwight, R., Brezillon, J.: *Effect of various approximations of the discrete adjoint on gradient-based optimization,* AIAA Aerospace Meeting, Reno, 2006.

[31] Giering, R., Kaminski, T.: *Recipes for Adjoint Code Construction*, ACM Trans. Math. Software 24(4), pp 437-474, 1998.

[32] Giering, R., Kaminski, T., Slawig, T.: *Applying TAF to a Navier-Stokes solver that simulates an Euler flow around an airfoil*, Future Generation Computer Systems 21(8), Elsevier 2005.

[33] Hazra, S.B., Schulz, V. Brezillon, J. Gauger, N.: *Aerodynamic shape optimization using simultaneous pseudo-timestepping.* Journal of Computational Physics, Vol. 204, No.1, pp. 46-64, 2005.

[34] Hazra, S. B., Schulz, V.: *Simultaneous pseudo-timestepping for aerodynamic shape optimization problems with constraints*, to appear in SIAM Journal for Scientific Computing, 2006.

[35] Hazra, S.B, Schulz, V. Brezillon, J.: *Simultaneous pseudo-timestepping for 3D aerodynamic shape optimization*, Forschungsbericht Nr. 05 – 2, Universität Trier, Mathematik/Informatik, 2005.

[36] Hintermüller, M., Schulz, V.: *On the convergence of approximate reduced space optimization iterations*, 2006 in preparation.

[37] http://www.synaps-ing.de.

[38] Ballmann, J. et al.: *The HIRENASD project: High Reynolds number aerostructural dynamics experiments in the European Trannsonic Windtunnel (ETW)*, annual meeting of the International Council of the Aeronautical Sciences (ICAS), Hamburg, Germany, 2006.

[39] Braun, C., Boucke, A., Ballmann, J.: *Numerical prediction of wing deformation of a high speed transport aircraft type wind tunnel model by direct aeroelastic simulation,* International Forum on Aeroelasticity and Structural Dynamics (IFASD) Munich, Germany, paper IF 147, 2005.

[40] Reimer, L., Braun, C., Ballmann, J.: *Analysis of the static and dynamic aero-structural response of an elastic swept wing model by direct aeroelastic simulation,* annual meeting of the International Council of the Aeronautical Sciences (ICAS), Hamburg, Germany, 2006.

[41] John, D.: *A Program to evaluate the boundary layer flow field from Navier-Stokes results in unstructured grids*, Airbus TR L00RP0409678, 2004.

[42] Noske, S.: *Validierung einer alternativen Methode zur Evaluierung von Widerstandskomponenten aus numerischen Loesungen der Navier-Stokes-Gleichungen für Transportflugzeuge*, Diploma Thesis March 2005.

[43] Destarac, D.: *Far-field drag extraction from hybrid grid computations*, AIAA DPW-II Orlando, June 21-22, 2003.

[44] *modeFRONTIER: The Multi-Objective Optimization and Design Environment*, http://www.esteco.com.

[45] Tremel, U., Deister, F., Sorenson, K. A., Rieger, H., Weatherill, N.P.: *An object-oriented parallel multidisciplinary simulation system – the SIMSERVER.* In Joubert, G. R., Nagel, W.E., Peters, F.J, Walter, W.V. (EDs), Parallel Computing: Software Technology, Algorithms, Architectures and Applications, Advances in Parallel Computing, pp 331-338, Elsevier, 2004.

[46] Tremel, U.: *Parallel unstructured adaptive remeshing for moving boundary problems.* PhD thesis, University of Wales Swansea, April 2005.

[47] Tremel, U., Hitzel, S., Sørensen, S.K., Weatherill, N.P.: *JDAM-store separation from an F/A-18C – An application of the multidisciplinary SimServer-system*, 23rd Applied Aerodynamics, AIAA 2005-5222, 2005.

[48] Bäck, T., Hammel, U, Schwefel, H.P.: *Evolutionary computation: Comments on the history and current state*, IEEE Transactions on Evolutionary Computation, 1(1):317, 1997.