THE DLR TAU-CODE: RECENT APPLICATIONS IN RESEARCH AND INDUSTRY

Dieter Schwamborn*, Thomas Gerhold[†] and Ralf Heinrich[‡]

* DLR, Institute for Aerodynamics and Flow Technology, Bunsenstr. 10, 37073 Göttingen, Germany e-mail: <u>Dieter.Schwamborn@dlr.de</u> Web page: http://www.dlr.de/as

[†] DLR, Institute for Aerodynamics and Flow Technology, Bunsenstr. 10, 37073 Göttingen, Germany e-mail: <u>Thomas.Gerhold@dlr.de</u> Web page: <u>http://www.dlr.de/as</u>

[‡]DLR, Institute for Aerodynamics and Flow Technology, Lilienthalplatz 7, 38108 Braunschweig, Germany e-mail: <u>Ralf.Heinrich@dlr.de</u> Web page: <u>http://www.dlr.de/as</u>

Key words: Computational Fluid Dynamics, unstructured hybrid flow solver, complex configurations

Abstract. This paper gives an overview of the TAU-Code, DLR's system for complex flow simulations on unstructured hybrid grids. Starting from a short description of the system and its components, e.g. mesh adaptation or grid deformation, its basic capabilities are discussed. Thereafter recent and on-going developments are presented, such as the adjoint approach or a pilot installation exploiting structured grids. The remainder of the paper discusses a number of recent applications of varying complexity, both from DLR and aircraft industry, such as simulation of half-model effects in wind tunnels, effects of trailing edge devices on full aircraft configuration, flow about a helicopter fuselage and generic Arianne-type launcher, full aircraft simulations such as A380-type airplane with undercarriage or complete EF2000.

1 INTRODUCTION

To date, Computational Fluid Dynamics (CFD) has reached wide acceptance as a mature tool complementary to wind tunnel and flight tests. With increased reliability and quality it has found entrance into the engineering offices of the European aircraft industry¹. At the same time, the numerical methods have still to be improved to meet the constantly rising requirements with respect to accuracy and efficiency and to reduce the response time for complex simulations, even if the relevant geometries and underlying physical flow models become increasingly complex. While it was sufficient in many cases in the past to obtain correct tendencies and deltas for different modifications of a geometry the major challenge for CFD is now to deliver highly accurate results and correct trends for new configurations in

order to become a reliable design tool in the process chains of the aeronautical industry.

Accordingly, future aircraft development will more heavily rely on (multi-disciplinary) simulation in the design phase, as recently announced by Airbus in its credo: "More Simulation, Less Testing". For the future Airbus simulation environment, currently being built up, it has been decided to employ the DLR TAU-Code code as the CFD-tool for complex configurations simulated with hybrid unstructured grids.

The development of TAU - started more than a decade ago at DLR in Göttingen^{2,3} - has taken place mostly in a number of internal projects, but also in national projects, like MEGAFLOW^{4,5,6}, and in several European projects, like FASTFLO, TAURUS, FLOMANIA and DESider. Although the main development of TAU takes still place in the Numerical Methods Department of the DLR Institute of Aerodynamics and Flow Technologies located in Braunschweig and Göttingen to date, the current status of TAU is to a great deal owed to the numerous supporters not only inside the institute and DLR but also at universities and especially in the aerospace industry which contribute in many ways to the development of new capabilities and to the validation of the TAU system.

Already in an early stage of the TAU development it was decided that the available resources were not sufficient to invest additionally in unstructured grid generation. As on the other hand fast, efficient and robust grid generation, especially for complex configurations, is an important prerequisite for efficient flow simulation a strategic cooperation has been established with the company CentaurSoft⁷, which provides the hybrid grid generation package CENTAUR. This software provides an interactive interface which allows reading in CAD data of the geometry under consideration, performing some CAD cleaning if necessary and setting up the grid generation process. Thereafter the surface and volume grid are generated automatically, where quasi-structured layers of prismatic cells around the geometric surface can be employed to ensure high resolution of boundary layers for viscous calculations. Furthermore, TAU has recently received an interface to the unstructured SOLAR mesh generator⁸.

This paper will give an overview on the TAU-Code, its current features and recent applications. In the following chapters we present first a brief description of the recent status of the TAU system and its features, before we demonstrate its capabilities on the basis of different applications provided by colleagues from both DLR and aeronautical industries.

2 THE DLR TAU-CODE

While TAU does not include grid generation, it comprises modules for grid modification, namely the adaptation and the deformation module (see below). TAU can be used with both (block-) structured and hybrid unstructured grids composed of hexahedrons, prisms, tetrahedrons and pyramids. The first two element types are usually used in semi-structured layers above surfaces allowing for a better resolution of boundary layers. Tetrahedrons are used to fill the computational domain in a flexible way, allowing for local refinement without hanging nodes while the pyramids are needed for the transition between elements with quadrilateral faces and elements with triangular faces.

The DLR TAU code is actually not one code but a modern software system for the

prediction of viscous and inviscid flows about complex geometries from the low subsonic to the hypersonic flow regime, employing hybrid unstructured grids. The system, in the following just called TAU, is composed of a number of modules and libraries to allow easier development, maintenance and reuse of the code or parts of it. The different modules of TAU, briefly described in the following, can both be used as stand-alone tools with corresponding file I/O or within a Python scripting framework which allows also for inter-module communication without file-I/O, i.e. using common memory allocation.

One of the important features of the TAU-Code is its high efficiency on parallel computers^{9,10,11,12} and its optimization for cache processors through specific edge colouring procedures. Parallelization is based on domain decomposition and the message passing concept using MPI. All modules of the system described in this chapter are capable of running in parallel, while some of the new developments described in chapter 3 are not yet (fully) parallelized unless otherwise stated

While TAU is mainly used for complex aircraft-type configurations (including coupling to structure^{13,14} and flight mechanics codes^{14,15}) there exist also extensions which allow the simulation of re-entry flows, i.e. real gas effects of oxygen and nitrogen can be taken into account, as described at the end of this chapter.

2.1 Grid partitioning

For parallel computations the grids are partitioned in the requested number of domains at the start of the simulation. Therefore a simple bisection algorithm is employed. The load balancing is performed on edge- and point weights which are adjusted to the needs of the solver, which is the most time consuming part of the simulation system. During the migration of the grid domains onto the different processes the communication tables are stored in the grid partition, which contains the necessary information for updating data on points in the overlap region between a domain and its neighbours. For adapted grids the grid hierarchy, which contains the information for grid de-refinement, is distributed over the domains as well.

After the grid partitioning, all other modules of TAU, which are described below, compute the requested data for a single domain per process. Grid re-partitioning is performed either if the grid was locally (de-)refinement in an adaptation or if the number of domains is changed, when a simulation is restarted on a different number of CPU's.

2.2 Pre-processing

The pre-processing needs to be employed once for a given primary grid. It computes the dual grid composed of general control volumes from the primary elements^[2]. They are stored in an edge based data structure, which makes the solver independent of the element types of the primary grid. All metrics are given by normal vectors, representing size and orientation of the faces, the geometric coordinates of the grid nodes and the volumes of the auxiliary cells (i.e. dual cells). The connectivity of the grid is given by linking the two nodes on both sides of each face to the corresponding edge from the primary grid elements. In order to enable the use of a multi-grid technique the agglomeration approach^[16] is employed to obtain coarse grids by fusing fine grid control volumes together. As the coarse grids employ the same type of metric

description as the fine dual grids, the solution on coarse grids can be computed with the same approach as on the finest grid. The transfer operators needed for the communication between the different grids are obtained directly during the agglomeration process.

In order to optimize the solver efficiency the edges of the dual grid are sorted such that cache-loads are minimised in the flux-computation part of the solver. The point indices are reordered to optimise memory and cache-line accesses as far as possible. This optimisation reduces the solver runtime to less than half for standard PC-architectures.

For the use in turbulence models wall distances are computed for each grid point and regions of laminar flow are flagged depending on user input or on the result of a transition prediction method.

2.3 Solver

The standard solver module uses an edge-based dual-cell approach, i.e. a vertex-centred scheme, where inviscid terms are computed employing either a second-order central scheme or a variety of upwind schemes using linear reconstruction (of the left and right states) for second-order accuracy. Viscous terms are computed with a second-order central scheme. Scalar or matrix artificial dissipation may be chosen by the user. Low Mach number preconditioning has also been implemented extending the use of the code into the incompressible flow regime.

Time integration was for a long time relying on various explicit Runge-Kutta schemes only, with additional convergence acceleration by a multi-grid algorithm based on agglomerated coarse grids generated by the pre-processing as indicated above^[2].



Fig. 1 Convergence behaviour of the hybrid TAU-Code for calculations of viscous flow around a delta wing at M=0.5, α =9°. Comparison shows baseline Runge-Kutta scheme (RK) and implicit LU-SGS scheme.

As the explicit approach leads to severe restrictions of the CFL number which in turn often resulted in slow convergence, especially in case of large scale applications an implicit approximate factorization scheme has recently been implemented¹⁷, in order to improve the performance and robustness of the solver. The LU-SGS (Lower-Upper Symmetric Gauss-

Seidel) scheme has been selected because 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. 1, where two convergence histories for viscous calculations on a delta wing are shown. The calculations were performed with multi-grid on 16 processors of a Linux cluster. The figure shows the residual and the rolling moment against iteration count. 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. This results in a reduction of the overall calculation time by a factor of 2.5.

For time accurate computations the dual time stepping approach of the Jameson is employed. As the solver respects also the geometric conservation law both grid deformation as well as bodies in arbitrary motion can be simulated.



2.4 Grid Adaptation

Fig. 2 Dynamic mesh refinement and de-refinement for the flow in a shock-tube 50, 70 and 110 µs after breaking of the second diaphragm. (left: computed Schlieren-pictures, right: grid development)

In order to efficiently resolve detailed flow features, a grid adaptation algorithm for hybrid meshes based on local grid refinement and wall-normal mesh movement in semi-structured near-wall layers was implemented². This algorithm has been extended to allow also for de-

refinement of earlier refined elements thus enabling the code to be used for unsteady timeaccurate adaptation in unsteady flows¹⁸. Fig. 2 gives an example of the de-/refinement process for the flow in a shock tube: After a second diaphragm to the right of the depicted area is broken, a complex interaction between shock waves and expansions begins. The figures display the density gradients resulting from the simulation as well as the grid which is automatically adapted to those gradients in each time step. This local refinement approach greatly reduces the number of grid points needed for the total simulation compared to a simulation on globally refined grids. Thus, given a limited computer memory it allows better resolution at the cost of additional CPU time (usually below 20 %) for adaptation per time step. Compared to the same resolution on globally refined grids this reduces the CPU and memory requirements considerably.

2.5 Grid Deformation

A grid deformation tool is used to account for moderate changes of the geometry, defined e.g. by an optimization technique during shape design or by the structural response of the geometry on aerodynamic loads in a coupled simulation, e.g.^{13,14}. An algebraic method has been developed in order to avoid time consuming iterative solutions of equations based e.g. on linear elasticity or spring analogy. The displacements which are the input for the deformation tool and the rotation of surface points are transported into the interior of the grid by an advancing front technique. Depending on the ratio between the local point displacement and the cell size the displacement is reduced by some fraction in each step of the front. This procedure ends, when no more grid points are moved during a sweep. In parallel computations, due to displacements coming from neighbouring domains the sweeps are continued until the grid does not change any more. Since a single sweep requires almost negligible effort, this is not a significant drawback of the parallel mode where usually an order of 10 to 20 sweeps is needed.

This algebraic method is robust enough for small and sometimes also for medium displacements and can handle e.g. wing tip deflections of one or several chord length. It has been observed that the limit, i.e. the deflection which results in a collapse of a first cell, can be extended considerably if it is accepted that a few more cells collapse. Repairing these cells with another algorithm, in a second stage of the deformation increases the robustness considerably and allows for large grid deformations. In this algorithm, each region in the grid containing collapsed cells is marked such that it is bounded by valid cells only. With the shape of this boundary in both the deformed and the non-deformed grid a transformation can be computed applying the volume spline technique, which allows rebuilding the collapsed cells as images of the original ones. As long as these regions remain small the additional computational costs for the local volume spline is low.

Fig. 3 indicates that this robust approach allows going beyond realistic deformations. It shows the maximum possible wing tip deflections for a hybrid grid composed of $2.5*10^6$ points. CPU time requirements on one single Opteron CPU is less than 2 minutes for small deflections (of about a chord length) and less than 10 minutes for the maximum wing tip deflection.



Fig. 3 Maximum wing tip deflections and details of the deformed hybrid grid for the DLR F6 configuration (available from the second DPWS)

2.6 CHIMERA technique

As the Chimera technique has been recognized as an important feature to efficiently simulate manoeuvring aircraft, it has been also integrated into the TAU-Code¹⁹. In the context of hybrid meshes the overlapping grid technique allows an efficient handling of complex configurations with movable control surfaces. For the point update on chimera boundaries linear interpolation based on a finite element approach is used in case of tetrahedral mesh elements. For other types of elements (prisms, hexahedrons, pyramids) either linear interpolation is performed by splitting the elements into tetrahedrons or non-linear interpolation for the different element types is used. The search for cells which are used for interpolation of the Chimera technique can handle both steady and unsteady simulations for inviscid and viscous flows with multiple moving bodies and is also available in parallel mode. Applications of this technique in TAU can be found e.g. in ^{5,6,14}.

2.6 Transition and Turbulence modelling

The turbulence models implemented within the TAU code include linear as well as nonlinear eddy viscosity models spanning both one- and two-equation model families.

The standard turbulence model in TAU is the Spalart-Allmaras model with Edwards modification, yielding highly satisfactory results for a wide range of applications while being numerically robust. The k- ω model provides the basis for the two equation models, where the one mostly used is probably the Menter SST model. Besides this, a number of different k- ω models, like Wilcox and Kok-TNT, are available. Also nonlinear explicit algebraic Reynolds stress models (EARSM) and the linearized LEA model²⁰ have been integrated. The implementation of RSM models is ongoing work. A number of rotation corrections for vortex dominated flows are available for the different models.

For a long time only the low Reynolds formulation has been available in TAU for accuracy reasons, but recently model specific "universal" wall-function have been introduced to achieve a higher efficiency of the solver, especially for use in design or optimisation as well as to allow for a "first quick look" on new configurations. Although tests are till on-going, this very promising approach²¹ seems to be able to deliver nearly as good results as the low Reynolds approach for pressure and skin friction distributions over a wide range of y^+ values for the first cell height at the wall, while saving up to 75 percent of computation time and 40 percent of memory.

Finally, there are options to perform Detached Eddy Simulations (DES)²² based on the Spalart-Allmaras or the Menter SST model or the so-called Extra-Large Eddy Simulation (XLES)²³. Since the DES method is a development strategy that can be applied, in principal, to any eddy viscosity turbulence model, the original implementation within the TAU code required only the calculation of additional terms and a suitable switch to activate the DES model within the calculation of the source term of the turbulence model. In computational terms, the overhead for the solution of a time step using Detached Eddy Simulation is negligible compared to URANS in three-dimensional cases. However, the true cost arises from the need to sufficiently resolve the temporal scales such that unsteadiness in a solution can grow in a physical way.

In order to allow for modelling of transitional flow the turbulent production terms are suppressed in regions which are flagged in the grid as being of laminar flow type. Flagging of laminar regions can be done in the pre-processing by the definition of polygon-lines which encircle the laminar region on the surface grid and the definition of a maximum height over the surface. The polygon lines for laminar regions can be defined by the user for simulations with fixed transition to turbulent flow or can be computed by a transition prediction module, which is ongoing development and is thus described in more detail in the following chapter.

2.7 TAU version for hypersonic and reacting flows

To extend the range of applicability of TAU to hypersonic²⁴ or high enthalpy flows additional modifications and extensions have been introduced into the code already a while ago, enabling e.g. simulations of re-entry vehicles including chemical reactions of air as a five component gas. These modifications range from stabilization of the solver for high Mach numbers over additions for thermo-chemical equilibrium flows to the consideration of non-equilibrium gases. Due to the latter additional conservation equations for the partial densities and the vibrations energies of the species are introduced in the code and to close the system, models for the state of species as well as fits for their viscosity (Blottner) and the resulting heat conductivity (modified Eucken correction) are taken into account. Furthermore mixture rules (Wilke and Herning/Zipperer), diffusion (following Ficks law) and detailed chemistry based on the Arrhenius-Ansatz with thermal coupling after Park are implemented together with thermal relaxation after Landau-Teller to allow for full thermo-chemical non-equilibrium simulations.

With respect to boundary conditions walls with full, finite or non catalytic surfaces can be considered as well as radiation-adiabatic walls or effusion-cooled walls (porous walls).

Currently data bases are under construction to consider

- air plasma with 11 components (including electrical conductivity for MHD effects),
- a CO₂ atmosphere (Mars) as well as
- H₂-O₂ combustion including probability density functions (PDF) for coupling of chemistry and turbulence.

3 RECENT AND ON-GOING DEVELOPMENTS

3.1 TAUijk – Structured algorithms in an unstructured code using a cell-centred metric

In the CFD community structured and unstructured codes are in daily use and under further development. As known unstructured codes like TAU enable a higher geometric complexity in the simulation, whereas structured codes like FLOWer⁵ and $elSA^{25}$ are usually characterized by better performance properties. From developer and user side it would be desirable to have one code being able to handle structured and unstructured meshes as well as grids containing both types of regions. At DLR a first prototype of a mixed structured / unstructured code named TAUijk²⁶ has been developed, in order to explore the potential of such a combined procedure. The desired features of TAUijk are:

- equal performance and accuracy as the existing TAU-code when using purely unstructured meshes
- equal or better performance and accuracy as the existing FLOWer-code when using a purely block-structured mesh
- simultaneous usage of structured and unstructured technologies in case of meshes including both block-structured and unstructured regions.

Using the prototype, techniques have been developed enabling a simple inclusion of structured loops over elements or faces (using i, j, k). Useful "structured" techniques helping to improve the performance and robustness are for example implicit smoothing techniques and the structured way of coarsening meshes for the multi-grid procedure. Both techniques have been implemented in TAUijk. Fig. 4 shows first promising results using TAUijk in comparison to the block-structured FLOWer code. A purely structured mesh around a NACA0012 airfoil has been generated. TAU has been used in "unstructured" mode and "structured" mode to perform an inviscid computation. "Unstructured" mode means employing exactly the same routines as currently used in the standard TAU, i.e. Runge-Kutta time stepping in combination with explicit residual smoothing and multigrid. "Structured" mode means to use an implicit smoothing and cell agglomeration like in FLOWer in combination with an increased CFL number (enabled by the implicit smoothing). The dashed line shows the L₂ norm of the density residual using TAUijk in unstructured mode. The convergence properties are strongly improved using TAUijk in structured mode (solid line) and are more or less the same as for FLOWer (dotted line). The computational costs for each multigrid cycle are approximately the same for both modes. So not only the convergence rates are improved but also the performance with respect to CPU time. This can additionally be improved by using a mixed unstructured / structured mesh as shown in 26 . Due to the success of the prototype it has been decided to transfer the techniques developed into the TAU-code.



Fig. 4 Comparison of convergence history of TAUijk in unstructured and structured mode and blockstructured FLOWer code for the inviscid flow about a NACA0012 airfoil

It should be noted that TAUijk is based on a cell-centred (cc) metric²⁷, i.e. the cells of the primary grid itself are used as the control volumes for several reasons. One reason is the simplified treatment of sharp edges compared to the dual mesh approach, avoiding any special treatment. Additionally, no "half-cells" at boundaries have to be handled. Furthermore it has been shown that for a given hybrid mesh a cc metric can help to improve the accuracy properties of TAU in the framework of upwind schemes. Therefore the cc metric will also be included in TAU before transferring the techniques from TAUijk to TAU.

A detailed comparison of the dual mesh approach and the cc-metric is ongoing work.

3.2 The adjoint-solver

Numerical shape optimization will play a critical strategic role in future aircraft design. It offers the possibility of designing and improving aircraft components with respect to geometrical and physical constraints. However the extremely high computational costs of straightforward methodologies currently in use prohibit the application of numerical optimization for industrially relevant problems. Optimization methods based on calculation of the derivatives of the cost function with respect to the design variables suffer from high computational costs if many design variables are used. However, these gradients can be efficiently obtained by solution of the adjoint equations.

Therefore a discrete adjoint of the Navier-Stokes equations has been developed in TAU^{28,29} within the project MEGADESIGN³⁰. The method consists of the explicit construction of the exact Jacobians of the spatial discretization with respect to the unknown variables allowing the adjoint equations to be formulated and solved. It is used for aerodynamic shape optimization in viscous turbulent flow and has been validated using realistic 2D cases by comparing the gradients of e.g. drag, lift and pitching moment with the gradients computed by approximate finite difference methods. The capability of the method has been demonstrated, e.g. for the optimization of the flap settings of a multi-element airfoil in take-off configuration (Ma=0.1715, Re=14.70x10⁶)²⁹, see Fig. 5. The initial geometry and aerodynamic conditions

of the configuration were defined in the European project EUROLIFT II³¹. In this case the gap and the deflection angle of the flap are the design variables, while the goal function is the drag (reduction) at constant lift. The lift is kept constant by changing the angle of incidence using the target lift functionality of TAU. To solve this problem the conjugate gradient optimization strategy is used. The evolution of drag, lift and angle of incidence with the optimization step is given in the Fig. 5 on the left. The geometries and pressure distributions at the initial and optimized configurations are seen on the right. The optimization required 6 steps to converge and resulted in a 67 counts lower drag at equal lift.



Fig. 5 History of the optimisation process and pressure distributions for high-lift flap design

3.3 Transition prediction

Transition prediction is a new module that has recently been coupled to the TAU code, taking advantage of the general functionality to prescribe transition lines which are known a priori almost arbitrarily on any surface of complex geometries. The transition prediction module comprises a number of different transition prediction approaches and can be applied to different types of configurations. Besides two e^N-database methods for Tollmien-Schlichting and cross flow instabilities, a fully automated linear stability code is available³². The parameters of the laminar boundary layers which are determined along inviscid streamlines are either computed by a laminar boundary-layer code for swept tapered wings based on the approximations for external conical flows or they are directly evaluated inside the TAU code using the solution from the RANS grid. Thus, the coupled system can be applied to complex configurations in an industrial context making use of the short computation times of the boundary-layer code, but allows alternatively to compute the necessary boundary-layer parameters from the RANS solution (provided a fine enough grid is available), e.g. when transition inside a laminar separation bubble has to be detected.

The coupled system can be applied to arbitrary two-dimensional and three-dimensional multi-element aircraft configurations including one- or multi-element wings, tail-planes, fuselages, and nacelles. The development has been carried out in close co-operation with the Technical University Braunschweig³³ within the German research project MEGADESIGN.

4 APPLICATIONS

4.1 Application and validation at DLR

The range of applications of the TAU Code at DLR is quite widespread according to the broad range of aerodynamic and aero-thermodynamic activities. This range extends from simple 2D airfoil flows at low speed to the computation of 3D space vehicles and rockets and hypersonic flows. In between lays a variety of applications covering fighter and transport aircraft as well as helicopter. Simulations for transport aircraft are made for cruise, high lift as well as for off-design conditions. They are carried out to predict aerodynamic coefficients and loads, flight mechanics¹⁵ and structural response¹⁴, to study the effectiveness of control devices or flow control devices, for engine airframe integration³⁴, for high-lift optimization and for validation versus experimental data available at DLR^{35,36}. This broad range of applications is very important for the TAU development because all weaknesses and strengths of the simulation software turn up quite rapidly for each new release making the code reliable for the industrial use.



Fig. 6 RANS simulation of model in the wind tunnel; upper left: set-up; upper right: vortical structures in the simulation; lower left: comparison of wall streamlines and oil flow; lower right: lift polar from experiment and simulation

A few example of TAU usage within DLR are presented in the following starting with some complex steady applications followed by recent unsteady ones.

The first example presents preliminary results from the DLR project ForMEx. One aim of this project is to quantify differences between wind tunnel and free flight data. The geometry under consideration is the DLR-ALVAST half-model with high-lift devices mounted on the wind-tunnel floor employing a peniche. The CFD simulation at Ma = 0.18 and Re = 2 million includes the wind tunnel walls of settling chamber, nozzle and test section as depicted in Fig. 6. To allow the use of identical meshes for different angles of attack the Chimera technique is employed combining a Chimera grid for the aircraft model with a background grid discretising the wind tunnel geometry. In the simulation all walls of the wind tunnel and the aircraft model are considered as viscous walls on which 30 layers of prismatic elements are used in order to resolve the turbulent boundary layers everywhere. The remaining part of the computational domain is discretized with tetrahedral elements resulting in a grid containing overall about 20 million grid points. Computations for complete angle-of-attack polars including maximum lift positions are scheduled for about 16 variations of the geometry of the wing root (to account for the effect of slat horns, fillets, etc.) and different heights of the peniche with and without gap. Fig. 6 shows furthermore areas of vortical flow such as the horseshoe vortex around the peniche, the vortices induced by flap edges and wing tip as well as structures resulting from flow through the gap between model and peniche. As can be seen from the polar in Fig. 6 computational results agree well with the data from the wind tunnel experiment without the need of wind tunnel corrections.

The next example stems from the European project AWIATOR and deals with the simulation of trailing edge devices for a transport aircraft model. Fig. 7 depicts the CAD geometry of the wing-body model with two pylons and nacelles and split-flaps in some areas along the wing trailing edge. Four grids were generated for this geometry with split flaps closed and at 7.5° for two different Re-numbers of 4.3 and 45 million resulting in grids of about six million and nine million grid points, respectively, for the cases with and without trailing-edge devices. Simulations were performed for a cruise Mach number of Ma = 0.82 and lift coefficients of c_L = .40, .50 and .60 utilising the constant lift iteration option of TAU.



Fig. 7 Model from the AWIATOR project: CAD geometry of the model with pylons and nacelles and splitflaps at the wing trailing edge (left); detail of the wing trailing edge area with split flaps (right)



Fig. 8 Lift and drag polar for the AWIATOR configuration (upper left/right) and comparison of experimental and computed pressure distribution at 4 wing stations for the case with deflected split flaps

Fig. 8 shows a comparison of experiments and simulation for the lower Re number both for the case with and without deflected split flaps. From the lift polar it is obvious that the simulation predicts a lift which is slightly smaller than the experimental one at the same angle of attack. At the same time the predicted drag (at the same lift) is lower than in the experiment as seen from the drag polar in Fig. 8. Both effects can at least partly be attributed to the fact that the experiments used a model with flap-track fairings not present in the simulation. This assumption is supported by the computed pressure distributions in Fig. 8, which agree otherwise quite well with the experimental data, except for the outboard stations where the experiments indicate a more upstream shock which is, however, due to some slight deformation of the outer wing in the experiments.

Most importantly, however, the simulations predict the correct shift both in the lift and drag polar due to the effect of the split flat deflection; even indicating the correct lift value for the cross-over of the drag polars with and without deflection.

As an example of the numerical simulation in combination with transition prediction at a low Reynolds- and Mach-number, a glider configuration³⁷ is considered next. The aim is to calculate the speed-polar of a "Standard Cirrus" for the following free stream conditions: $H_{\text{flight}} = 1000 \text{ m}, T_{\infty} = 281.65 \text{ K}, p_{\infty} = 89875 \text{ Pa} \text{ and } \rho_{\infty} = 1.112 \text{ kg/m}^3$.



Fig. 9 Simulation of a glider (Standard Cirrus); left: Surface mesh; centre: Streamlines on the wing with transition lines (red) on upper / lower surface for a speed of V=100km/h (upper), V=220km/h (lower); right: Speed polar (sink speed vs. Air speed) with measurements (green), fully turbulent simulation (blue) and simulation of transition (red)

Fig. 9 shows on the left the surface mesh of the configuration. The hybrid mesh includes 5.5 million nodes in total. In the middle part of the figure streamlines and transition lines on the upper wing surface are depicted resulting from simulations at different velocities. At V_{∞} = 100 km/h a laminar separation bubble is found on the upper side over most of the wing. The transition moves downstream at higher speed due to the reduced angle of attack leading to a reduced boundary layer load and a disappearing laminar separation bubble. Below $V_{\infty} = 90$ km/h flow separation takes place on the upper wing in the simulation resulting in a drop in the speed polar (right part of Fig. 9), which is not found in the experiment and probably due to an over-prediction of separation by the turbulence model (Menter SST). Note, that since horizontal and vertical tail are not simulated viscous-, profile- and induced drag of the tail is estimated and added to the simulated speed polars. At higher speeds the fully turbulent simulation has an increased drag and sink-speed, respectively, compared with the flight measurements, while the polar with transition prediction indicates a reduced sink velocity compared to the measurement. Possible reasons are a too small separation bubble drag due to a too coarse grid in that area and the missing induction of the horizontal tail on the wing, which would lead to an increased induced drag of the wing.

The next application has been performed to validate TAU for the isolated fuselage of the EC 145 helicopter³⁸. For this type of flow it is important that the flow separation in the region

of the back door of the fuselage is accurately predicted to obtain accurate drag values as well as a correct interaction of the vortices shed from the back door region with the helicopter tail. The left part of Fig. 10 shows the computed surface pressure distribution with skin friction lines depicting the flow separation on the back of the fuselage of an EC 145 helicopter. The middle part of the figure shows the grid in the symmetry plane which is refined in the area of interest. On the right the comparison between TAU and the structured FLOWer code is plotted against experimental data, which are from a slightly different geometrical shape. Good agreement between TAU and the reference results from FLOWer can be seen when the same turbulence model (i.e. Menter-SST) is employed.



Fig. 10 Simulation of the flow about a EC 145: pressure contour and separation pattern on the backdoor area (left); unstructured grid pre-refined in a box about the rear part of the helicopter (middle); predicted drag polars (see text) in comparison with experimental data from a slightly different configuration

The following examples deal with unsteady simulations, the first one being the flow about an oscillating OA209 airfoil used for helicopter blades. A URANS simulation was performed at a Mach number of 0.3085, Reynolds number of 1.15 million for an angle of attack oscillation of $\alpha = 9.8^{\circ} + 9.1^{\circ} \sin(kt)$ with a reduced frequency of k=0.05 employing the SAE model. The resulting force coefficients are depicted in Fig. 11 (left) in comparison with experimental data from 128 oscillation cycles indicating that URANS is capable of delivering reasonable results which are within the variation found in the measurements. The simulation results in the form of vorticity contours and streamlines are shown at the right of Fig. 11. The upper picture represents the flow at $\alpha = 17.5^{\circ}$ in the up-stroke (maximum lift) while the lower one depicts the situation at $\alpha = 17.2^{\circ}$ in the down-stroke where a number of vortices can be seen whose interaction leads to the sinusoidal oscillation of the force coefficients. Compared to the experiment maximum lift as well as drag and moment is somewhat overestimated by the URANS simulation, which is not able to resolve the small-scale statistical variation in the flow as one could expect for example from a DES.

URANS is also employed in next application which deals with fluid-structure coupling (see also¹⁴) in the time domain to estimate the flutter boundary. Normally much simpler methods such as doublet lattice (DLM) and transonic doublet lattice methods (TDLM) are used for this purpose. In comparison with measurements these methods, especially the DLM, tend to fail in the regime of the transonic dip, due to the strong nonlinear effects in this flow

regime. And they are not able to simulate so-called Limit Cycle Oscillations (LCO), i.e. small amplitude oscillations of the coupled system which are not as dangerous as flutter as they do not lead to wing destruction except maybe through fatigue after a very long time.



Fig. 11 Dynamic Stall of a Helicopter blade profile OA209: lift, drag and momentum coefficient over angle of attack (left); computed streamlines and vorticity at $\alpha = 17.5^{\circ}$ in the up-stroke and 17.2° in the down-stroke (right)



Fig. 12 Structural model and CFD surface grid of AMP wing (upper left) with results of numerical estimation of the transonic dip (right): the URANS simulations (coloured dots) allow a good estimate of flutter stability

In the simulation presented in Fig. 12 the AMP wing³⁹ is used to compare results from

measurement as well as DLM and TDLM with TAU coupled to NASTRAN to evaluate its capability to estimate the flutter boundary in the region of the transonic dip. The NASTRAN FE model employs 75 nodes resulting in 225 translational degrees of freedom, while the CFD grid had one million nodes with 27.600 on the surface. The available results from the coupled simulations are in good agreement with the measured data and indicate additionally that for a Mach number of 0.82 and a critical total pressure of 1.1 bar a case of LCO is found (pink dot in Fig. 12). Of course, it has to be noted that coupled URANS or even Euler solutions are much more expensive in terms of computer resources then linearized methods like TDLM working in the frequency domain.



Fig. 13 DES simulation of the flow about the nozzle area of an Ariane-type launcher with boosters

As an example for the application of DES to real world situations we present a simulation of the flow about a generic Ariane-5-type launcher. This DES is made in the framework of a study which aims finally at fluid-structure coupling to examine and understand effects of buffeting in the area of the nozzle of the rocket and the boosters. Fig. 13 presents a view of the structured grid in the symmetry plane of an Ariane configuration and results from a steady RANS simulation (on the upper left) as well as views of the averaged flow between the nozzles both from a DES simulation and a URANS in this area (upper right). To reduce the effort for the unsteady simulation only a box about the lower part of the complete configuration was used, prescribing the inflow conditions to the box from the steady RANS solution. The lower part of Fig. 13 depicts the comparison between the simulation and an experiment. As the experimental set-up included also a helium tank on one side of the configuration not taken into account in the simulation the corresponding data are not symmetric in the upper part of the nozzle. Interestingly, the agreement between the RMS-values of the pressure from the experiment and the simulation is better for the side of the configuration which is closer to the helium tank.

4.2 Applications by the aeronautical industry

TAU is in routinely use in the German aeronautical industry and it is started to be used at all other Airbus sites. Two- and three-dimensional configurations with reduced complexity are daily applications. Full configurations of higher complexity are more and more often subject of numerical simulations of turbulent flows. Because of the level of geometrical details considered in the simulations the grid sizes in use are increasing every year. As a consequence the number of processes required for the large scale computations increases also, such that massive parallel computations become the usual case. Some examples for complex applications in industry are described below.

Fig. 14 shows some results from a numerical investigation for lateral stability of a transport aircraft which has been performed by Airbus Germany. At the left of the figure the grid about the configuration is shown which is refined in the tail region of the aircraft to study the flow near the tail, which is induced by the sponsons. This investigation has been carried out for different shapes of the sponsons for small yaw angles of the flow. The geometry includes the strakes at the tail of the aircraft. In comparison to experimental data a good match of the increments in drag due to the different sponson shapes was found. The right part of the figure showing a comparison between computed skin friction lines and an oil flow picture from the wind tunnel indicates good qualitative agreement of the flow below the tail.



Pressure on the surface, vorticity in a cut through the grid and stream lines

Comparison of wind tunnel oil flow and CFD streamline picture on lower tail plane (view from below)



The next application shown in Fig. 15 has higher geometrical complexity. It is a full high lift configuration (without tail planes) of the A380 with landing gears, which has been used to simulate the ground effect. The results were compared with experimental data from the German-Dutch windtunnel. The experimental set-up is also shown in the figure. The simulation for the turbulent flow employing a two-equation $k-\omega$ -type model has been performed by Airbus Germany. The comparison has shown that the predicted results are quite accurate, even if the grid is quite coarse locally, which can be seen in the figure showing the

surface grid of the aircraft. This was necessary due all the geometrical details which are meshed in this Navier-Stokes grid leading finally to 48 million grid points. A series of large scale computations for different points of the landing phase was performed on a cluster machine, using 64 and more processes per computation in parallel. The good agreement between the CFD data and the experimental results lead to a wider confidence of aerodynamic engineers into CFD.



ressure distribution on numerical model with 1FS Si

Side view, stream tubes about wing and undercarriage

Fig. 15 Simulation of ground effect: Aircraft approaching ground at 6°, H/B=0.15

In the last two examples TAU is employed to simulate the flow about complex delta wing configurations which have been simulated by EADS-M. Fig. 16 gives an overview about the pressure and vortex distribution about a F16XL aircraft which is used as a test case in the RTO/AVT-113 Task Group. The simulation is for an angle of attack of 19.84°, a Mach number of 0.242 and a Re-number of 32 million. Apparently there is no indication of vortex burst at this angle of attack, neither in the qualitative view in the upper part of the figure nor in the chord-wise pressure distributions at the wing stations indicated by white lines on the aircraft. The results from TAU (blue lines in lower part of Fig. 16)) compare quite well with flight data from NASA-Langley (dots), except that the suction peak at the leading edge is a bit more pronounced in the simulation. Due to convergence problems at high angle of attack it was found necessary to perform time-accurate RANS calculations to arrive at converged steady solutions, making these simulations expensive.

Finally, we turn to a simulation of the EF2000, which has been simulated at EADS-M in

many different configurations ranging from the plain aircraft to a fully equipped one (see upper left of Fig. 17).



Fig. 16 F16XL test case (see text): simulated surface pressure (upper left) and total pressure in cuts through the vortical areas (upper right); pressure distributions in two chord-wise wing sections (indicated by white lines) from TAU (blue line), NASA flight tests and another code not discussed here.

EADS-M uses the TAU code mainly within its in-house simulation environment "SimServer" whose components are sketched in the lower left of Fig. 17. The "SimServer" is an efficient tool for (time-accurate) mono- and multidisciplinary simulations focusing on advanced non-linear methods such as Euler, RANS, DES and their close coupling to relevant disciplines like structural mechanics, flight mechanics and flight control systems. Here this tool was used to obtain aerodynamic longitudinal control characteristics for different EF2000 configurations employing TAU in RANS mode with the LEA turbulence model. As an example full polars for lift, drag and pitching moment are displayed on the right of Fig. 17 for one configuration together with the surface distributions of pressure and skin friction





Fig. 17 EADS-M EF2000 full configuration (upper left); structure of the EADS-M SimServer (lower left); aerodynamic longitudinal control characteristics (middle) and some surface data for a EF2000 configuration

5 CONCLUSIONS

We have tried to give a concise overview on the TAU system for flow simulation on unstructured grids and its capabilities. As the examples given have shown TAU is being used extensively in Germany both in research and in industry for solving complex aerodynamic problems with very encouraging and often very good results.

Despite the high level of numerical flow simulation established today there is still a demand for more accuracy and especially for faster response times. In addition to further improvements to satisfy this demand even for problems of high complexity, it is necessary to extend the system capabilities to user-friendly multidisciplinary optimisation as well as to more complex multi-disciplinary simulations. Work in these directions is under ways in running DLR and national project such as MEGADESIGN and MegaOpt as well as in a planned project on CFD for stability and control.

ACKNOWLEDGEMENT

The authors would like to thank their colleagues from the DLR Institute of Aerodynamics and Flow Technology and from the Institute of Aeroelasticity as well as from Airbus Germany and from EADS-M for providing their results for this paper. Unfortunately, we were not able to present all this material due to space limitations.

Furthermore we would like to acknowledge the support which the development of TAU has received over the time from numerous European (like FASTFLO, TAURUS, FLOMANIA, DESider, AWIATOR), national (MEGAFLOW, MEGADESIGN) and DLR projects (like AeroSUM, Amanda, MEGAFLOW II, SikMa, HighPerFlex, MegaOpt), which are too numerous to be all mentioned by name.

REFERENCES

- [1] J. B. Vos, A. W. Rizzi, D. Darracq, E. H. Hirschel, *Navier-Stokes Solvers in European Aircraft Industry, Progress in Aerospace Sciences*, Vol. 38, pp. 601-697 (2002).
- [2] T. Gerhold, O. Friedrich, J Evans, M. Galle, *Calculation of Complex Three-Dimensional Configurations Employing the DLR-TAU-Code*, AIAA-paper 97-0167, 1997
- [3] T. Gerhold, V. Hannemann, D. Schwamborn, On the Validation of the DLR-TAU Code, in W. Nitsche H.-J. Heinemann, R. Hilbig (Eds.), New Results in Numerical and Experimental Fluid Mechanics, Notes on Numerical Fluid Mechanics, 72, Vieweg, S. 426 - 433, ISBN 3-528-03122-0, 1999
- [4] N. Kroll and J.K. Fassbender (Eds.), *MEGAFLOW Numerical Flow Simulation for Aircraft Design*, Notes on Numerical Fluid Mechanics and Multidisciplinary Design, Volume 89, Springer (2005).
- [5] N. Kroll, C.-C. Rossow, D., Schwamborn, K. Becker, G. Heller, *MEGAFLOW A Numerical Flow Simulation Tool For Transport Aircraft Design*, ICAS Congress 2002, Toronto (can), 09.-13.09.2002, ICAS, CD-Rom, S. 1.105.1-1.105.20, (2002)
- [6] N. Kroll, C.-C. Rossow, D. Schwamborn, *The MEGAFLOW-Project Numerical Flow Simulation for Aircraft*. In A. DiBucchianico, R.M.M. Mattheij, M.A. Peletier (Eds.), *Progress in Industrial Mathematics* at ECMI 2004, Springer, New York, S. 3 33, 2005
 [7] Contempode CENTAUR http://www.soutewaseflow.com/
- [7] CentaurSoft, CENTAUR, http://www.centaursoft.com/
- [8] D. G. Martineau, S. Stokes, S. J. Munday, A. P. Jackson, B. J. Gribben, *Anisotropic Hybrid Mesh Generation for Industrial RANS Applications*, AIAA-paper 2006-534, AIAA Aerospace Conference, Reno, 2006
- [9] M.Galle. T. Gerhold, J. Evans, Parallel Computation of Turbulent Flows around Complex Geometries on Hybrid Grids with the DLR-TAU Code, A. Ecer, D.R. Emerson (eds.), Proc. 11th Parallel CFD Conf., 23-26 May 1999, Williamsburg, VA, North Holland, 1999.
- [10] N. Kroll, Th. Gerhold, S. Melber, R. Heinrich, Th. Schwarz, B. Schöning, Parallel Large Scale Computations for Aerodynamic Aircraft Design with the German CFD System MEGAFLOW. Parallel CFD 2001, Egmond aan Zee, The Netherlands, May 21-23, 2001
- [11] P. Aumann, H. Barnewitz, K. Becker, R. Heinrich, B. Roll, M. Galle, N. Kroll, T. Gerhold, D. Schwamborn, M. Franke, *MEGAFLOW: Parallel Complete Aircraft CFD*. Parallel Computing, Vol. 27, 415-440, 2001

- [12] T. Alrutz, Investigation of the parallel performance of the unstructured DLR-TAU-Code on distributed computing systems, to appear in Proceedings of Parallel CFD 2005, Elsevier, 2006
- [13] A. Beckert, T. Gerhold, G. Günther, D. Schwamborn, K. Weinman, Application of the DLR-TAU Code to Fluid-Structure Interaction on Hybrid Grids, Proceedings of ECCOMAS 2000, Barcelona, 11.-14. September 2000
- [14]G.Einarsson, A. Schütte, A. Raichle, B. Schöning, W. Mönnich, J. Neumann, J. Arnold, T. Alrutz, J. Heinecke, T. Forkert, H. Schumann, *Prediction of the Unsteady Behavior of Maneuvering Aircraft by CFD Aerodynamic, Flight-Mechanic and Aeroelastic Coupling.* RTO (eds.), AVT-Symposium, Budapest, 2005-04-26, 2005
- [15] A. Schütte, G. Einarsson, A. Madrane, B. Schöning, W. Mönnich, W.-R. Krüger, Numerical Simulation of Maneuvering Aircraft by CFD Aerodynamic and Flight-Mechanic Coupling, RTO-AVT Symposium 2002, Paris 22.-24.04.2002
- [16] V. Hannemann, *Structured multigrid agglomeration on a data structure for unstructured meshes*. International Journal for Numerical Methods in Fluids, 40, 361-368, 2002
- [17] R. Dwight, A Comparison of Implicit Algorithms for the Navier-Stokes Equations on Unstructured Grids, Proceedings of the ICCFD Conference, Toronto, Canada, 2004
- [18] T. Alrutz, Hybrid Grid Adaptation in TAU, , in: N. Kroll, J.K. Faßbender, (Eds.) MEGAFLOW - Numerical Flow Simulation for Aircraft Design, Notes on Numerical Fluid Mechanics and Multidisciplinary Design (NNFM), 89, Springer pp.163 - 178, ISBN 3-540-24383-6, 2005
- [19] A. Madrane, R. Heinrich, T. Gerhold, Implementation of the chimera method in the unstructured hybrid DLR finite volume TAU-Code, 6th Overset Composite Grid and Solution Technology Symposium, Ft. Walton Beach, Florida, USA, October 8-10, 2002, http://www.arl.hpc.mil/Overset2002/contents.html
- [20] T.Rung, H. Lübcke, M. Franke, L. Xue, F. Thiele, S. Fu, Assessment of Explicit Algebraic Stress Models in Transonic Flows, Proceedings of the 4th Symposium on Engineering Turbulence Modeling and Measurements, France, pp. 659-668 (1999)
- [21] T. Knopp, T. Alrutz, D. Schwamborn, *A grid and flow adaptive wall-function method fro RANS turbulence modelling*, accepted for Journal of Computational Physics, 2006
- [22] M. Shur, P. Spalart, M. Strelets, A. TRAVIN, *Detached Eddy simulation of an airfoil at high angle of attack*. In: Rodi, W. and Laurence, D. (ed.), Proc. 4Th Int. Sym. Eng. Turb. Modelling and Measurements, Elsevier, Amsterdam, pp 669-678, 1999.
- [23] J. C. Kok, H.S. Dol, B. Oskam, H.van der Ven, *Extra-Large Eddy Simulation of Massively Separated Flows*, AIAA-Paper, 2004-0264 (2004)
- [24] A.Mack, V.Hannemann, Validation of the Unstructured DLR TAU-Code for Hypersonic Flows, AIAA 2002-3111, 2002
- [25] M. Gazaix, A. Jolles, M. Lazareff, *The elsA Object-Oriented Computational tool for industrials applications*, 23rd Congress of ICAS, Toronto, Canada, September 8-13, 2002
- [26] R. Heinrich Implementation and Usage of Structured Algorithms within an Unstructured CFD-Code. STAB-Tagung 2004, Bremen, in "Notes on Numerical Fluid Mechanics and Multidisciplinary Design", Volume 92, Springer (2006)

- [27] M. Widhalm, C. C. Rossow, Improvement of upwind schemes with the Least Square method in the DLR TAU code 13. DGLR-Fach-Symposium AG STAB, Munich Nov. 2002, in New Results in Numerical and Experimental Fluid Mechanics IV, Springer,2004
- [28] R. P. Dwight, J. Brezillon, *Effect of Various Approximations of the Discrete Adjoint on Gradient-Based Optimisation*, 44th AIAA Aerospace Meeting, 9-12 Jan. 2006, Reno,2006
- [29] J. Brezillon, R. P. Dwight, 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, Munich, September 2005
- [30] N. Kroll, N. Gauger, J. Brezillon, K. Becker, V. Schulz, *Ongoing Activities in Shape Optimization Within The German Project MEGADESIGN*, in: CD of ECCOMAS 2004, Jyväskylä (FI), July 24-28, 2004
- [31]J. Wild, R. Mertins, D. Quagliarella, J. Brezillon, J. Quest, O. Amoignon, F. Moens, Applying numerical optimization to realistic high-lift design of transport aircraft- An overview of the aerodynamic design optimization investigation with the EUROLIFT II project, EUROGEN 2005 - Sixth Conference on Evolutionary and Deterministic Methods for Design, Optimization and Control with Applications to Industrial and Societal Problems, Munich, September 2005
- [32] A. Krumbein, Automatic Transition Prediction for High-Lift Systems Using a Hybrid Flow Solver, Journal of Aircraft, Vol. 42, No. 5, pp. 1150-1164, 2005
- [33] N. Krimmelbein, R. Radespiel, C. Nebel, *Numerical Aspects of Transition Prediction for Three-Dimensional Configurations*, AIAA-2005-4764, 2005
- [34]O. Brodersen, Drag Prediction of Engine-Airframe Interference Effects Using Unstructured Navier-Stokes Calculations. Journal of Aircraft, 39 (6), pp. 927 935, 2002
- [35]S. Melber-Wilkending, R. Rudnik, A. Ronzheimer, T. Schwarz, Verification of the MEGAFLOW-Software for High Lift Applications, in: N. Kroll, J.K. Faßbender (eds.) MEGAFLOW - Numerical Flow Simulation for Aircraft Design, Notes on Numerical Fluid Mechanics and Multidisciplinary Design (NNFM), 89, Springer pp.163 - 178, ISBN 3-540-24383-6, 2005
- [36] M. Rakowitz, B. Eisfeld, D. Schwamborn, M. Sutcliffe, *Structured and Unstructured Computations on the DLR-F4 Wing-Body Configuration*, Journal of Aircraft, 40 (2), pp. 256 264, 2003
- [37] S. Melber-Wilkending, G. Schrauf, M. Rakowitz, Aerodynamic Analysis of Flows with Low Mach- and Reynolds-Number under Consideration and Forecast of Transition on the Example of a Glider. STAB-Tagung 2004, Bremen, in "Notes on Numerical Fluid Mechanics and Multidisciplinary Design", Volume 92, Springer, 2006
- [38] A. D'Alascio, K. Pahlke, F. Le Chuiton, *Application of a structured and an unstructured CFD Method to the Fuselage Aerodynamics of the EC145 Helicopter*, in: CD, ECCOMAS 2004, Jyväskylä (fi), 24.-28.07.2004
- [39] R. Voß, Aeroelastic Simulations in Aircraft Design and Analysis using different Level CFD Methods, 50th Annual General Meeting & Conference, Canadian Aeronautics and Space Institute, Ottawa, Canada, 28-30 April 2003