ICAS-98-2,7,4

MEGAFLOW - A NUMERICAL FLOW SIMULATION SYSTEM

N. Kroll, C.C. Rossow
DLR, Institute of Design Aerodynamics, Braunschweig, Fed. Rep. of Germany

K. Becker
Daimler-Benz Aerospace Airbus GmbH, EFV, Bremen, Fed. Rep. of Germany

F. Thiele
Technical University of Berlin, Herrmann-Föttinger Institute, Berlin, Fed. Rep. of Germany

Abstract

Within the framework of the German aerospace research program, the CFD project MEGAFLOW was initiated. Its goal is the development and validation of a dependable and efficient numerical tool for the aerodynamic simulation of complete aircraft in cruise as well as in take-off and landing configurations. In order to meet the requirements for industrial implementation, a concentrated co-operation with the aircraft industry, DLR and several universities has been set up. The project started mid 1995 and finishes by the end of 1998. This paper gives an overview of the plan of action and presents the major achievements of the project. It is concluded with an outlook towards future developments.

1. Introduction

During the last decade, considerable progress has been made in the development and validation of numerical simulation tools for aerodynamic applications. As a result, next to the traditional tools such as wind tunnel and flight testing, Computational Fluid Dynamics (CFD) is widely accepted as an integral part of the aerodynamic design procedure for new aerospace vehicles.

However, even despite recent advances, CFD still suffers from deficiencies in accuracy, robustness and efficiency for complex applications, such as complete aircraft flow predictions. From the industry's point of view, numerical simulation tools are expected to deliver detailed viscous flow analysis for complete configurations at realistic Reynolds numbers, prediction of aerodynamic data with assured high accuracy, fast response time per flow case at acceptable total costs as well as aerodynamic optimization of the main aircraft components. In order to meet these requirements in earlier and earlier stages of the design process, considerable improvements of the CFD methods currently available in industry are necessary. One of the largest problems in computational fluid dynamics is the generation of appropriate computational meshes for complex configurations. Another problem for many industrial applications relates to the insufficient modelling of turbulent flows and the associated high numerical costs. State of the art in industry is the calculation of inviscid flows around complete aircraft configurations based on the solution of the Euler equations. The simulation of viscous flows by solving the Reynolds-averaged Navier-Stokes equations is limited to basic configurations such as isolated wings or wing-body configurations using a rather simplified modelling of turbulence. Consequently, current computational methods are not capable of a critical evaluation of the aerodynamic potential and performance limits of realistic aircraft designs with sufficient accuracy.

Within the framework of the German aerospace research program, the project MEGAFLOW was initiated under the leadership of DLR with the objective of enhancing the capabilities of current CFD methods and supporting the establishment of numerical simulation as an effective tool in the industrial design process. The goal of the project is to produce a dependable, efficient and quality controlled program system for the aerodynamic simulation of complete transport aircraft in cruise as well as take-off and landing configuration. The software system is based on block-structured grids and includes the grid generator MegaCads and the Reynolds-averaged Navier-Stokes solver FLOWer. It is designed to meet the requirements of industrial implementation. The performance of the complete system will be validated using industry relevant applications. The extensive development and validation work is carried out in concentrated cooperation with aircraft industry, DLR and universities. The MEGAFLOW project is partially funded by the German Federal Ministry for Education, Science, Research and Technology (BMBF). It started mid 1995 and finishes by the end of 1998. The total effort includes about 125 personnel years, 45% covered by DLR, 25% by industry and 30% by universities. The founding partners are Daimler-Benz Aerospace Airbus, DLR and the universities of Berlin, Braunschweig, Darmstadt and München. During the course of the project several organizations joined the consortium and support the goals and activities of the MEGAFLOW project. These are Daimler-Benz Aerospace AG, Technologiezentrum Nord, BMW Rolls Royce, GMD, the universities of Stuttgart and Aachen as well as the Hochschule Bremen.

The MEGAFLOW project consists of five major activities[1]: development of the block-structured grid generator MegaCads, improvement and enhancement of the block-structured parallel Navier-Stokes solver FLOWer, valida-

21st ICAS Congress
13-18 September 1998
Melbourne, Australia

tion of the software system for industrial applications for cruise and take-off/landing conditions, software quality management of the simulation system and investigation of the capabilities of alternative methods based on unstructured meshes. In addition, the post-processing tool MEGADRAG for the computation of aerodynamic forces and the aerodynamic optimization system MEPO are being developed. It should be emphasized that industrial applications are the basis for all development activities and that all partners are working on the same software according to clearly defined rules.

The present paper gives an overview of the current status of the project. It will present the plan of action, highlight the major achievements and point out the future developments.

2. Grid Generator MegaCads

The generation of high-quality computational grids for viscous simulations of complex aircraft configurations is one of the most challenging and pacing technologies for industrial CFD. It determines to a large extent the problem turn-around-time and the accuracy of the numerical solution.

Within the MEGAFLOW project, the interactive grid generation system MegaCads (Multiblock Elliptic Grid Generation and CAD System) is being developed[2] with the objective of complying as much as possible with the industrial requirements of aircraft manufacturers. The complete grid-generation sequence of multi-block grids, as e.g. definition of block topology, distribution of grid points on block boundaries and generation of surface and volume grids, is performed in a parametric manner and stored in a protocol file using a simple script language. In combination with interactive editing, the sequence can be reused for similar grid generation task with a minimum of user interaction.

MegaCads is separated into three major blocks: the graphical user interface, the graphic module for visualization and the module for CAD and grid generation techniques. The modules are connected by a few well-defined interfaces, which ensure the possibility of using the grid generator in a batch mode without the graphical interface and of using the visualization routines as a sub-function in design and optimization software. The visualization is based on OpenGL or Mesa, if the emulation on X11 basis is necessary.

A major effort is devoted to the improvement and enhancement of techniques for surface and volume grid generation for aircraft applications. Specific projection techniques were developed and implemented which allow the generation of grids on arbitrary surfaces. Furthermore, different algorithms for the determination of surface/surface interaction of aircraft components were integrated. Efficient two- and three-dimensional algebraic grid-generation techniques are available, which enable the construction of initial grids of high quality. In order to smooth the algebraic grids, elliptic techniques were implemented, allowing different controls for the point distribution at grid boundaries. Further activities to improve the elliptic grid generation for Navier-Stokes grids include advanced techniques for the specification of the control terms in the interior of the grid, algorithms to ensure grid smoothness across block boundaries and multigrid for convergence acceleration. The automation of MegaCads was greatly improved by implementing both functions for generating two- and three-dimensional surface offset grids as well as the data structure of a basic grid topology model. The latter allows the storage of the grid topology via an almost automatic detection of faces and blocks. Using this data structure, point distributions on block boundaries can be propagated through the grid and allow the automatic filling of surface and volume grids in the corresponding blocks.

Besides the development activities of MegaCads, three-dimensional grids around various MEGAFLOW related configurations were generated.[3] Fig. 1 shows the Navier-Stokes grid around the DLR-F6 wing/body/pylon/nacelle configuration. The grid consists of 55 blocks and 3.2 million points. In order to accurately resolve the boundary layers, all aircraft components were wrapped into C-type grids in streamwise directions and into O-type grids in spanwise directions. The blocks close to the boundary were embedded into H-type grid blocks which define the far field and the symmetry plane.

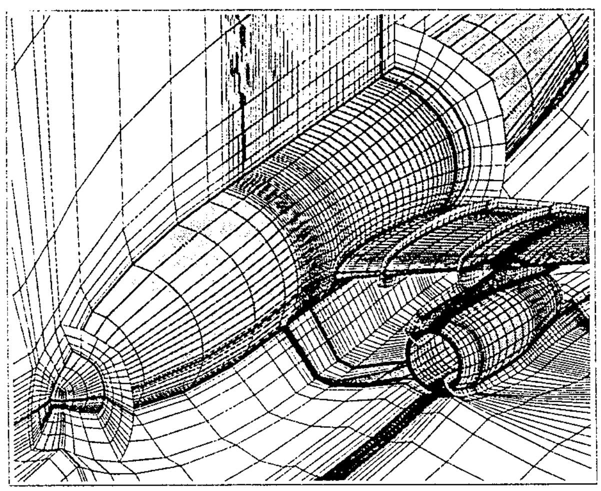


Fig. 1: Navier-Stokes grid for DLR-F6 configuration, 55 blocks, 3.2 million grid points.

Another application of MegaCads is the HH-type grid for a wing/body configuration with elevator and vertical fin, as shown in Fig. 2. The mesh consists of three blocks, a nose block extending from the nose to the upper far field, one block below and one block above the configuration. The objective of this application is to investigate whether a simple grid topology is able to resolve complex flow phenomena, in particular flow around the rear fuselage. First inviscid calculations are encouraging.

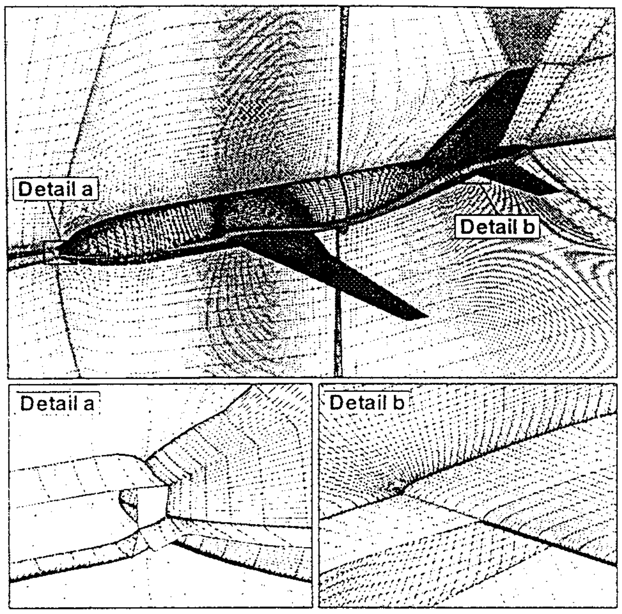


Fig. 2: HH-type grid around wing/body configuration with elevator and vertical fin.

The DLR-ALVAST high-lift configuration consists of a fuselage and wing with a nose slat along the complete wingspan and an inboard and outboard flap (Fig. 3). The grid generated with MegaCads for inviscid simulations is shown in Fig. 4 The slat and flaps are surrounded by O-type grids extend in spanwise direction to the far field. The current Euler grid consists of 53 blocks with approximately 3.5 million grid points. In this case, the subscript technique of MegaCads was intensively used which allow identical grid processes to be used repeatedly leading to a significant reduction of the grid generation effort. The extension of the Euler grid to a Navier-Stokes grid is under way.

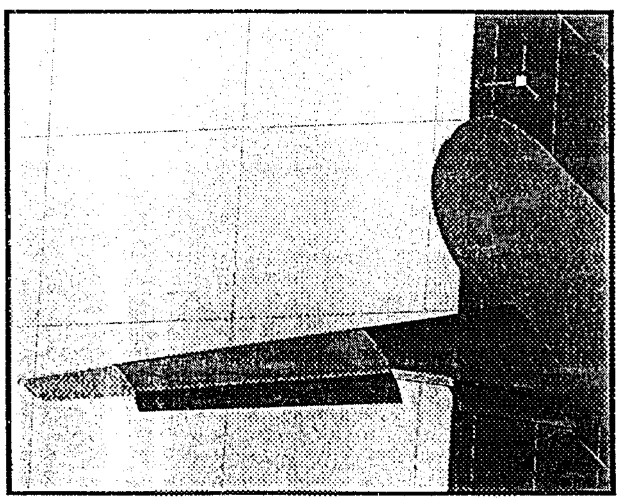


Fig. 3: DLR-ALVAST high lift configuration.

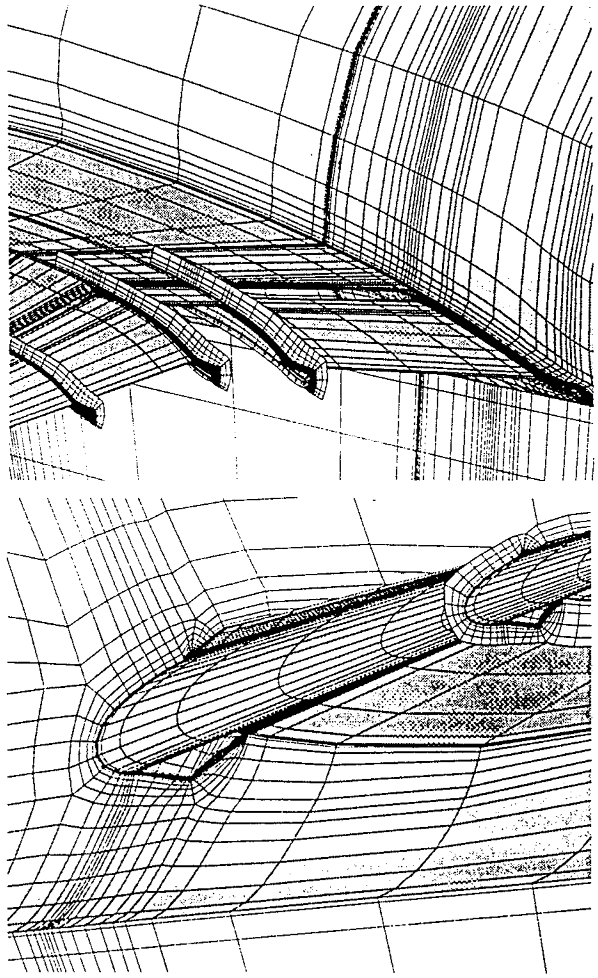


Fig. 4: Euler grid around DLR-ALVAST configuration, 53 blocks, 3.5 million grid points.

Generation of structured grids can be greatly simplified using overlapping grids. Fig. 5 shows an overlapping grid structure of a Navier-Stokes grid for a wing with flap-track fairing.

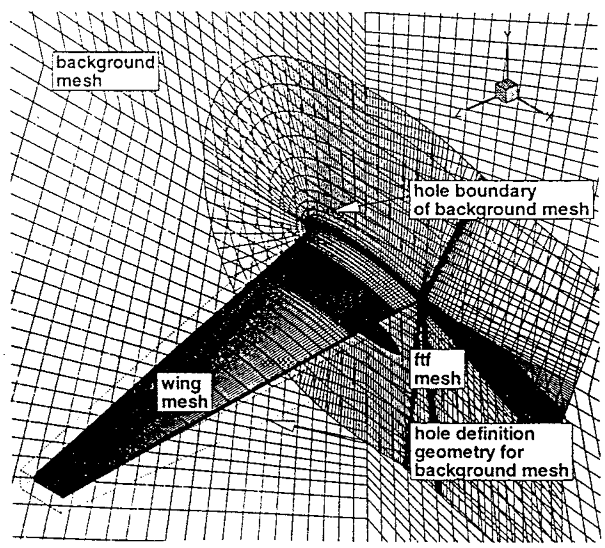


Fig.5:

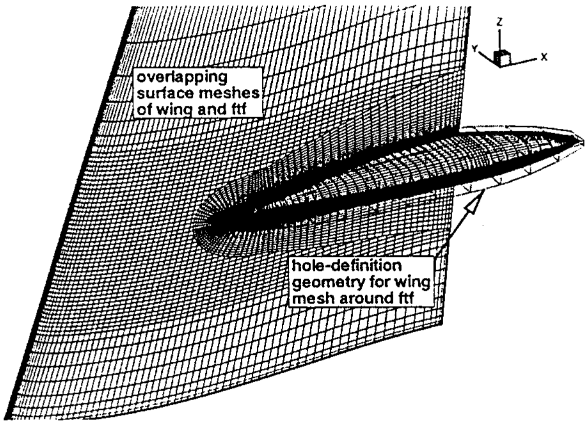


Fig. 5: Overlapping grid structure for a wing with flap-track fairing, Navier-Stokes grid.

As a result of the parametric concept and the availability of a batch mode, MegaCads is being used in an aerodynamic optimization loop. Fig. 6 shows the automatically updated grid for two flap positions of an optimization process to determine the optimal flap position of a multi element high lift airfoil.[4] The slat, main airfoil and flap are surrounded by C-grids to resolve the confluent flow of the different wakes. The flap position may vary between +/- 5% of the main airfoil chord length in x, +/- 2.5% in y-direction and between 15^0 and 35^0 in the deflection angle without loss of grid quality.

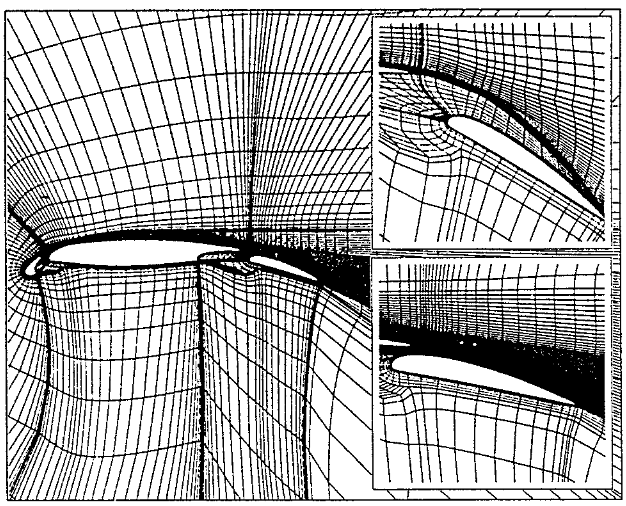


Fig. 6: Variation of flap position within an optimization loop.

3. Navier-Stokes Code FLOWer

The FLOWer code solves the compressible, three-dimensional Reynolds-averaged Navier-Stokes equations. It uses a cell-vertex, finite-volume formulation on block-structured meshes. The baseline method employs central space discretization with artificial viscosity and explicit multistage time-stepping schemes. For steady calculations convergence is accelerated by using local time stepping, implicit residual smoothing and multigrid. FLOWer has been directly evolved from the DLR CEVCATS code[5], and it includes special features from the DASA Airbus MELINA code and the DASA-M IKARUS code. In the framework of the German research project POPINDA, in close cooperation with the German aeronautical industry and the German National Research Center for Information Technology (GMD), it was extended to be a parallel, fully portable code[6]. The parallelization is based on the communication library CLIC-3D which was jointly developed by the GMD and the C&C Research Laboratories of NEC. Since the CLIC Library supports both MPI and PARMACS, the library and therefore the FLOWer code are portable across all machines for which these message passing interfaces are available.

Within the MEGAFLOW project, the development activities for the flow solver are concentrated on the improvement of efficiency and accuracy for viscous flow simulations. One objective is the utilization of the parallel FLOWer code for complex applications. Since the effective use of parallel computers requires load balancing of the computation, a tool was developed which splits an arbitrary block structured grid based on the number of processors available and modifies the block topology accordingly. Fig. 7 shows the speed-up on an IBM-SP2 for an Euler simulation of the transonic flow around a generic wing/body/pylon/engine configuration.

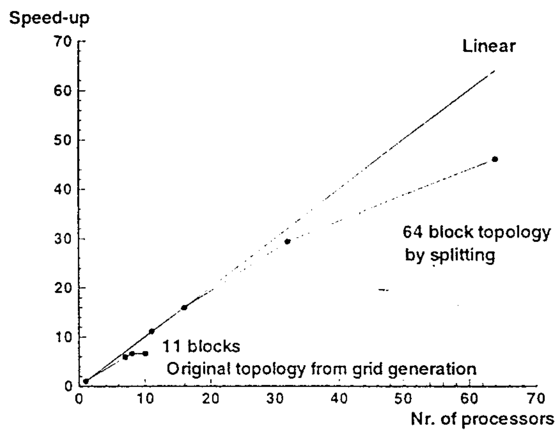


Fig. 7: Speed-up for a generic wing/body/pylon/engine configuration on IBM-SP2, Euler calculation on grid with 575.000 cells.

Due to reasons of grid generation, the initial block-structured grid consisted of 11 blocks with greatly varying block sizes. In total, the mesh has 575.000 cells. Increasing the number of allocated processors and mapping the block structure thereon, there is an increase in speed-up-as long as the largest block is not solely solved on one single processor. Using the pre-processor to split the block structure into blocks with almost equal block size, satisfactory speed-ups with up to 64 processors could be obtained. On 64 processors the computation time was less than 10 minutes. A Navier-Stokes calculation for a wing/body configuration using a 2-equation turbulence model on a CRAY-T3E with 128 processors demonstrated a performance of more than 7 GFLOPs.

Based on the work in the POPINDA project, a technique for self-adaptive local-grid refinement was further developed. Those parts of the grid are refined where high discretization errors of the flow quantities occur. The communication library CLIC-3D clusters the detected cells, creates a new block structure and distributes it in a load-balanced way to the allocated processors. The new block structure is embedded as a finer level in the multigrid structure. First results obtained for airfoils and wing/body configurations(7) are very encouraging (see Fig. 8). Currently work is being devoted to improve the fine/coarse grid interface treatment.

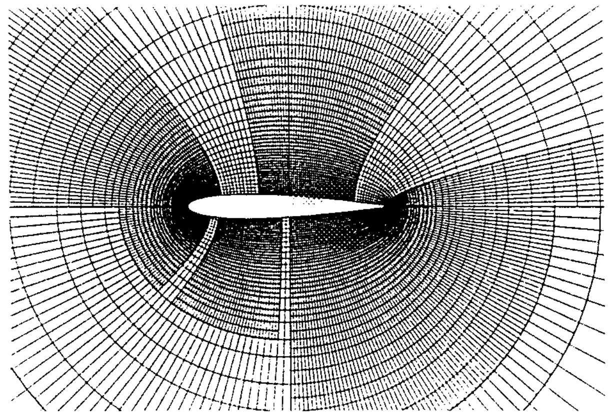


Fig. 8: Improved shock resolution for an infinite NACA 0012 wing at M_∞=0.8 and α=1.25° using local grid refinement.

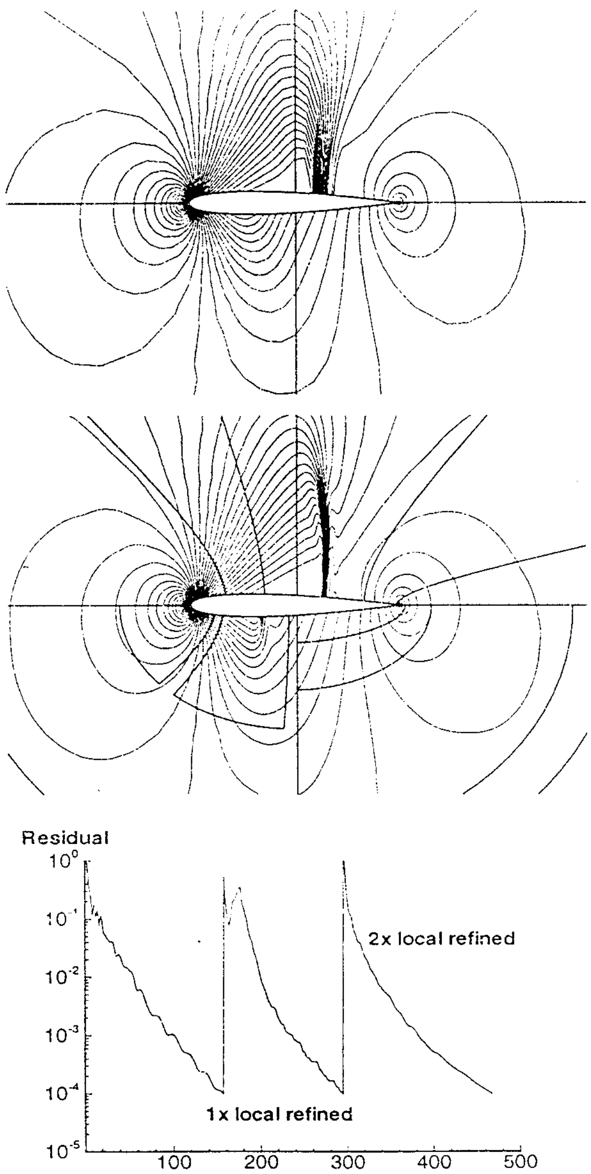


Fig. 8‘:

For the calculation of incompressible flows, preconditioning of the compressible flow equations according to Ref.(8) was implemented in FLOWer. This technique allows both, an improved accuracy and efficiency for low Mach number cases, such as high-lift flows. Fig. 9 presents results of a Navier-Stokes calculation for a 3-element airfoil with take-off conditions (M_∞=0.22, α=12.2°, Re=4*10^6^). The grid consists of 7 blocks and has 55.000 grid points. The calculation was carried out with the k-ω 2-equation turbulence model using the multigrid acceleration technique. The improvement of convergence through preconditioning is clearly demonstrated.

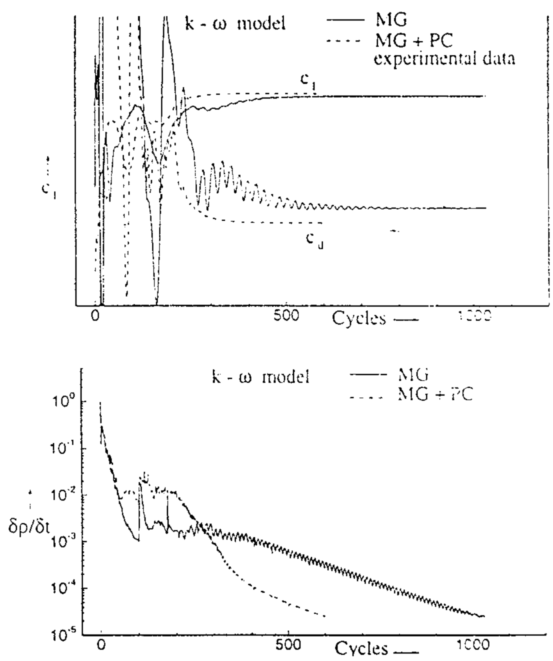


Fig. 9: Convergence behavior for the Navier-Stokes calculation with k-ω turbulence model for a 3-element airfoil with take-off condition (M_∞=0.22, α=12.2°) with and without preconditioning.

Multiblock grids for complex configurations often contain geometrical and topological singularities. In a cell-vertex code, a special treatment of singular points or lines is necessary in order to avoid accuracy and efficiency losses. A procedure was implemented which detects singularities in a given grid and discretizes corresponding inviscid, viscous and dissipative flux in a consistent manner.

In order to improve the drag computation in viscous flows, high resolution schemes are being implemented in the FLOWer code. Based on the experience gained with the DLR hypersonic code[9], the AUSM scheme[10] was integrated. For high aspect ratio cells a refined dissipation scaling is used which provides sufficient damping without smearing viscous shear layers[11]. As an alternative, the matrix dissipation approach is being implemented.

Since the prediction of aircraft or aircraft components requires an accurate simulation of the viscous, turbulent effects, the improvement of turbulence modelling in the FLOWer code is one of the major activities of MEGAFLOW. Based on the experience gained with the DLR CEVCATS code[12], the k-ω model of Wilcox was implemented as a standard 2-equation transport model. The model was implemented in a generalized way, in order to allow an effective integration of other, improved models which are being developed in the framework of the MEGAFLOW project (see chapter 4). Furthermore, in FLOWer the 1-equation model of Spalart-Allmaras is available. A point-implicit treatment of the source terms of the turbulence transport equations was realized resulting in an improved efficiency of the solution method. Besides the better description of turbulent flows and the wider range of physical applicability compared to algebraic models, transport equation models deliver results which are independent of the block-structure used in the calculation[13]. This is a necessary feature for an optimal use of parallel computer systems.

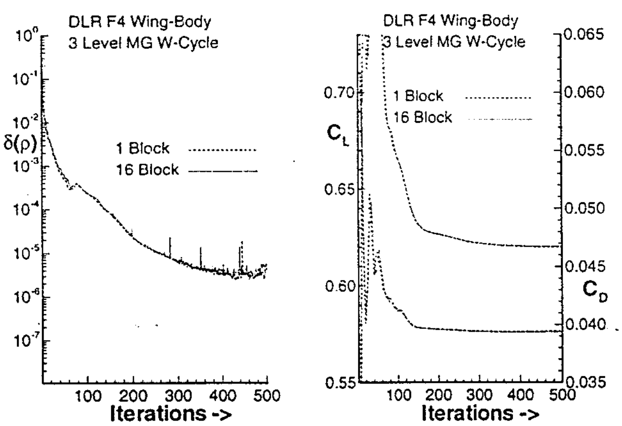


Fig. 10: Convergence behavior for k-ω turbulence model for DLR F4 wing/body configuration, M_∞=0.75, α=0.93°, Re=3*10^6.

Fig. 10 demonstrates the flexibility of the k-ω model with regards to multiblock grid topologies, for example the DLR F4 wing/body configuration. There are no differences in convergence rate and total forces between the 1- and 16- block calculation.The k-ω model forms the basis for the applications carried out in the MEGAFLOW validation phase (see chapter 5). Current activities are devoted to the improvement of the robustness and efficiency of the k-ω model for complex configurations and high Reynolds numbers.

In order to simplify the generation of grids for complex configurations, the Chimera technique was implemented into the FLOWer code. This approach allows the grid blocks to overlap each other. The communication from mesh to mesh is realized through interpolation in the overlapped area. Hence, searching algorithms were implemented to locate the donor cells. In the case when a mesh overlaps a body which lies inside another mesh, hole cutting procedures have to be used in order to exclude the invalid points from computation. At the hole-boundaries, the flow quantities have to be provided through interpolation. The points of the background mesh lying inside the holes are blanked out. In FLOWer various strategies are implemented to efficiently create holes, even in the case of complex configurations. The Chimera technique was verified for different test cases. Fig. 11 shows results of a viscous simulation for an infinite wing.

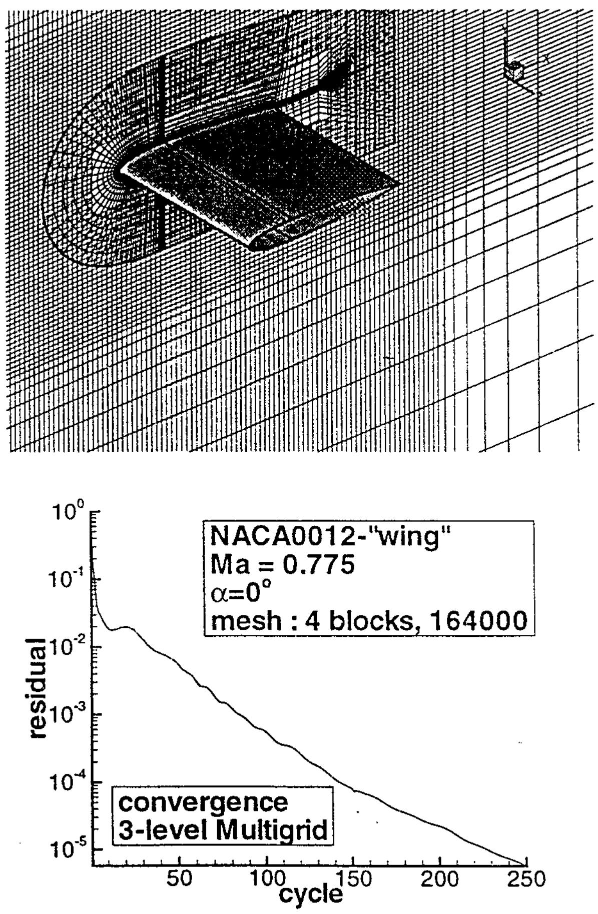


Fig. 11':

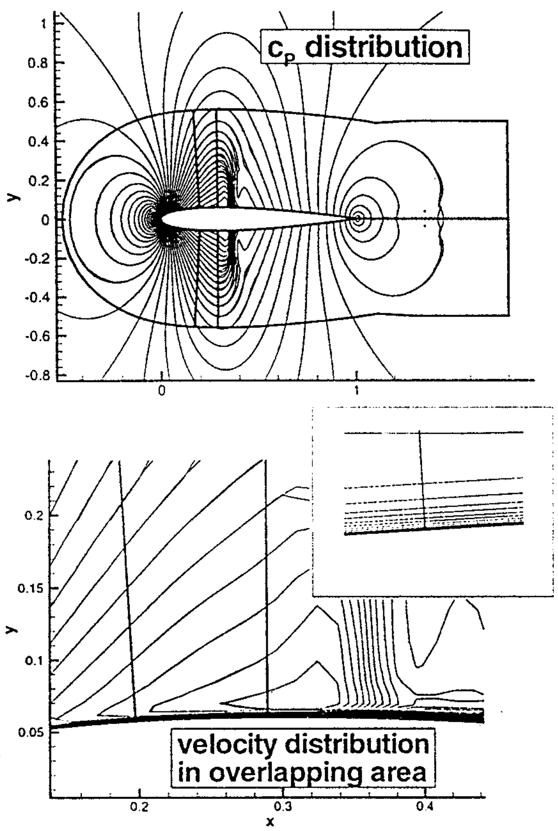


Fig. 11: Navier-Stokes simulation on a Chimera grid for an infinite wing.

In this case overlapping grids on the surface were used. A quite good convergence behavior using multigrid was obtained. A close inspection of the solution shows that there is no influence of the Chimera boundaries even in the boundary layer. Currently the Chimera technique is being applied to more complex configurations (Fig. 5).

With respect to unsteady calculations, the FLOWer code was extended to simulate time-accurate flows around a rigid body in arbitrary motion. The motion is taken into account by a transformation of the governing equations. In order to bypass the severe time-step restriction associated with explicit schemes, a simple implicit method, known as the dual time stepping approach[14], was implemented[15]. In combination with multigrid acceleration, this scheme allows very efficient calculations of viscous time-accurate flows: For oscillating wings (see Fig. 21) a speed-up of about three orders of magnitude compared to the basic explicit scheme was obtained. Recently, the code was extended to allow grid deformation. This enables the FLOWer system to be used for aeroelastic simulations[16]. Fig. 12 presents Navier-Stokes results for the oscillating NLR 3701 airfoil (AGARD test case CT5). There are no differences between the calculation using a fixed or a deforming mesh. The comparison of the numerical results are in good agreement with experimental data.

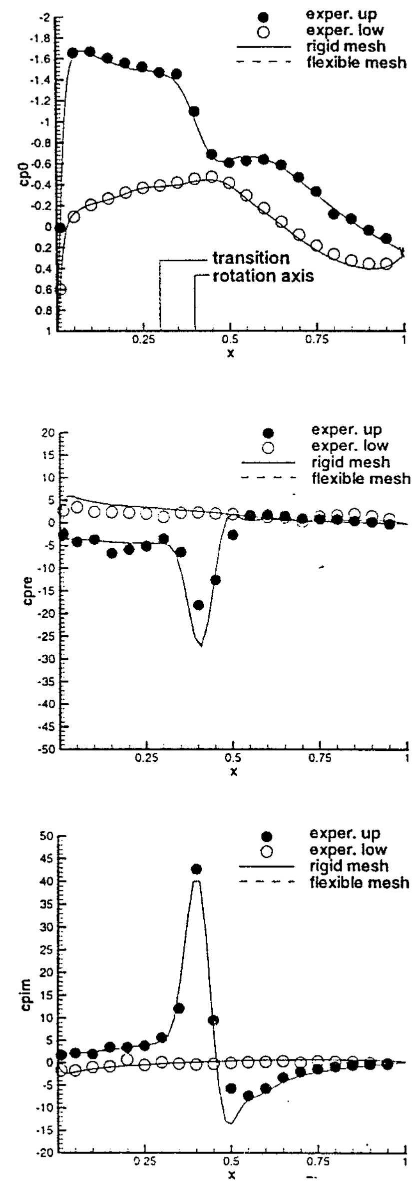


Fig. 12: Navier-Stokes results with fixed and deforming grid for the oscillating NLR 3701 airfoil, $M_\infty=0.7$, $Re=2.12*10^6$, $k=0.192$, $\alpha=2.0^o+0.5^o\sin(\omega t)$.

4. Turbulence Modelling for Complex Aerodynamic Flows

Accurate numerical simulation of aerodynamic configurations at optimum-design or off-design conditions requires an appropriate computational modelling of the entire flow problem. In such circumstances, the predictive quality of the flow's characteristics is governed by seemingly subtle details pertaining to the representation of turbulence in apparently innocuous portions of the flow. Therefore, the present joint collaborative research effort focuses not only on refined numerical approximation techniques, but also on the accurate predictive response to complex turbulent motion based upon a Reynolds-averaged approach.

Recent studies have demonstrated the inability of the most prominent standard 'eddy-viscosity' turbulence models to mimic the fundamental physics of turbulence in flows exposed to non-equilibrium high-load conditions. The principal aim of this MEGAFLOW subtask is to incorporate much more sound turbulence physics into these models while retaining their numerical advantages. Attention is confined to the development of linear(17) and non-linear(18) eddy-viscosity models covering an enhanced range of validity, and their assessment in complex flows of industrial relevance. The key features of the modelling procedure are:

- topography-independent low-Re formulations which facilitate a reliable description of turbulence transport across the entire domain with arbitrary geometries, improved representation of stress-strain relations,
- adherence to the realizability principle in order to prevent severely unphysical results, like negative turbulence energy components,
- modelling of general non-equilibrium and non-inertial effects.

The final formulations are cast in terms of a wide range of one- and two-equation baseline models.

The numerical simulation of high-lift aerodynamics is the first important validation example related to modern aircraft design. The considered test case, the Onera A-airfoil(19) at 13.3° angle of attack for $Re = 2 \cdot 10^6$ and $M_\infty = 0.15$, features separation from the trailing edge as well as a laminar separation bubble upstream of the transition point on the suction side.

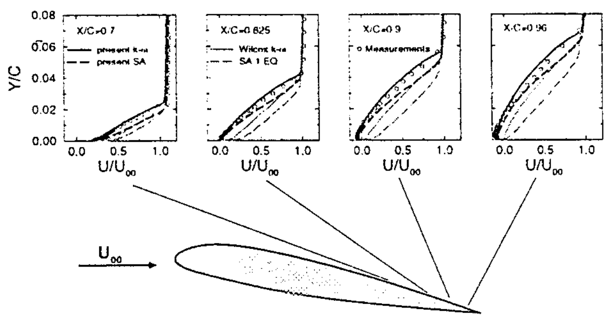


Fig. 13: Validation of linear isotropic viscosity models for airfoil flows exposed to high-lift conditions.

As indicated in Fig. 13, the linear models under development reveal the improvements of an advanced modelling practice in high-load conditions, where maximum lift is obtained in conjunction with high localized strain rates and an attenuated level of stress anisotropy.The linear isotropic-viscosity concept still has a variety of deficits. Isotropic-viscosity models indubitably fail to describe turbulent flows with enhanced 3D and body-force effects due to a curvature-induced variation of shear stress or any secondary motion driven by severe normal-stress anisotropies. Although a general approach would, arguably, be based on a second-moment closure, the present research effort tries to adopt a computationally cost-effective solution. To remedy the shortcomings of the linear approach, a non-linear constitutive relation based on the principles of rational mechanics is introduced. This approach represents an explicit algebraic solution to the second-moment closure in the limit of equilibrium turbulence, and can be regarded as a generalized non-linear two-parameter model.

A careful examination of the applied turbulence closure is particularly important with regards to shock boundary-layer interaction. Fig. 14 demonstrates the benefits of the explicit algebraic stress model over the linear isotropic-viscosity concept when applied to Delery's(20) well-known compressible bump flow (Case C).

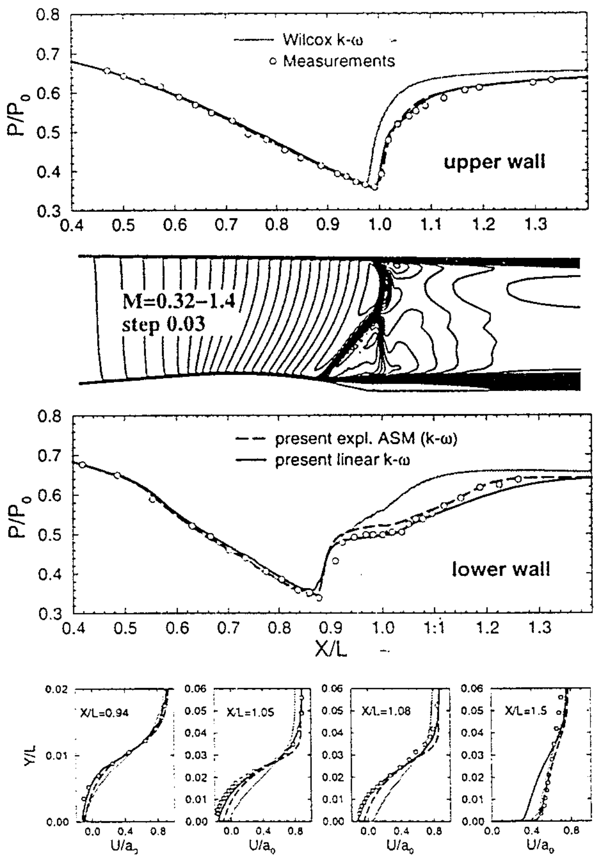


Fig. 14: Results for Delery's compressible bumb flow (Case C, $Re=1.1x10^6$, $M_{in}=0.6$, $p_{out}/p_0=0.595$).

In the reported example, a high-pressure reservoir drives a fully-turbulent flow through a 2D channel with a bump on the lower wall. A lambda shock-structure forms on the trailing edge of the bump provoking a shock-induced separation of the boundary-layer. Both, the linear and the non-linear models under development have a reasonable predictive accuracy in the upstream part of the separated flow region. However, the proposed non-linear model clearly outperforms the linear alternatives in the recovery regime downstream of the reattachment. Due to a more accurate representation of the severe stress-anisotropy in the vicinity of the shock an augmented return-to-isotopy process is initiated in downstream direction, which results in a sufficient recovery of the mean momentum.

The further development of turbulence models is being supported by investigations based on direct numerical simulation (DNS). In particular the physical modelling of the shock/boundary-layer interaction requires the knowledge of the flow details which at the present time can not be provided by experiments. DNS calculations are being carried out for a channel flow at a supersonic Mach number with and without induced separation.

5. Validation

Validation is a central issue for industrial application of CFD codes. Confidence in the numerical results is essential if CFD should be used as a reliable tool for the aerodynamic design of aircraft. Consequently, validation is one of the major tasks of the MEGAFLOW project. The validation objectives are

- to check the consistency of the numerical results
- to judge them against experimental data
- to study the sensitivity of the flow solver against variations of numerical and physical parameters
- to develop strategies for the most effective and reliable use of CFD within aerodynamic design cycles.

There are several critical cases and conditions for which the FLOWer code has to be validated. The range is from simple 2D to complex 3D configurations, from inviscid to viscous flows, from low to high subsonic free stream Mach numbers, from low to high Reynolds numbers, from laminar to turbulent flows and from attached to separated flows. All these items are relevant for large transport aircraft aerodynamics. Thus the validation of the MEGAFLOW system must cover a wide range of these flows in order to make the system a useful tool for aircraft industry.

Cruise Flight Test Cases

First, Euler calculations for a complete aircraft at free-stream Mach number M_∞=0.85 were performed. To make those calculations useful for aerodynamic investigations, the wing geometry was expanded by an estimate of the boundary layer displacement thickness. This estimate was calculated using an iteratively coupled full potential/boundary-layer code.[21] The aircraft configuration (half configuration) consists of body, wing, two pylons with nacelles, six flap track fairings and the tailplanes. The structured mesh has 40 blocks and about 3.4 million grid points. 145 points are distributed around the wing surface in streamwise direction and the resolution in spanwise and normal direction is 85 and 89 points, respectively. Due to the high subsonic free stream Mach number, the flow locally exceeds the speed of sound in a very large region. The calculation with a 3-level multigrid method using 50 iterations on the coarse and medium level each and 200 iterations on the finest level took 4 hours CPU time on a single NEC SX-4 processor. Convergence was quite good as can be seen in Fig. 15.

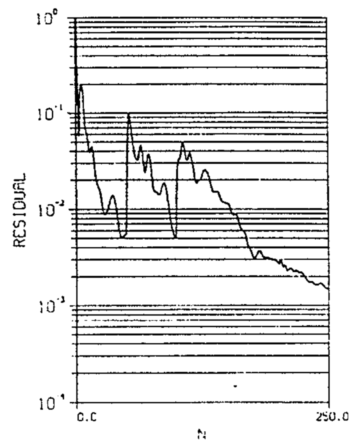


Fig. 15: Convergence behavior for Euler simulation of complete aircraft configuration for M_∞=0.85 and α=2.07^0, 40 block mesh with 3.4 million grid points.

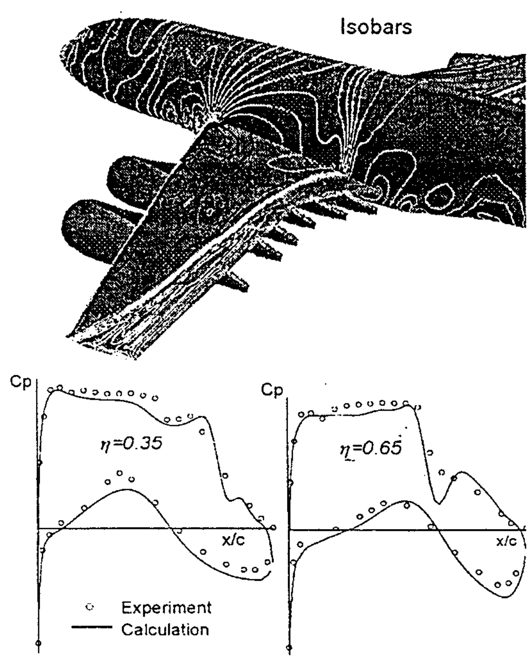


Fig. 16: Surface isobars and pressure distributions on wing sections for complete aircraft configuration for M_∞=0.85 and α=2.07^0, Euler simulation with boundary-layer displacement of the wing.

The pressure distribution in Fig. 16 indicates the degree of accuracy that can be achieved with this approach. The calculated pressure fits quite well to the measured values except near the shock and in the expansion region, as must be expected from the insufficient modelling of the viscous effects.

The next validation test case presents Navier-Stokes calculations for high speed wing/body combinations. Navier-

Stokes calculations have become standard for the evaluation of the 3D flow over new wing designs at Daimler Benz Aerospace Airbus. Typical tasks are pressure, drag and lift predictions for a range of Mach numbers and angles of attack at wind tunnel and free flight Reynolds numbers. The mesh topology used here is a one block C-O mesh with 289 points in circumferential direction, 89 points in spanwise direction and 89 points perpendicular to the wing surface. The k-ω turbulence model is used with about 30 mesh points in the boundary layer. Again 3-level multigrid is applied. A typical run on a NEC SX-4 processor takes about 1.5 hours with 200 multigrid cycles on the finest grid. Fig. 17 shows the comparison between the calculations and wind tunnel measurements for M_∞=0.85, Re=2,8*10^6,α=2,3°.

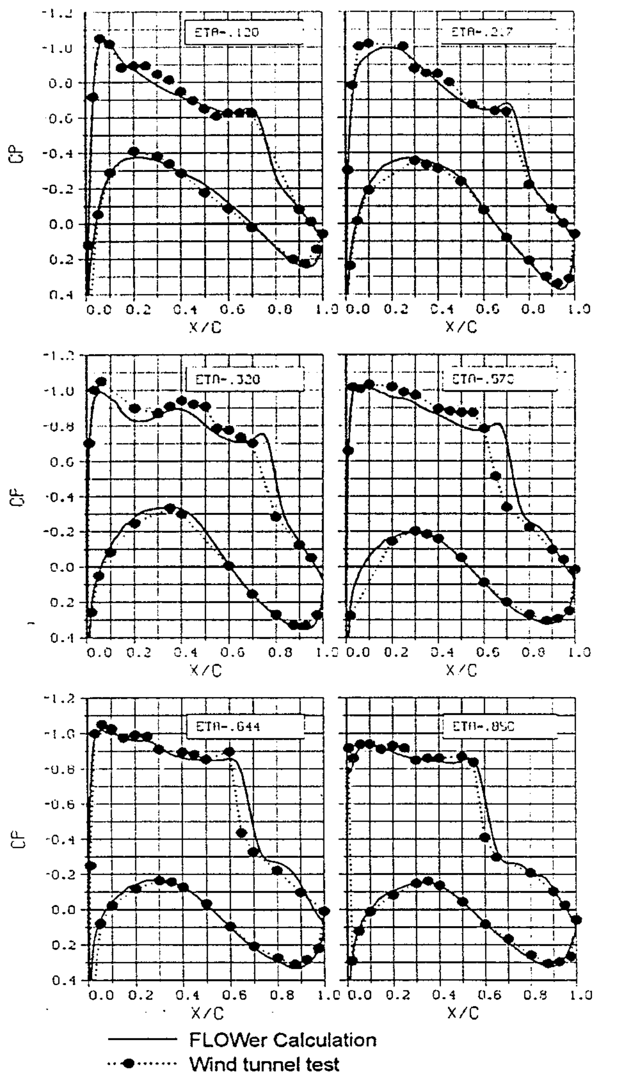


Fig. 17: Navier-Stokes solution for a wing/body-configuration with k-ω turbulence model, M_∞=0.85, Re=2,8*10^6,α=2,3°.

A wing/body/winglet configuration has been tested at flight conditions. The C-O type mesh has about 2.2 million grid points. Due to the large Reynolds number of 61 million, the mesh exhibits cells with considerable high aspect ratios in the vicinity of the wall in order to achieve y^+ values of the order of one. The numerical solution produced with the Baldwin/Lomax model is in good agreement with the free flight measurements (Fig. 18). The convergence of the method is not all affected by these extreme flow conditions (M_∞=0.87, Re=61*10^6,α=1.5°).

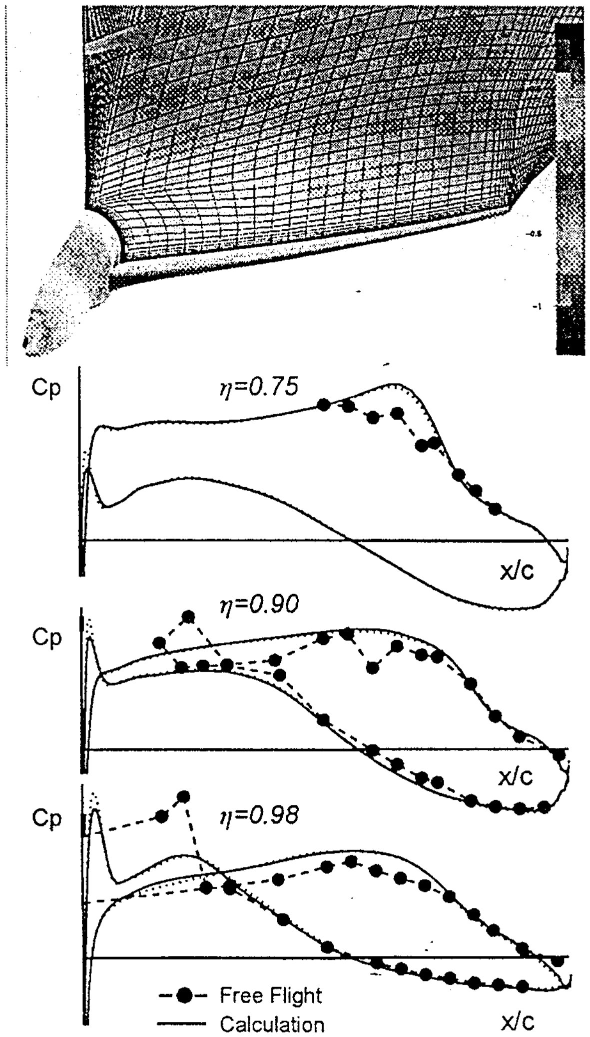


Fig. 18: Navier-Stokes solution for wing/body/winglet-configuration at free flight condition, M_∞=0.87, Re=61*10^6,α=1.5°.

In industry it is essential to have very fast turn-around-time, at least for basic configurations such as wing/body configurations. At Daimler-Benz Aerospace Airbus, the turn-around time for Navier-Stokes simulations for wings using the MEGAFLOW system can be broken down to half an hour for the preparation of the new geometry, 10 minutes for 3D mesh generation and about 90 minutes for the flow calculation using FLOWer. Solution analysis takes about 20 minutes, including data transfer time between the externally located supercomputer and the in-house workstations. Thus the overall time for a flow analysis is 150 minutes for a completely new designed wing, which reduces to 110 minutes for the simulation of an additional flow case.

One of the major validation objectives is the prediction of transonic cruise drag for wing/body/engine configurations with an accuracy of 2-3%. The test case selected is the DLR F6 configuration shown in Fig. 1. The numerical results obtained with the Baldwin/Lomax turbulence model are in good agreement with the experimental data. Fig. 19 shows the isobars on the surface and the pressure distributions in two sections near the engine for $M_\infty=0.75$ $\alpha=0.98^0$ and $Re=3*10^6$. The drag polar is given in Fig. 20. The drag could be predicted with an accuracy of less than 5%. Currently calculations with the k-ω two equation turbulence model are under way.

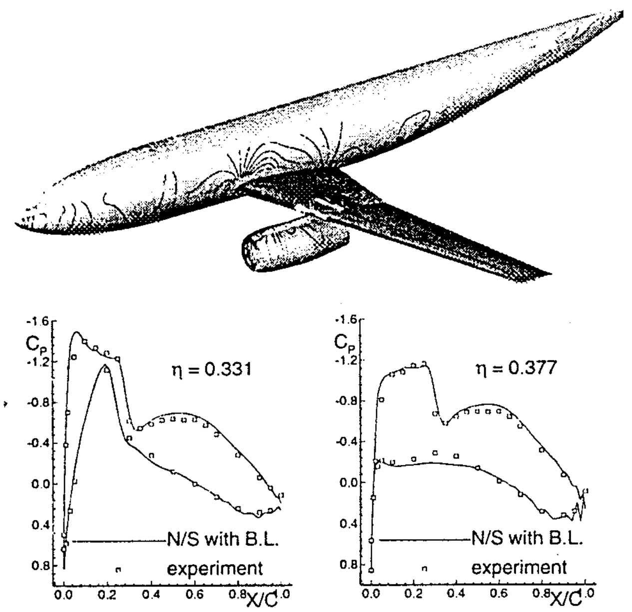


Fig. 19: Surface isobars and surface pressure distributions at two wing sections near the engine for the DLR F6 configurations, Navier-Stokes solutions with algebraic turbulence model.

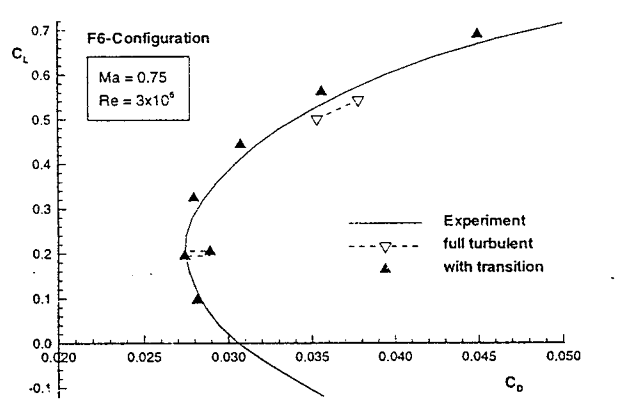


Fig. 20: Drag polar for DLR F6 configuration computed with algebraic turbulence model.

The time-accurate FLOWer code has been validated for the oscillating LANN wing.(15) Two test cases, known in the literature as CT5 and CT9, were selected. Whereas the flow is attached over the entire wing for the CT5 test case, the CT9 test case is characterized by a shock induced separation. Since no elasticity effects were taken into account, the mesh was described by a rigid body movement. The grid is a CH-type grid and contains about 425.000 points. Fig. 21 shows the results of the Fourier analyzed pressure distributions of Navier-Stokes calculations using an algebraic turbulence model for one typical wing section (47.5%). In addition, results of an inviscid calculation and the experimental data are plotted. For the CT5 test case, where the flow is attached, good agreement of the Navier-Stokes results and experimental data could be achieved. In the case of the separated flow (CT9), the shock is computed too far downstream. Compared to the inviscid calculation the Navier-Stokes result is much better, but there are still remarkable differences between simulation and experiment. It is assumed that the use of a two-equation model will permit a better description of the flow phenomena in the regions of separated flow. Corresponding calculations are under way.

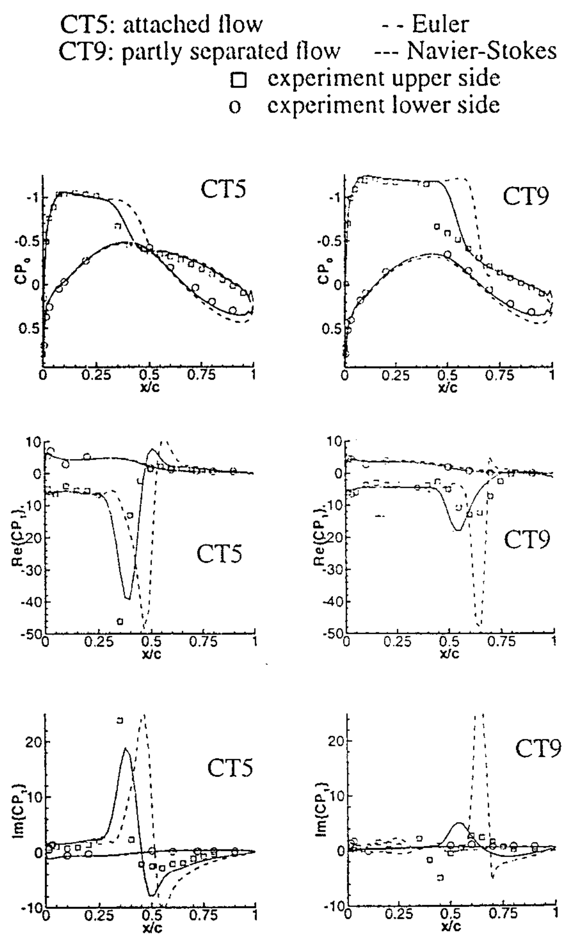


Fig. 21: Mean pressure distribution and first Fourier components for CT5 and CT9 test case of the LANN wing, Navier-Stokes calculations with algebraic turbulence model.

The calculation for one test case took one hour on the

NEC-SX4 using 4 processors. This shows that with the dual-time stepping approach and parallelization 3D viscous time-accurate simulations are feasible.

High-Lift Test Cases

With respect to high-lift configurations, the MEGAFLOW system was successfully applied to several single and multi-element airfoils using 2-equation transport turbulence models.(12) As an example, results are shown here for the L1T2 3-element airfoil with a slat (deflection angle 25°) and a Fowler flap (deflection angle 20°).(22) The flow conditions are M_∞=0.197, α=20.18° and Re=3.52* 10^6 Navier-Stokes calculations were carried out on a block-structured grid with about 180.000 grid points.(23) Fig. 22 shows the calculated streamlines in the front and rear part of the configuration. In the coves of the slat and the main airfoil large recirculation zones occur. In both cases a dominant vortex with a counterrotating small vortex is generated. Fig. 23 shows the comparison of the predicted and measured pressure distributions. The agreement is rather good. The lift could be predicted within 2% and the drag within 1%.

Fig. 22: Streamlines for L1T2 3-element airfoil calculated with k-ω turbulence model.

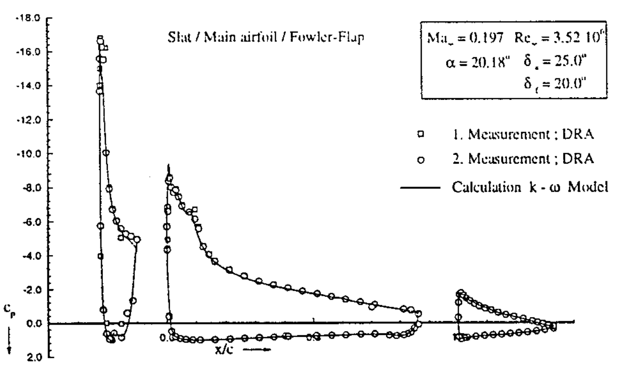


Fig. 23: Surface pressure distribution for L1T2 3-element airfoil calculated with k-ω turbulence model.

As a first step towards the validation of three-dimensional high lift configurations, a generic configuration consisting of an unswept wing with a flap was calculated.(23) The flap covers half of the wingspan and is deflected by 30°. The flow conditions are M_∞=0.2, Re=3.7*10^6 and α=10°. The grid generated by MegaCads has 13 blocks and about 2.2 million points. The calculation was carried out with the k-ω turbulence model. Fig. 24 presents the pressure contours and streamlines on the on the upper surface of wing and flap. In addition, the vortex which generated at the side edge of the flap is shown.

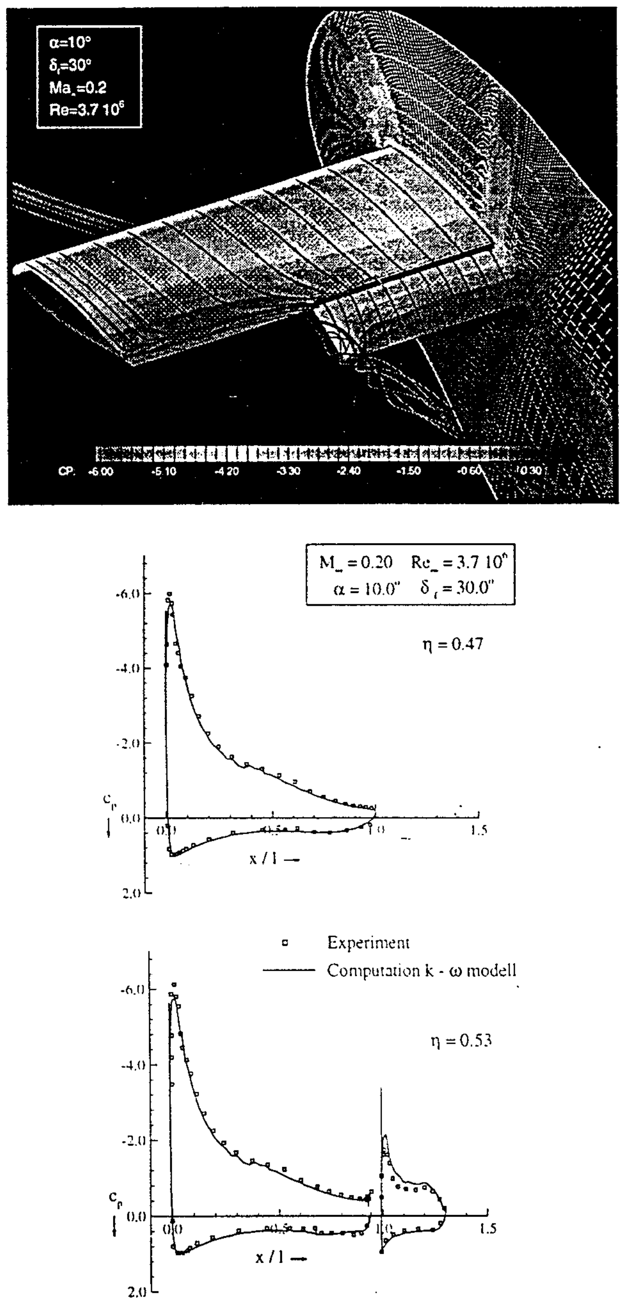


Fig. 24: Navier-Stokes computations for a wing/flap configuration using k-ω turbulence model.

It is evident that in the outer part of the wing near the trailing edge the flow exhibits a highly three-dimensional character. The stream lines are turned towards the mean section and run into the edge vortex. Due to the extreme curvature, the flow separates near the trailing edge. In the region of the wing, where the flap is deflected, as well as on the flap itself the curvature of the stream lines is much weaker. The comparison of the calculated and measured surface pressure in two wing sections near the side edge of the flap demonstrates the good agreement of prediction and experiment.

The FLOWer code was used for Navier-Stokes calculations for a wing/body configuration at high angles of attack that are typical for the development of the high-lift devices and low speed aerodynamics. Again the k-ω 2-equation turbulence model was used. As can be seen in Fig. 25, the Navier-Stokes code converges quite well even for the low free stream Mach number $M_\infty = 0.2$. The comparison with the experimental surface pressure data (see Fig. 26) shows the quite high quality of the numerical flow solution. However, the tendency of the flow to separate near the trailing edge is not predicted by the current code. It is anticipated that future developments of the MEGAFLOW system will be able to more accurately describe transition and separation effects. The complexity of the flow field for $\alpha = 12°$ is indicated in Fig. 27.

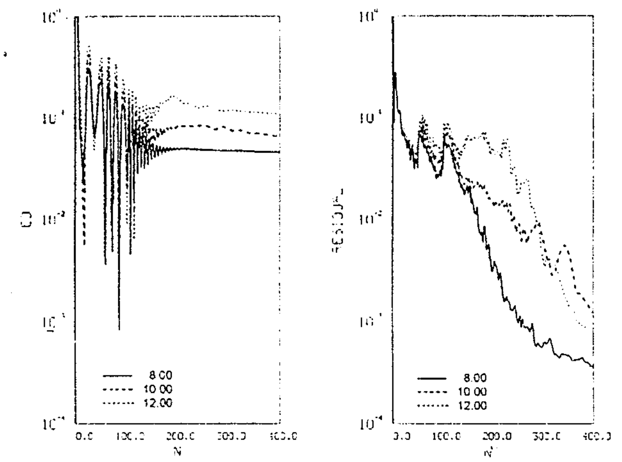


Fig. 25: Convergence behavior for Navier-Stokes code FLOWer for a wing/body configuration for low free stream Mach number ($M_\infty = 0.2$) and high angle of attack.

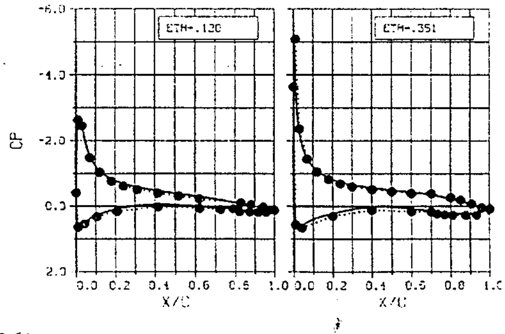


Fig. 26':

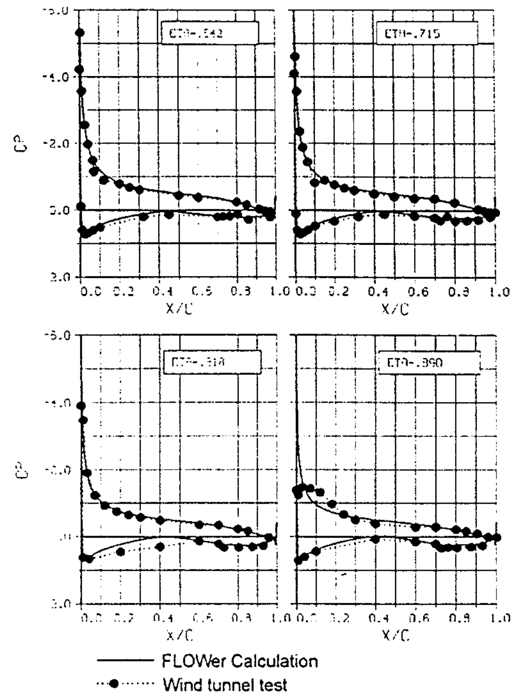


Fig. 26: Surface pressure distributions for a wing/body configuration at $M_\infty = 0.2$ calculated with the k-ω turbulence model.

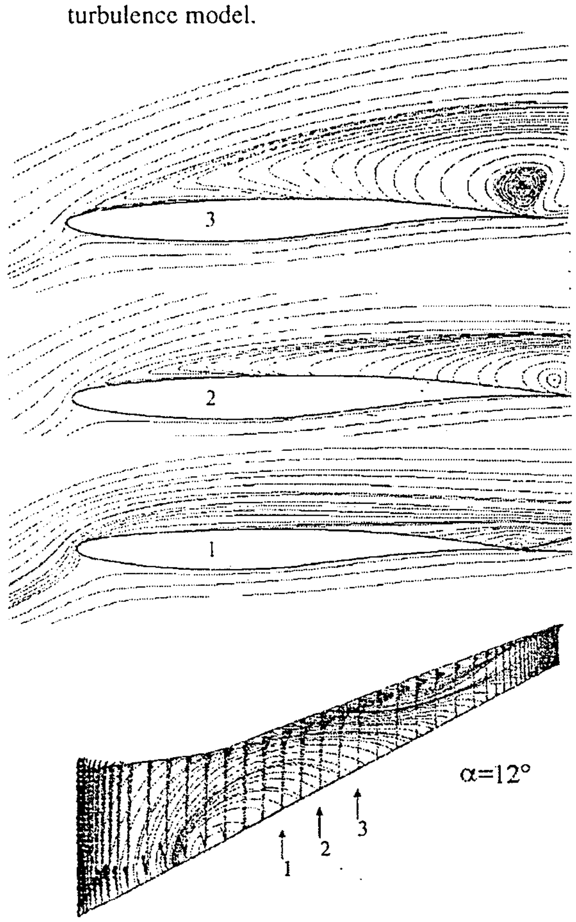


Fig. 27: Streamlines for wing/body configuration at $M_\infty = 0.2$ and $\alpha = 12°$ calculated with the k-ω turbulence model.

Work is under way to validate the FLOWer code for the DLR-ALVAST high-lift configuration which consists of a fuselage and a wing with nose slat and inboard and outboard flaps, see Fig. 3. As a first step a block-structured grid for inviscid simulations was generated (Fig. 4). Results of an Euler calculation for the free stream conditions M_∞=0.22 and α= 12,03° are presented in Fig. 28 and Fig. 29. The calculation was carried out with a 3-level multigrid method. The residual drops down 3.5 orders of magnitude within 500 cycles.

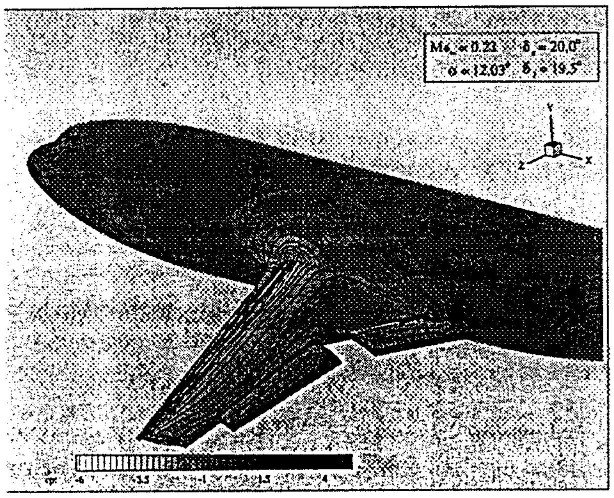
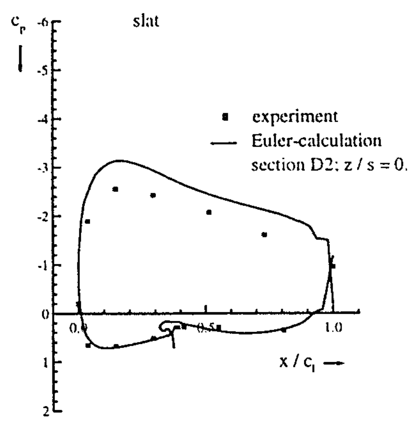


Fig. 28: Isobars on the surface of the DLR-ALVAST high-lift configuration, Euler calculation.



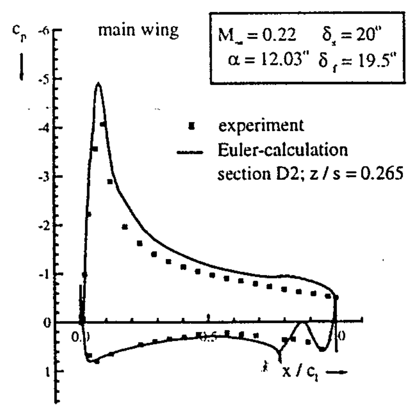
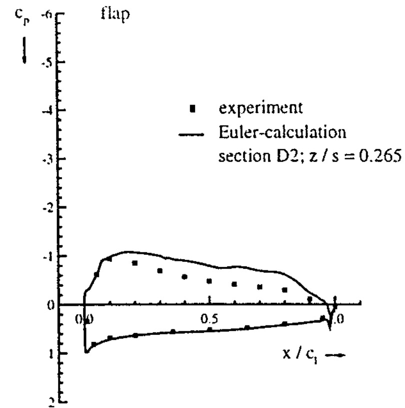


Fig. 29':



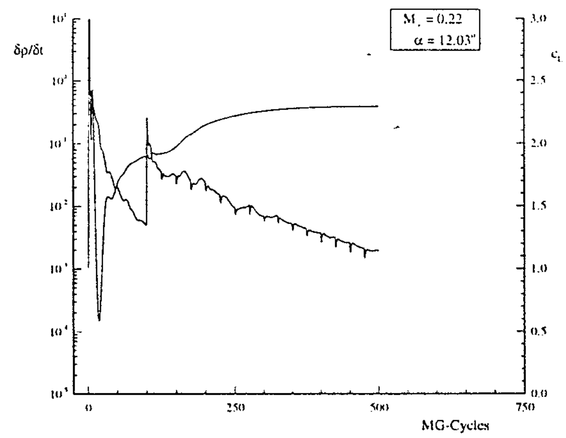


Fig. 29: Convergence behavior and surface pressure distributions at inboard wing section for the DLR-ALVAST high-lift configuration, Euler calculation.

As expected, the comparison of predicted and measured surface pressure distributions shows consistent quantitative discrepancies. The computed lift is somewhat above the experimental data. However, the qualitative behavior of the flow is predicted quite well. Navier-Stokes calculations are in progress.

Postprocessing Tool for Drag Prediction

For aerodynamic developments of transport aircraft the central question is about the forces. Therefore, a postprocessing and analysis tool called MEGADRAG is under development(24), which enables the separation of physical drag sources such as viscous drag, wave drag and induced drag. Additionally, the location of drag generation can be made visible which may be the basis of further aerodynamic improvements of any configuration. Implementation of MEGADRAG is done in consistency with the coding of the Navier-Stokes code FLOWer. First validation exercises for generic test cases and industrial wing/body flows yielded quite satisfactory results

The capability of MEGADRAG to predict lift induced drag was demonstrated for an untwisted wing with elliptic planform and aspect ratio of $\Lambda=7$. Euler calculations for subsonic flow ($M_\infty=0.5$) were carried out on three different fine meshes. Since the flow is subsonic, the total drag only consists of lift induced drag. The MEGADRAG post-processing tool calculates the induced drag by using a wake-integral method. All flow quantities are interpolated onto a computational plane behind the wing which is perpendicular to the free stream direction. The induced drag is obtained by solving the Laplace equation for the streamfunction in this plane. Fig. 30 shows the drag coefficient as a function of the distance behind the wing trailing edge. From the table it is obvious that the drag calculation using a wake-integral method is quite accurate even on relatively coarse meshes. The pressure integration on the wing surface leads to a wrong result.

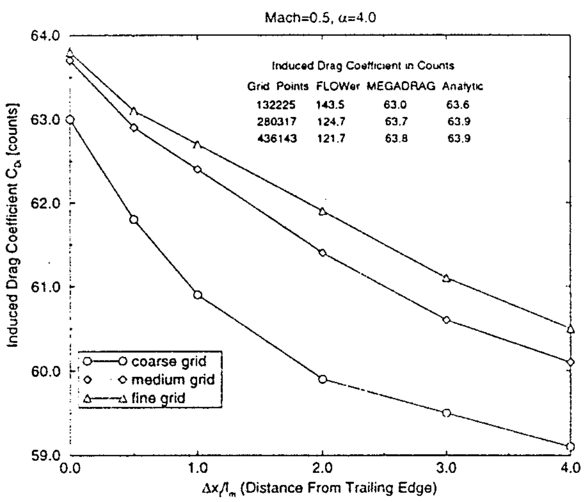


Fig. 30: Prediction of induced drag in inviscid flow for a wing with no twist and elliptic planform.

6. Software Quality Management

In order to insure the supportability, dependability and further expandability of the MEGAFLOW system, modern quality management and software engineering methods were implemented. In close cooperation with the DLR software quality control department, mandatory procedures were established to integrate the software components developed by the different partners into the central versions of MegaCads and FLOWer. Guidelines were developed for programming, documentation, testing, error reporting and problem reporting. For software version control, the commercially available configuration management tool CONTINUUS was introduced. The corresponding documents are continuously updated.

7. Alternative Methods

Although the main objective of the MEGAFLOW project is the development of a prediction capability based on the block structured concept, research activities are also being devoted to unstructured approaches. The goal is to assess the potential and limits of unstructured methodologies for predicting turbulent flows at high Reynolds numbers. The work is concentrated on the further development of the hybrid DLR τ-Code for transport aircraft application[25]and on detailed comparisons with the structured FLOWer code for technically relevant test cases. This effort is supported through activities carried out in the framework of a DLR/NLR cooperation as well as the Brite-Euram CFD projects FASTFLO I and II.

The τ-code uses hybrid grids, which consist of a combination of prismatic and tetrahedral cells and therefore combine the advantages of regular grids for the accurate resolution of viscous shear layers in the vicinity of walls and the flexibility of grid generation techniques based on unstructured meshes. The use of a dual mesh makes the solver independent of the type of cells that the initial grid is composed of. Various discretization schemes were implemented including a central scheme with artificial dissipation and upwind methods based on the AUSM and Roe formulation. For the simulation of turbulent flows, the one-equation model of Spalart-Allmaras was implemented. A substantial effort was devoted to efficiency improvement. In order to accelerate convergence, a multigrid procedure was developed based on the agglomeration of the control volumes of the secondary grid for coarse grid computations. For the agglomeration a strategy is implemented which allows semi-coarsening for highly stretched cells. In order to efficiently detect detailed flow features, a grid adaptation algorithm for hybrid meshes based on local grid refinement was implemented. Residual and gradient based indicators are available and can be chosen via input parameter. Special refinement techniques were developed to ensure the robustness and efficiency of the adaptation algorithm in the case of complex configurations. Furthermore, the τ-Code is optimized for vector and parallel computers. Message passing parallelization is realized using the MPI-library. In a preprocessing module, the initial grid is partitioned in as many subdomains as processors used.

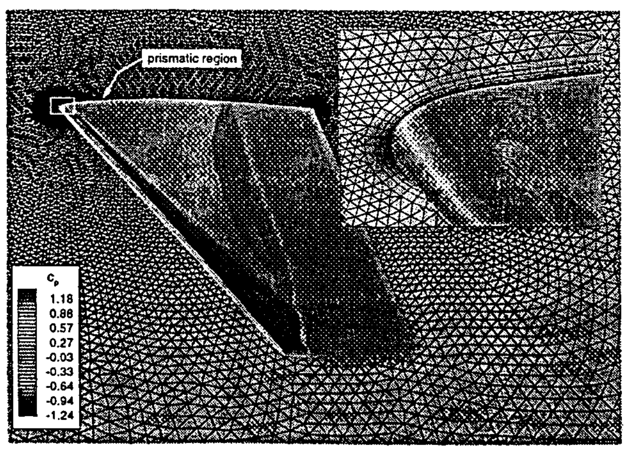


Fig. 31':

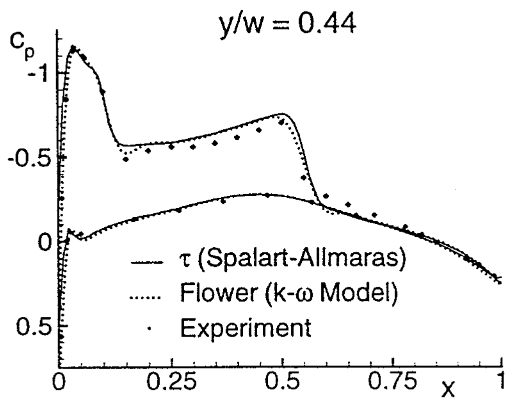


Fig. 31: Hybrid mesh, pressure contours and surface pressure distribution at 44% span section for flow around ONERA M6-wing at M_∞=0.84, α=3.06^0, Re=1.1x10^6, unstructured τ-Code

In Fig. 31 results for the simulation of the turbulent flow around the ONERA M6 wing on a hybrid mesh are presented. Although different turbulence model were used, the pressure distribution calculated on the hybrid mesh shows a good agreement with results obtained by the structured FLOWer code. Fig. 32 presents results for the DLR-F6 configuration mentioned earlier. The hybrid grids contains 1.5 million points. The comparison with the structured solver shows that a still finer mesh is needed to resolve all features of the flow with sufficient accuracy. Calculations on a finer grid are under way. Typical performance of the τ-code on a single processor on a NEC-SX4 is around 600 Mflops resulting in 10 hours computation time for a converged solution (around 300 multigrid cycles) for this test case. Employing parallelization on the NEC-SX4, a speed up factor of about 7 is obtained using 8 processors.

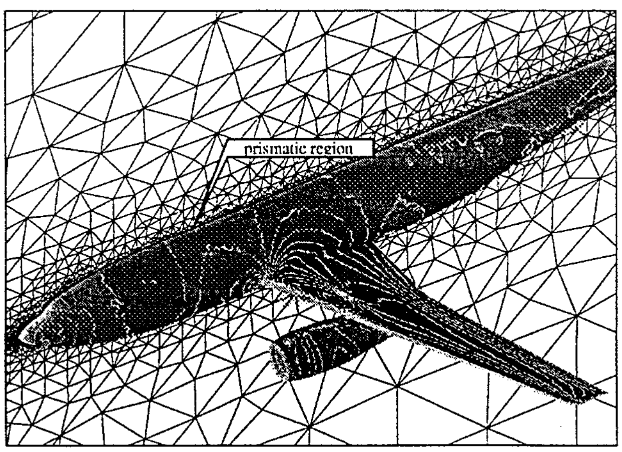


Fig. 32':

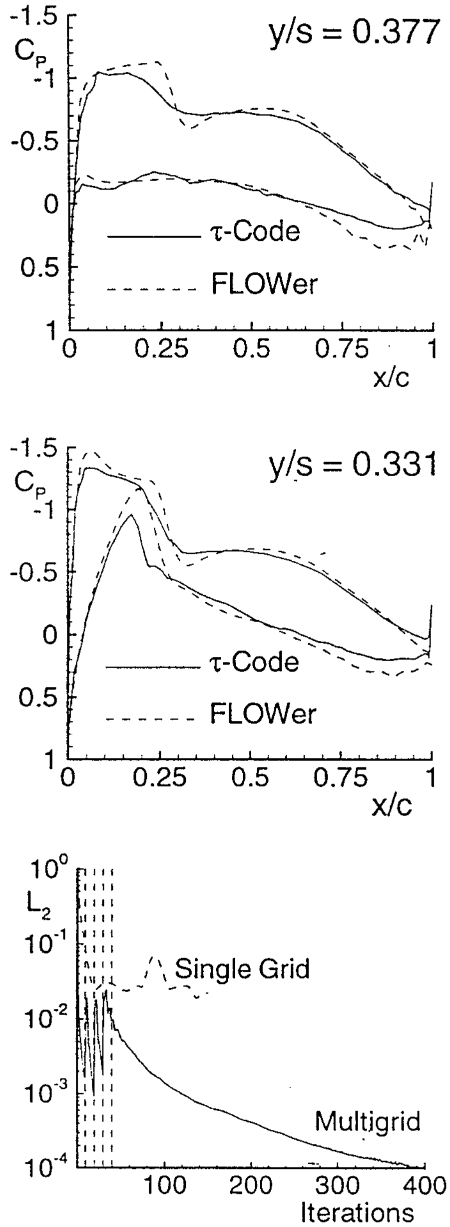


Fig. 32: Hybrid mesh, surface pressure contours and convergence history for turbulent flow around DLR-F6 configuration at M_∞=0.75, a=0.98^0, Re=3x10^6, unstructured τ-Code.

8. Aerodynamic Optimization System MEPO

Numerical shape optimization will play a strategic role for future aircraft designs. It offers the possibility of designing or improving aircraft components with respect to a prespecified figure of merit subject to geometrical and physical constraints. However, the extremely high computational expense of straightforward methodologies currently in use prohibits the application of numerical optimization for industrial relevant problems.

Within the framework of the MEGAFLOW project, the numerical optimization system MEPO (Multipurpose Environment for Parallel Optimization) is being developed[26]. In order to significantly reduce turn-around time, a high degree of parallelization has been realized and more sophisticated geometry models and optimization strategies are being developed and implemented. Based on preparatory work of the Technical University of Braunschweig [27], parallel computing is applied on three levels, as indicated in Fig. 33.

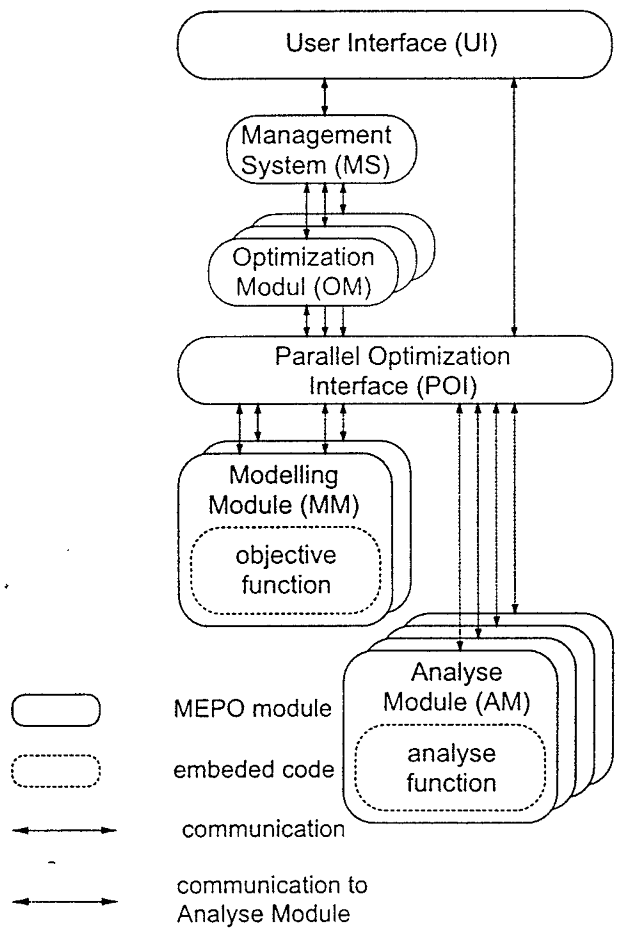


Fig. 33: Structure of the optimization system MEPO.

Fine grain parallelism is based upon the parallel execution of the block-structured flow solver FLOWer. A so-called medium grain parallelism is realized by splitting up the analysis tasks for multipoint optimizations. Finally, coarse grain parallelism is implemented through the inherent parallelism of some of the optimization strategies. MEPO is a highly modular software package. Its core is the Parallel Optimization Interface (POI). It performs dynamic load balancing for all parallel processes and thus ensures shortest overall computation time. Furthermore, POI controls the optimizer and the aerodynamic modelling module, which includes evaluation of the cost function as well as geometry modelling and grid generation. Data transfer and module management is based on the PVM (Parallel Virtual Machine) library.

Different optimizers including evolution strategies and gradient based strategies can be selected. A management system is being developed that combines various optimizers in a hybrid strategy. Several techniques for two- and three-dimensional geometry modelling, such as B-splines and NACA-functions, are integrated. The analysis code used is FLOWer in combination with MegaCads and a simple airfoil grid generator.

Aerodynamic optimizations carried out so far with MEPO cover inverse design and multipoint drag optimization for airfoils. Preliminary wing applications have also been carried out. For example the result of a 2D multipoint optimization is shown in Fig. 34. The objective of this optimization was to design an airfoil with minimum drag increase for increasing lift. The Mach number is $M_\infty = 0.765$ and the Reynolds number is $Re = 2.01 \times 10^6$. The design lift coefficients were $c_l = 0.2$, $c_l = 0{,}5$ and $c_l = 0.6$. A constraint was used for the airfoil thickness and a limit on the drag increase for $c_l = 0.65$ was prescribed[28]. The figure shows the development of the aerodynamic drag during optimization. In addition, the final geometry and pressure distribution in the main design point ($c_l = 0.5$) are shown. The drag reduction was achieved by reducing the strong shock, which appears for the NACA 2409 start geometry. The MEPO system has proved to be very stable in operation due to the module concept, which in addition makes the incorporation of new modules rather simple.

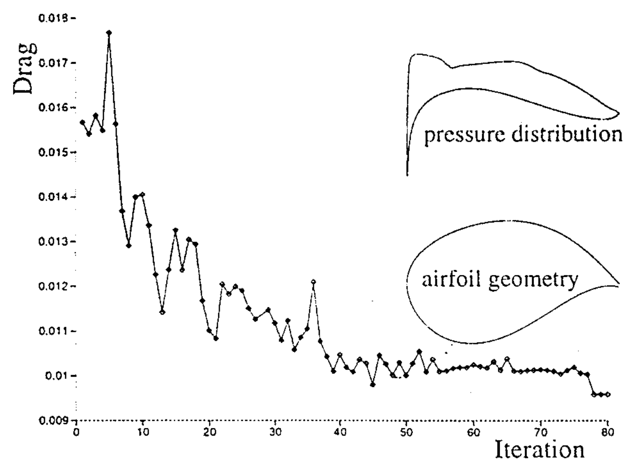


Fig. 34: Drag optimization for an airfoil at three design points $c_l = 0.2$, $c_l = 0{,}5$ and $c_l = 0.6$; $M = 0.765$, $Re = 2.01 \times 10^6$.

Some aerodynamic optimization methods are based on the calculation of the derivatives of the cost function with respect to the design variables. These gradients can be efficiently obtained by solution of the adjoint flow equation. DLR is actively developing and validating adjoint solvers in cooperation with Princeton University based upon the work of Jameson and Reuther[29].

9. Conclusions and Future Prospects

The MEGAFLOW initiative is the central German activity in the area of CFD development for application to aircraft design and involves the aircraft industry, government research centers (DLR, GMD) and several universities. Its main objective is the production of a dependable, effective and quality controlled program system for the aerodynamic simulation of complete aircraft. The project, sponsored by the German Ministry of Education, Science, Research and Technology, has opened the way to a new type of collaboration between multiple partners. Since all development and validation activities are carried out within a single software package, a direct link between research and industrial applications has been established. The time consuming transfer of results from research organizations into production codes is now avoided. Furthermore, industrial needs are being directly routed to developers in research institutes and at universities.

The MEGAFLOW system has reached a high level of quality and efficiency even for complex industrial problems. Furthermore, an extensive validation of the system was carried out. As a consequence, the MEGAFLOW system is being intensively used by Daimler-Benz Aerospace Airbus in the design processes for a new aircraft. Reviewing all results achieved so far from both the development and application side, MEGAFLOW is a very successful project.

However, since industry is still demanding for more accurate and faster simulation tools, further development is aimed at

- improvement of physical modelling through implementation of advanced turbulence and transition models,
- simplification and automation of structured grid generation through enhancement and utilization of the Chimera technique,
- significant reduction of problem turn-around time by development of a hybrid solver, which would allow the use of the algorithmic advantages of structured grids as well as the flexibility offered by unstructured meshes,
- extended validation,
- provision of interfaces for multidisciplinary simulations,
- enhancement of the aerodynamic optimization system for 3D configurations in viscous flows.

It is planned to tackle these objectives in a follow-on project.

10. Acknowledgments

The authors would like to thank all colleagues and participating institutions of the MEGAFLOW project who carried out the developments and obtained the results shown in this paper. Furthermore, the funding of the MEGAFLOW project through the German Government in the framework of the air transport research program is gratefully acknowledged.

11. References

1. Kroll, N: "National CFD Project MEGAFLOW-Status Report", Notes on Numerical Fluid Mechanics, Vol. 60, pp 15- 23, Vieweg-Verlag, Braunschweig, 1997.

2. Brodersen, O.; Hepperle, M.; Ronzheimer, A.; Rossow, C.-C.; Schöning, B.: "The Parametric Grid Generation System MegaCads", Proc. of the 5th Intern. Conference on Numerical Grid Generation in Computational Field Simulations, Ed.: B.K. Soni, J,F. Thompson, J. Häuser, P. Eisemann, NSF Mississippi, pp 353-362, 1996.

3. Brodersen, O.; Ronzheimer, A.; Ziegler, R.: Kunert, T.; Hepperle, M.; Wild, J.; Heinrich, R.: "Aerodynamic Applications using MegaCads", Proc. of the 6th Intern. Conference on Numerical Grid Generation in Computational Field Simulations, London, United Kingdom, 1998.

4. Wild, J.: "Voruntersuchungen zur numerischen Optimierung von Hchauftriebskonfigurationen mit Navier-Stokes-Verfahren", DLR-IB 129-97/19.

5. Kroll, N.; Radespiel, R.; Rossow, C.-C.: "Accurate and Efficient Flow Solver for 3D Applications on Structured Meshes", AGARD R-807, pp 4.1-4.59, 1995.

6. Eisfeld, B.; Bleecke, H.-M.; Kroll, N.; Ritzdorf, H.: "Parallelization of Block Structured Flow Solvers", AGARD R-807, pp 5.1-5.20, 1995.

7. Bleecke, H.M.; Heinrich, R.; Monsen, E.; Leicher, H.; Ritzdorf, H.: "FLOWer and CLIC-3D - A Portable Flow Solving System for Block-Structured Applications; Status and Benchmarks", Conference on CFD 1997, Manchester, 1997.

8. Turkel, E.; Radespiel, R.; Kroll, N.: "Assessment of Two Preconditioning Methods for Aerodynamic Problems", to appear in Computers and Fluids.

9. Radespiel, R.; Longo, J.M.A.; Brück, S.; Schwamborn, D.: "Efficient Numerical Simulation of Complex 3D Flows with Large Contrast", AGARD-CP-578, Paper No. 33, 1996.

10. Liou, M.S.; Steffen, C.J.: "A New Flux Splitting Scheme", Journal of Comp. Physics, Vol. 129, pp. 23-39, 1993.

11. Radespiel, R.; Kroll, N.: "Accurate Flux Vector Splitting for Shocks and Shear Layers", Journal of Comp. Physics, Vol. 121, pp. 66-78, 1995.

12. Rudnik, R.: "Untersuchungen der Leistungsfähigkeit von Zweigleichungs-Turbulenzmodellen bei Profil-

umströmungen", DLR-FB 97-49, 1997.

13. Monsen, E.; Rudnik, R.; Bleecke, H.: "Flexibility and Efficiency of a Transport-Equation Turbulence Model for Three-Dimensional Flow", Notes on Numerical Fluid Mechanics, Vol. 60, pp 237-244, Vieweg-Verlag, Braunschweig, 1997.

14. Jameson, A.: "Time Dependent Calculation Using Multigrid with Applications to Unsteady Flows past Airfols and Wings", AIAA 91-1596, 1991.

15. Heinrich, R.; Bleecke, H.: "Simulation of Unsteady Three Dimensional Viscous Flows Using a Dual Time Stepping Method", Notes on Numerical Fluid Mechanics, Vol. 60, pp 15- 23, Vieweg-Verlag, Braunschweig, 1997.

16. Britten, G; Ballmann, J: "Strömungs-Struktur-Wechselwirkung an einem elastischen Flügel in transsonischer, reibungsbehafteter Strömung", Paper accepted for the aeroelastic conference in Göttingen, 1998.

17. Rung, T., Thiele, F.: "Computational Modelling of Complex Boundary-Layer Flows", Proc. of the 9th Int. Symp. in Thermal-Fluids Engineering (ISTP-9), Singapore, pp. 321-326, 1996.

18. Rung, T., Thiele, F., Fu, S.: "On the Realizability of non-linear stress strain relationship for Reynolds-stress closures", Proc. of the 11th Symp. of Turbulent Shear Flows, Grenoble, pp. 13.1-13.6, 1997.

19. Haase, W., Chaput, E., Elsholz, E., Leschziner, M.A., Mueller, U.R. (Editors): "ECARP - European Comp. Aerod. Research Project: Validation of CFD Codes -and Assessment of Turbulence Models", Notes on Numerical Fluid Mechanics, Vol. 58, Vieweg-Verlag, Braunschweig, pp.327-346, 1997.

20. Delery, J.: "Investigation of strong shock turbulent boundary layer interaction in 2D flows with emphasis on turbulence phenomena", AIAA Paper 81-1245, 1981.

21. Wichmann, G: "Ein Verfahren zur Berechnung der Umströmung von Flügel-Rumpf-Konfigurationen unter Berücksichtigung der Tragflügelgrenzschicht", DGLR Bericht 90-06, S. 56-60, 1990.

22. Moir, J.R.M.: "Measurements on a Two-Dimensional Aerofoil with High-Lift Devices", AGARD-AR-303 Vol. II, pp A2.1-A2.12, 1994.

23. Rudnik, R; Ronzheimer, A.; Schenk, M.; Rossow, C.-C.: "Berechnung von 2- und 3-dimensionalen Hochauftriebsströmungen durch Lösung der Euler-Gleichungen", Tagungsband der DGLR Jahrestagung, Dresden, 1996.

24. Ewald, B.; Kreuzer, P.: "Alternative Drag Calculations from Off Body Flow Quantities Using the FLOWer Code", Notes on Numerical Fluid Mechanics, Vol. 60, Vieweg Braunschweig, 1997.

25. Gerhold, T.; Friedrich, O.; Evans, J.; Galle, M.; "Calculation of Complex Three-Dimensional Configurations Employing the DLR τ-Code", AIAA 97-0167, 1997.

26. Axmann, J.K.; Hadenfeld, M.; Frommann, O.: "Parallel Numerical Airplane Wing Design", Notes on Numerical Fluid Mechanics, Vol. 60, Vieweg-Verlag Braunschweig, 1997.

27. Axmann, J.K.: "Raumflugtechnische Optimierung mit adaptiven Evolutionsalgorithmen auf Parallelrechnern", Jahrbuch der DGLR, Band 1, 1994.

28. Frommann, O.: "Objective Function Construction for Multipoint Optimization Using Fuzzy Logic", EUROGEN97, Trieste, Italy, 1997.

29. Reuther, J.; Jameson, A.: "Aerodynamic Shape Otimization of Wing and Wing-Body Configurations Using Control Theroy", AIAA Paper 95-0123, 1995.