Available online at www.sciencedirect.com

ScienceDirect

Journal of Computational and Applied Mathematics 203 (2007) 397–411

JOURNAL OF COMPUTATIONAL AND APPLIED MATHEMATICS

www.elsevier.com/locate/cam

Flow simulation and shape optimization for aircraft design

Norbert Kroll[a,*], Nicolas R. Gauger[a], Joël Brezillon[a], Richard Dwight[a], Antonio Fazzolari[a], Daniel Vollmer[a], Klaus Becker[b], Holger Barnewitz[b], Volker Schulz[c], Subhendu Hazra[c]

[a] *German Aerospace Center (DLR), Lilienthalplatz 7, 38108 Braunschweig, Germany*
[b] *Airbus Deutschland GmbH, D-28183 Bremen, Germany*
[c] *University of Trier, Department of Mathematics, D-54286 Trier, Germany*

Received 1 December 2004; received in revised form 9 August 2005

Abstract

Within the framework of the German aerospace research program, the CFD project MEGADESIGN was initiated. The main goal of the project is the development of efficient numerical methods for shape design and optimization. In order to meet the requirements of industrial implementations a co-operative effort has been set up which involves the German aircraft industry, the DLR, several universities and some small enterprises specialized in numerical optimization. This paper outlines the planned activities within MEGADESIGN, the status at the beginning of the project and it presents some early results achieved in the project.

Keywords: Shape optimization; Adjoint; One-shot method; Industrial application

1. Introduction

Aerospace industry is increasingly relying on advanced numerical flow simulation tools in the early aircraft design phase. Today, computational fluid dynamics has matured to a point where it is widely accepted as an essential, complementary analysis tool to wind tunnel experiments and flight tests. Navier–Stokes methods have matured from specialized research techniques to practical engineering tools for a vast number of industrial problems on a routine basis [20]. Due to the high computational effort required for flow simulations around realistic 3D configurations, industrial computational fluid dynamics tools are rather used for analysis and assessment of given geometries than for shape design and optimization. However, within the next few years numerical shape optimization will play a strategic role for future aircraft design. It offers the possibility of designing or improving aircraft components with respect to a pre-specified figure of merit subject to geometrical and physical constraints. Consequently, worldwide a large effort is devoted to developing efficient optimization strategies for industrial aerodynamic aircraft design.

In Germany, the national project MEGADESIGN was initiated recently. Its main objective is to establish numerical shape optimization tools within industrial aircraft design processes. In order to meet the requirements of industrial

* Corresponding author.

E-mail addresses: norbert.kroll@dlr.de (N. Kroll), nicolas.gauger@dlr.de (N.R. Gauger), joel.brezillon@dlr.de (J. Brezillon), richard.dwight@dlr.de (R. Dwight), antonio.fazzolari@dlr.de (A. Fazzolari), daniel.vollmer@dlr.de (D. Vollmer), klaus.becker@airbus.com (K. Becker), holger.barnewitz@airbus.com (H. Barnewitz), volker.schulz@uni-trier.de (V. Schulz), hazra@uni-trier.de (S. Hazra).

0377-0427/$ - see front matter
doi:10.1016/j.cam.2006.04.012

implementations a co-operative effort has been set up which involves the German aircraft industry, the DLR, several universities and some small enterprises specialized in numerical optimization. The duration of the MEGADESIGN project is from September 2003 to May 2007. The MEGADESIGN project will not start from scratch; it is a continuation of the national initiative MEGAFLOW [12,13] in which dependable and efficient flow solvers for aerodynamic simulations were developed and validated. The MEGAFLOW software system includes the block-structured Navier–Stokes code FLOWer and the unstructured Navier–Stokes code TAU. Both codes have reached a high level of maturity and they are intensively used by the German aerospace industry, DLR and universities for a wide range of steady and unsteady applications.

The MEGADESIGN project deals with several key issues regarding the establishment of an efficient and flexible numerical optimization capability. These include suitable techniques for geometry parameterization, meshing and mesh movement methods, efficiency and accuracy improvements of the flow solvers as well as the flexible and efficient deterministic and stochastic based optimizers. It should be emphasized that the development activities are driven by industrial applications and that all partners are working with the same software components. New developments will be integrated into official software releases and will be made available to all partners as soon as possible. Consequently, the process of transferring latest research and technology results into codes used in industry will be considerably accelerated.

2. Improvement of flow solvers

The flow solvers used within MEGADESIGN are the block-structured code FLOWer and the unstructured hybrid code TAU. Both codes, mainly developed at DLR, solve the compressible, three dimensional Reynolds-averaged Navier–Stokes equations for rigid bodies in arbitrary motions. They routinely use one- or two-equation turbulence models and they employ state-of-the-art algorithms for space and time discretization. For shape optimization and simulation of aeroelastic effects both codes have been extended to allow geometry and mesh deformations. Due to a comprehensive validation effort, FLOWer and TAU have reached a high level of maturity and reliability for aeronautical applications.

Despite the advances in numerical algorithms, shape optimization for industrial problems still requires large computational resources. The efficiency of the flow solver has a large impact on the turn-around time of the optimization process, as it tends to be the most costly part of the chain by far. Within MEGADESIGN two major activities are devoted to improving the solver performance. On the one hand effective implicit time stepping schemes for hybrid meshes are being implemented and on the other hand adaptive capabilities are being developed utilizing the solution of the adjoint flow equations.

The basic hybrid TAU code currently uses an explicit Runge–Kutta multistage scheme in combination with an explicit residual smoothing and a multigrid method based on mesh agglomeration. The explicit character of the solution method severely restricts the CFL number which in turn often leads to slow convergence, especially in the case of large scale applications. In order to improve the performance and robustness of the TAU code, an approximately factored implicit scheme has been implemented [3]. The LU-SGS (lower–upper symmetric Gauss–Seidel [15]) scheme has been selected as a replacement for the Runge–Kutta scheme. In contrast to fully implicit schemes, this method has low memory requirements, low operation counts and it 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 multigrid 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.

Another way for decreasing calculation time is to use an adapted mesh which allows for—in theory—accurately solving the aerodynamic problem with the smallest number of mesh points possible. A mesh adaptation strategy was already developed and implemented in the TAU code during the MEGAFLOW project. This strategy is based on the gradient or differences of local flow quantities such as the total pressure loss as adaptation indicators. However, in aerodynamic optimization, global quantities, like the drag or the lift, are of main interest and a target oriented adaptation strategy may be more efficient. The accuracy of the aerodynamic coefficients can be estimated by using adjoint fields and a so called adjoint based mesh adaptation will be developed during the MEGADESIGN project. Therefore, one

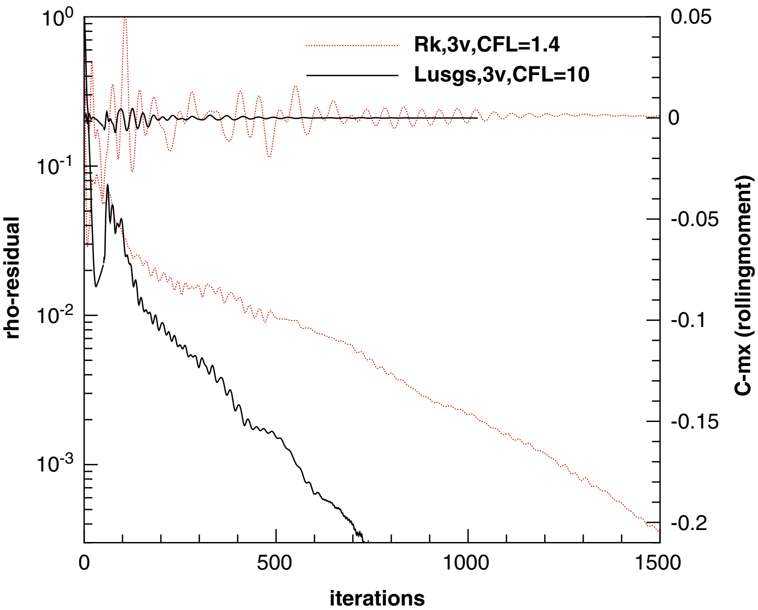

Fig. 1. Convergence behavior of the hybrid TAU code for calculations of viscous flow around a delta wing at $M_\infty = 0.5$, $\alpha = 9°$. Comparison of the baseline Runge–Kutta scheme (RK) and the implicit LU-SGS scheme.

key activity is the implementation of a continuous adjoint approach in the TAU code. While this implementation phase is in progress, experience is gained with the help of the block-structured FLOWer code which already offers an adjoint solver.

In the following, let H be the coarse and h the next finer global mesh size or level. Say u_H is the approximate solution of the Euler equations on the coarse mesh level, i.e. it holds that

$$R_H(u_H) = 0, \tag{1}$$

where $R_H(u_H)$ is the residual of the Euler equations on the coarse mesh level, and u_H is assumed to be a fully converged approximate solution. If then u_h^H is the approximate solution on the coarse mesh level extrapolated to the next finer mesh level h, it surely holds

$$R_h(u_h^H) \neq 0, \tag{2}$$

where $R_h(u_h^H)$ is the residual on the finer mesh level of the primal solution computed on the coarse level and extrapolated to the next finer level. Furthermore, let $I(u)$ be some aerodynamic coefficient. In this context the dual or adjoint solutions ψ_h and ψ_H are defined. ψ_h^H is then the approximate adjoint solution on the coarse mesh level extrapolated to the next finer level and with $\psi_h|u_h^H$ we label the approximate adjoint solution on the finer mesh level computed with the approximate primal solution on the coarse mesh level extrapolated to the next finer mesh level as input.

As shown in [19],

$$I(u_h^H) - I(u_h) \approx \psi_h|u_h^H \cdot R_h(u_h^H) \tag{3}$$

is a first-order approximation of the error for the computed coefficient $I(u_h^H)$, compared to its value $I(u_h)$ on the next finer mesh level. If we can estimate this error in a numerically efficient way, we can apply it as correction to the coefficient computed on the coarse level. However, the numerical cost for the evaluation of $\psi_h|u_h^H$ is comparable to that of $I(u_h)$. Therefore, this term is rearranged as

$$\begin{aligned} \psi_h|u_h^H \cdot R_h(u_h^H) &= \psi_h^H \cdot R_h(u_h^H) + (\psi|u_h^H - \psi_h^H) \cdot R_h(u_h^H) \\ &\approx \psi_h^H \cdot R_h(u_h^H) \end{aligned} \tag{4}$$

Table 1
Cell numbers for different mesh levels (NACA 0012, $M_\infty = 0.63$ and $\alpha = 2.0°$)

Level 1	$768 \times 256 = 196{,}608$ cells
Level 2	$384 \times 128 = 49{,}152$ cells
Level 3	$192 \times 64 = 12{,}288$ cells
Level 4	$96 \times 32 = 3072$ cells

Table 2
Drag coefficients and adjoint based error estimators for different mesh levels on the NACA 0012 airfoil at $M_\infty = 0.63$ and $\alpha = 2.0°$ (level 4–3 means $H = 4$ and $h = 3$)

Level	4–3	3–2	2–1
$C_D(u_h^H)$	0.0036637	0.0009747	0.0004667
$C_D(u_h)$	0.0009919	0.0004889	0.0004019
$C_D(u_h^H) - C_D(u_h)$	0.0026718	0.0004858	0.0000648
$\psi_h \vert u_h^H \cdot R_h(u_h^H)$	0.0018453	0.0004051	0.0000788
$\psi_h^H \cdot R_h(u_h^H)$	0.0009619	0.0002830	0.0000505

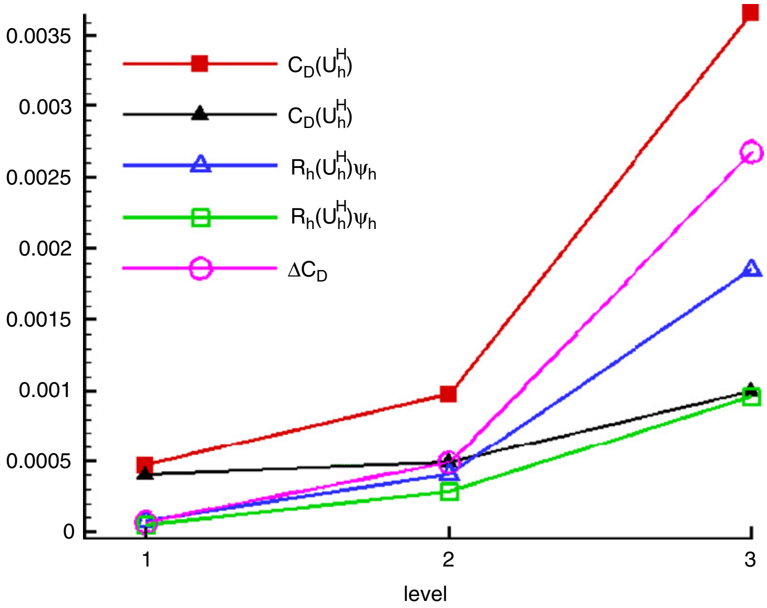

Fig. 2. Drag coefficients and adjoint based error estimators for different mesh levels (NACA 0012, $M_\infty = 0.63$ and $\alpha = 2.0°$).

and the computable correction—$\psi_h^H \cdot R_h(u_h^H)$—can be efficiently calculated since it is computed on the coarse mesh level. The question which remains is, how good is the computable correction—$(\psi|u_h^H - \psi_h^H) \cdot R_h(u_h^H)$—and how does it perform as an indicator for mesh adaptation.

As a first test case the inviscid subsonic flow around the NACA0012 airfoil at $M_\infty = 0.63$ and $\alpha = 2.0°$ is considered. In this case the theoretical drag coefficient C_D is zero since the flow is inviscid and has no shocks. For numerical solutions this cannot be reached due to the discretization error and the numerical dissipation introduced by the numerical method. However, the finer the mesh is, the closer the approximate value for C_D should be to zero.

Table 1 shows the number of cells for four different mesh levels for this NACA0012 case. Here, one jump in level means one global mesh refinement. With this set of refined meshes a mesh consistency study for the adjoint based error estimators can be performed. Results of this investigation are given in Table 2 and Fig. 2.

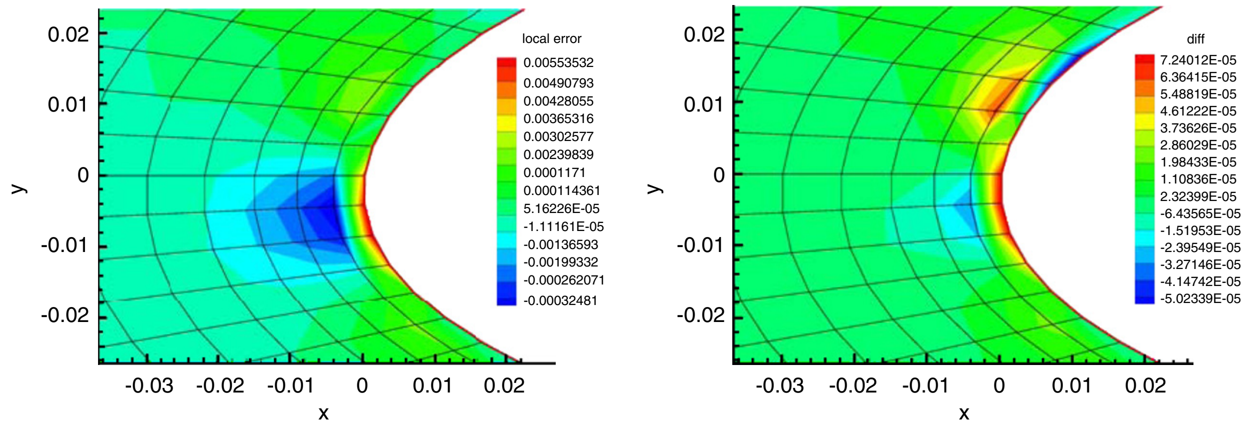

Fig. 3. Local adjoint based “computable” error indicators at the nose (left) and its error (right) (NACA 0012, $M_\infty = 0.63$ and $\alpha = 2.0°$).

It can be seen that the finer the mesh level H is, the better the computable correction $\psi_h^H \cdot R_h(u_h^H)$ and the difference $C_\mathrm{D}(u_h^H) - C_\mathrm{D}(u_h)$ agree. Also on coarse mesh levels H, the adjoint based error estimator $\psi_h^H \cdot R_h(u_h^H)$ gives the right trend and can be used to obtain an improved (corrected) drag coefficient $C_\mathrm{D}(u_h^H) - \psi_h^H \cdot R_h(u_h^H)$.

Moreover, Fig. 3 indicates that the term $\psi_h^H \cdot R_h(u_h^H)$ can be used as an indicator for local mesh adaptation. On the left side the term is plotted in the nose region of the NACA0012 airfoil while the right figure shows the term

$$\mathtt{diff} := \psi_h^H \cdot R_h(u_h^H) - \psi_h|u_h^H \cdot R_h(u_h^H). \tag{5}$$

It can be seen that the values of this term are one order of magnitude smaller than the values of the term $\psi_h^H \cdot R_h(u_h^H)$. Thus the term $\psi_h^H \cdot R_h(u_h^H)$ can be used as a local adaptation indicator. Currently this is being investigated in more detail using the block mesh adaptation capability of FLOWer [18].

3. Numerical optimization techniques

3.1. Adjoint flow solver

Because detailed aerodynamic shape optimizations still suffer from high computational costs, efficient optimization strategies are required. Regarding the deterministic methods, the adjoint approach is seen as a promising alternative to the classical finite difference approach [10,11]. Accordingly, within the MEGAFLOW project an adjoint solver following the continuous adjoint formulation has been developed and widely validated for the block-structured flow solver FLOWer [6,1]. The adjoint solver, which was implemented by hand, can deal with the boundary conditions for drag, lift and pitching-moment sensitivities. The capabilities of the adjoint approach and its application are shown in Section 4. The adjoint option of the FLOWer code is validated for several 2D as well as 3D optimization problems controlled by the (adjoint) Euler equations. Within MEGADESIGN the robustness and efficiency of the adjoint solver will be improved, especially for the Navier–Stokes equations. As a first step the multigrid capability, an important ingredient for the FLOWer code, has been made available for the adjoint mode as well (see Fig. 4).

In case of Navier–Stokes applications, up to now the turbulence model is frozen in the adjoint mode. It is planned to make use of Algorithmic Differentiation (AD) in order to create adjoint turbulence models, which will then be linked to the hand coded adjoint solver.

Furthermore, it is planned to transfer the adjoint solver, implemented in FLOWer, to the unstructured Navier–Stokes solver TAU. Here, the implementation work has already begun.

3.2. Adjoint solver for fluid/structure coupling

The use of single disciplinary optimizations in case of a multidisciplinary optimization problem is not only inefficient but in some cases has been shown to lead to wrong, non-optimal designs [16].

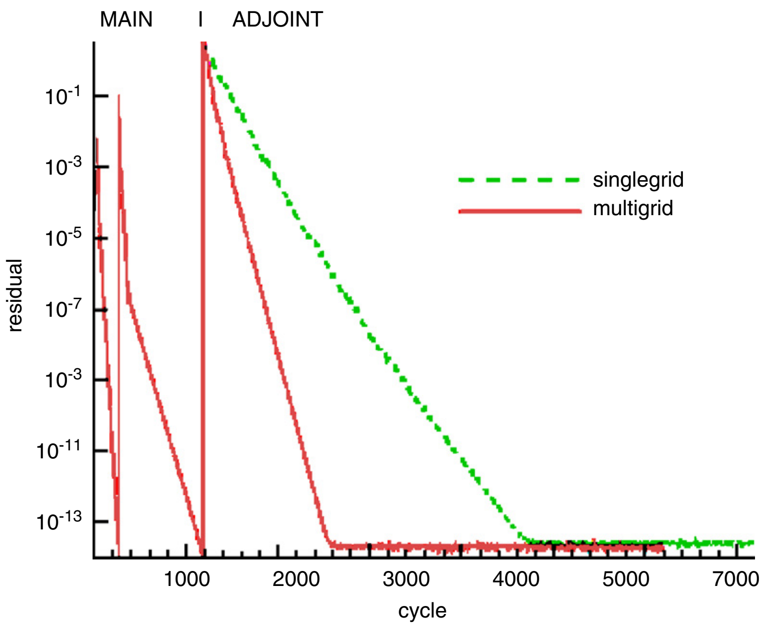

Fig. 4. Convergence history of the aerodynamic (MAIN) and adjoint flow solver FLOWer.

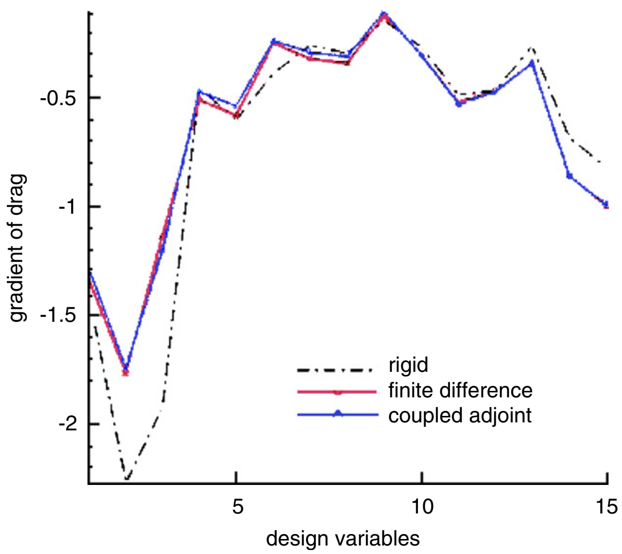

Fig. 5. Validation of the aero-structural coupled adjoint with finite differences (AMP wing, $M_\infty = 0.78$ and $\alpha = 2.83°$).

Although multidisciplinary optimization is possible with classical approaches for sensitivity evaluation by means of finite differences, this method is extremely expensive in terms of calculation time, requiring the reiterated solution of the coupled problem for every design variable. A new approach that allows the evaluation of the gradient with low computational cost takes advantage of the adjoint formulation of the multidisciplinary optimization problem [16,17].

Therefore, the FLOWer adjoint option has been coupled with the structure solver MSC Nastran for an efficient coupled aero-structure adjoint solver. This approach, its implementation and validation are described in detail in [4]. For the validation of the resulting coupled aero-structure adjoint approach the aero-structure model of the AMP wing [9] has been used. The cost function is the reduction of drag. Fig. 5 shows the gradient of the drag with respect to the aerodynamic design parameters. If the aero-structure coupling is taken into account, a good agreement between the

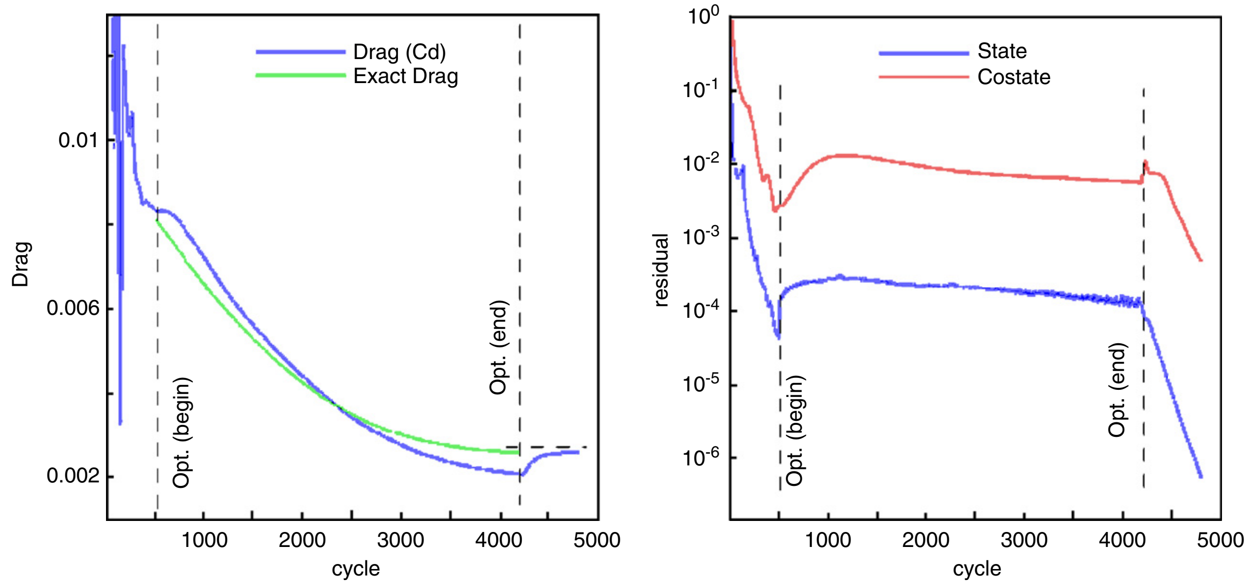

Fig. 6. Convergence history of design (inexact/exact drag), state and costate (RAE 2822, $M_\infty = 0.73$ and $\alpha = 2.0°$).

finite difference and the coupled adjoint approach is achieved. However, there is a large discrepancy if the wing is taken as a rigid body leading to a non-physical optimum.

3.3. Optimization methods

One-shot methods for the fast numerical optimization of shape designs will be developed using simultaneous pseudo-timestepping. The algorithmic approach is based on an embedding of optimization strategies within the iterations of the respective flow solver. A continuous reduced SQP method is developed to solve the optimization problem in one joint pseudo-timestepping iteration for all variables (flow state, adjoint and wing design variables) [7,8]. In this way we look for the steady states of the pseudo-time embedded non-stationary system of state, costate (or adjoint state) and design equations. The preconditioner used corresponds to Karush–Kuhn–Tucker matrices, which are used in an approximate reduced SQP method.

A first demonstration of the capability of the one shot method is given for the drag reduction of the RAE 2822 airfoil in inviscid flow with $M_\infty = 0.73$ and $\alpha = 2.0°$.

Fig. 6 (right) presents the convergence history of the optimization iterations. The optimization is started with the initial solution of the state and costate equations obtained after 500 steps with Runge–Kutta time integration. The convergence of the optimization is achieved after 3700 optimization iterations. After convergence is achieved for optimization, we perform another 600 time iterations for state and costate solvers to reduce the residual of these two variables further to get more accurate values of surface pressure and force coefficients. Fig. 6 (left) shows the inexact and exact drag reduction during the optimization iterations. Inexact here means that the drag is evaluated for the less converged state and costate variables used in the design loop. Afterwards, on the trace of modified shapes generated during the one-shot approach, the drag was recomputed up to an accuracy of 7 digits and compared with the inexact one. The final drag reduction after the optimization is about 68% and the shock completely vanished (see Fig. 7) as expected for inviscid cases.

Fig. 7 presents the comparison of the initial and final surface pressure distributions achieved with the one-shot approach (present) and with the conventional gradient based adjoint approach (steepest descent). Altogether, the numerical cost of the one-shot optimization is of the magnitude of just four flow simulations, which is a dramatic reduction in computation time compared with the conventional approach.

Besides gradient based optimization strategies, direct search methods such as genetic algorithms (GA) are widely used for aerodynamic shape optimization. GA are capable of finding a global optimum and are well-suited for problems

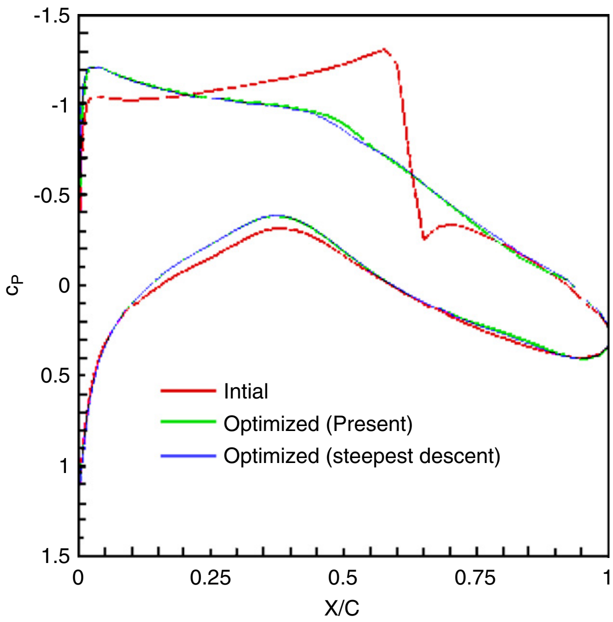

Fig. 7. Initial and optimized pressure distribution (RAE 2822, $M_{\infty} = 0.73$ and $\alpha = 2.0°$).

with multiple objectives but at the cost of numerous flow evaluations, up to 1000 in some cases. One way to improve the efficiency would be to couple the GA with the adjoint method. Within the MEGADESIGN project, such hybrid methods will be investigated on relevant configurations and then integrated in the modeFRONTIER framework specialized for optimization based on GA.

3.4. Optimization framework

In parallel to the development of optimization strategies, suitable procedures for aerodynamic as well as for multi-disciplinary optimizations are also under investigation within the MEGADESIGN project.

During the MEGAFLOW project the optimization framework SynapsPointerPro [5] has been investigated for solving aerodynamic shape optimization. It was demonstrated that this environment offers an efficient and flexible procedure to connect all elementary modules necessary to associate a given geometry to an aerodynamic state, and to carry this information over to the optimizer [1]. Therefore this framework has been selected as the central optimization environment at Airbus, DLR and some of the universities involved in the project. During MEGADESIGN the Synaps company will further improve its capabilities. The framework will be extended to treat realistic multi-disciplinary optimization. Innovative optimization strategies will be developed and integrated, among other the fuzzy logic for defining reliable goal functions, especially in the case of multi-disciplinary optimizations.

Furthermore, methods for guiding a numerical design in an efficient way are also under investigation. A standardization of methods and processes will be sought and a best-practice method will be developed and integrated into the framework. The goal of this is to facilitate the use of numerical optimization in industrial contexts.

All these activities will lead to a better use of the framework and to a drastic reduction of turn-around time associated with the trial-and-error method occurring very often in numerical optimization.

4. Applications

The development activities in the MEGADESIGN project are driven by industrial applications. First results are already available.

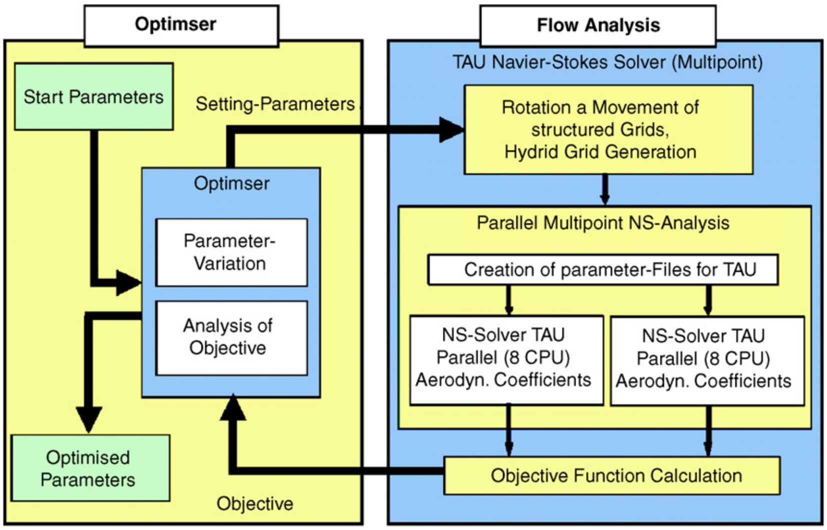

Fig. 8. Optimization processing within SynapsPointerPro system.

4.1. High lift applications

As a major interest of Airbus, high lift system optimization is a task for numerical optimization within aerodynamics. An attempt has been made to improve setting optimization for a typical 2D case, and a considerable improvement can be noted compared to a previous baseline experiment. The key to this improvement was an enhanced strategy to determine maximum lift which made the search process for an optimum of the cost function much more stable. As a consequence, this kind of optimization is already considered as being mature enough to help design engineers in finding their design solutions.

The two-point design problem is a test case for a 2D three-element high lift landing configuration. Design parameters are the slat and flap angle with gap and overlap for slat and flap. Geometry variations are not taken into account. The objective function is a linear combination of the lift coefficients at two different flow conditions:

$$\mathrm{OBJ} = C_L(\alpha_1, M_1) + C_L^{\mathrm{MAX}}(\alpha_2, M_2). \tag{6}$$

C_L denotes the lift coefficient, α_i and M_i are the angles of attack and Mach numbers, respectively.

The two design conditions (n = 1, 2) are specified as follows. The main design point is at $\alpha = 10°$ incidence, $M_1 = 0.2$ and $Re = 8.0 \times 10^6$. The flow is assumed to be fully turbulent on lower and upper surfaces. The off-design point is at $\alpha_2 = \alpha(C_L^{\mathrm{MAX}})$ incidence, $M_2 = 0.18$ and $Re = 8.0 \times 10^6$. The flow is also fully turbulent on lower and upper surfaces. The angle α for maximum lift has to be determined. As constraint the value of the maximum lift coefficient has to be maintained or may be greater than that for the original setting configuration:

$$C_L^{\mathrm{MAX}} \geqslant C_{L,\mathrm{REFERENCE}}^{\mathrm{MAX}}. \tag{7}$$

Six design parameters are used for the setting optimization. These are the coordinates of the slat and flap positions and the slat and flap angles.

As for all optimization tasks, Airbus Germany is using the SynapsPointerPro optimization framework with the gradient free Downhill Simplex Method to find the optimum. Fig. 8 gives an overview of the processing within this system.

For the high lift case treated here, the flow analysis tool is the unstructured Reynolds-Averaged Navier–Stokes solver TAU with the Spalart-Allmaras one-equation turbulence model. A mixed structured/unstructured grid of around 30,000 nodes with 21,000 triangles and 19,000 quadrilaterals is generated for each new configuration. Fig. 9 gives an impression of a typical hybrid grid for this case.

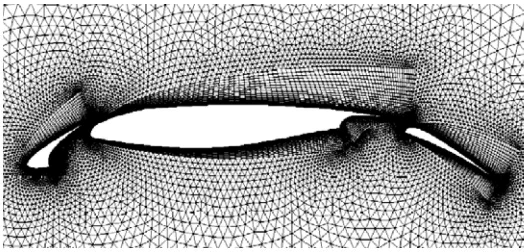

Fig. 9. Hybrid grid around three-element configuration.

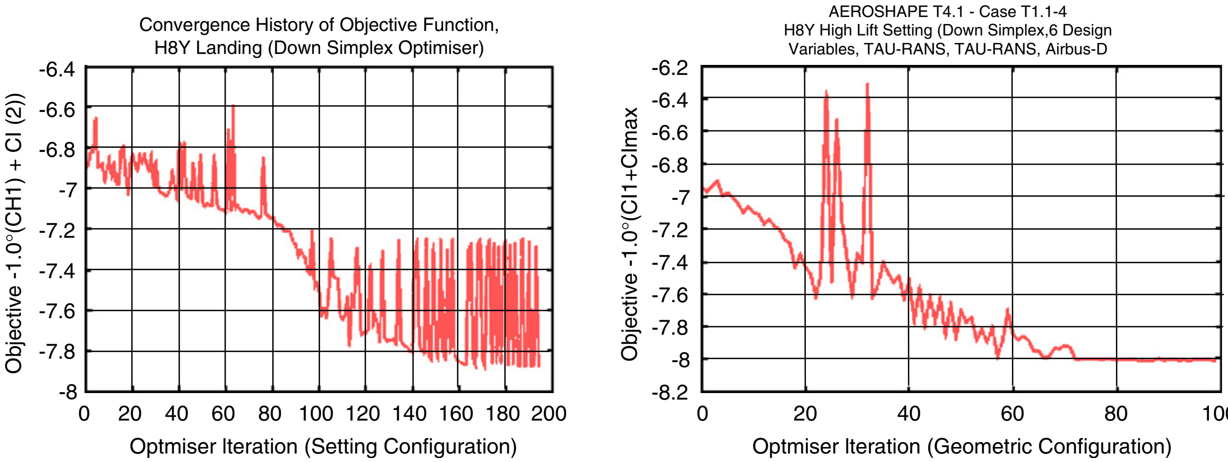

Fig. 10. Convergence of optimization process—baseline (left) and improved method (right).

The specific problem with this optimization task is that the objective function as well as the constraint evaluation requires the determination of $C_{\mathrm{L}}^{\mathrm{MAX}}$ which needs several flow calculations. Consequently, this task is quite costly and it requires an efficient strategy.

For the baseline case, the Downhill Simplex optimizer required about 180 test settings to reach the optimum. Now, with an improved determination of $C_{\mathrm{L}}^{\mathrm{MAX}}$ and a more robust and efficient TAU solver the optimized result seems to be already available after about 75 steps (see Fig. 10).

The final optimum setting that has been found by the optimizer yields some interesting features. Fig. 11 shows that for the optimum the load on the main airfoil and the flap increased considerably while the slat load has been reduced. Fig. 12 shows the calculated flow field at both design conditions for the reference and the optimized geometry. For design point 2, it is obvious that the reference setting does not work properly. Irregularity of the slat flow disturbs the whole flow field and causes break-down of lift immediately there after. This is quite different from the optimized setting. The influence of the slat flow on the downstream elements is much smaller. This is caused by the much bigger gap between slat and main airfoil. A much higher maximum lift coefficient could be achieved this way.

The application of high lift setting optimization is very interesting for high lift design within aircraft development projects. Previously many settings have usually been measured in order to find the optimum for take-off and landing. However, this had always been a discrete search which took place in the low Reynolds number regime of the wind tunnel. With the help of the numerical optimization approach it has now become possible to encircle the optimum much more precisely prior to any measurement. This saves a lot of cost and time. Additionally the same kind of optimization is done for the high Reynolds number case. Design engineers get much more insight into the topography of the optimum and its dependence on the Reynolds number.

Another goal of the project is the aerodynamic optimization of high-lift devices using the adjoint approach for computing the gradient of the cost function. While the implementation of the adjoint equation based on the Navier–Stokes

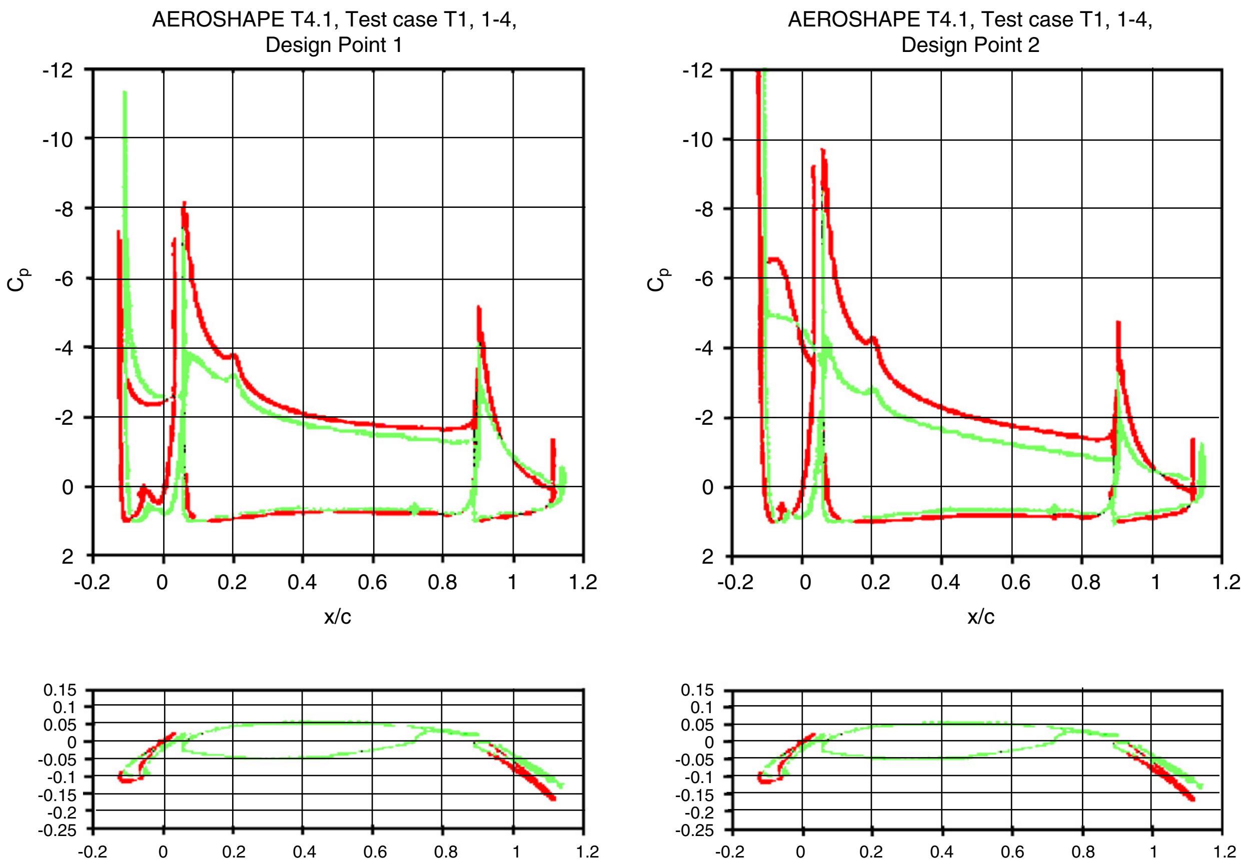

Fig. 11. Pressure distribution and geometry at design points DP1 (left) and DP2 (right), optimum (red) versus reference setting (green).

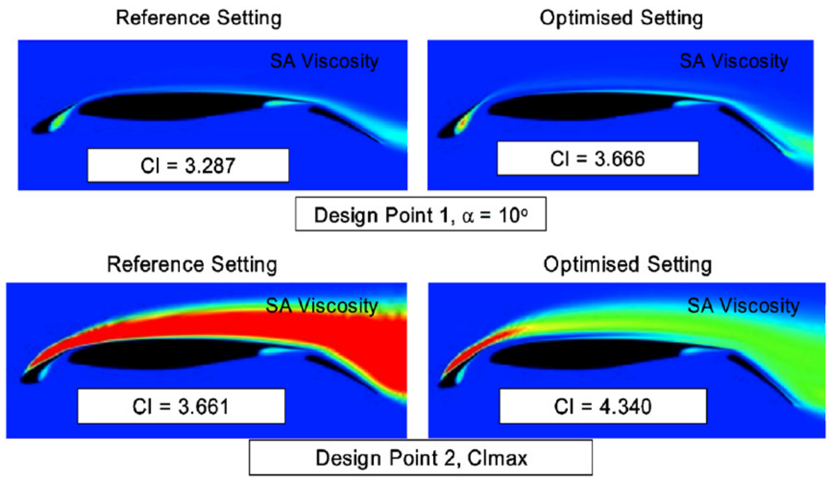

Fig. 12. Viscosity plots—baseline method result (left) compared to result of improved method (right).

equations is in progress, a preliminary step is devoted to checking the feasibility and the efficiency of the gradient based approach for high lift applications. The gradient is computed using finite differences.

The case treated here is the maximum lift optimization of the L1T2 high lift configuration. While the flap remains unchanged during the optimization, the maximum lift is improved by moving the slat in the two coordinate directions

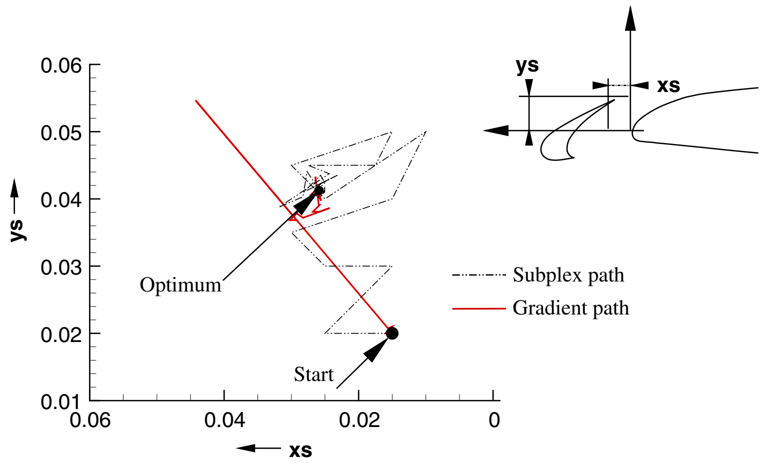

Fig. 13. Paths of the simplex and gradient based methods through the design space.

(x's, y's). The optimal configuration provided by the gradient based approach is compared to the optimal solution obtained with the simplex strategy, which is considered as the reference in industry. While the two strategies are close together, the gradient based approach needs about 25% less flow computations to reach the optimum (Fig. 13). Replacing the finite differences strategy by the adjoint approach to compute the gradient would lead to 30% less iterations compared to the simplex strategy.

Of course, even better efficiency is expected with the adjoint approach for an optimization problem with more than two design variables. However, it can be pointed out that the simplex strategy is more robust to noise resulting from the degree of accuracy of the aerodynamic flow solution than the gradient based method. This indicates that in the case of gradient based optimization well converged solutions are required which may be quite costly for more complex applications. Furthermore, it is well known that the gradient based approach is only able to find the local minimum surrounding the starting configuration. These two aspects are under investigation in MEGADESIGN for more complex high-lift configurations. One promising way seems to be the coupling of both optimization strategies in a suitable way.

4.2. Supersonic transport aircraft

The aim of the exercise is to demonstrate the capability of the adjoint approach to handle many design parameters with low cost for the optimization of a supersonic transport wing/body configuration.

The baseline geometry is based on the EUROSUP geometry [14], which is a supersonic commercial aircraft of 252 seats capacity, designed for 5500 nautical miles range with a supersonic cruise flight at Mach number $M_\infty = 2.0$. In the framework of the EUROSUP research programme a number of geometry modifications were introduced in order to ease CFD grid generation and analysis work (see Fig. 14). The optimization goal is to minimize the drag at a fixed lift coefficient of $C_L = 0.12$. The fuselage incidence is allowed to change in order to maintain the lift coefficient but it should not be greater than 4° to the onset flow.

In order to use the full potential of the adjoint technique, no specific restrictions are set to define the parameterization. To change the twist 74 design variables were used, the thickness and the camber line at specific wing sections and 10 more design variables allow to change the radial distribution of the fuselage. A minimum allowable value of the fuselage radius and a minimum wing thickness law were imposed in order to prevent unrealistic aircraft. After geometrical modifications, the intersection of wing and fuselage is recalculated automatically by the DLR inhouse grid generator MegaCads [2] for each new configuration. The mesh generation is performed by MegaCads running in batch mode and produces a structured mono-block type with about 230,000 cells. This mesh has been designed to have three multigrid levels.

At $M_\infty = 2.0$, the main aerodynamic effects are well predicted using the Euler equations. Therefore, the aerodynamic states are computed by FLOWer running in Euler mode. The baseline method employs a central space discretization with artificial viscosity and convergence is accelerated using local time stepping, implicit residual smoothing and multigrid techniques.

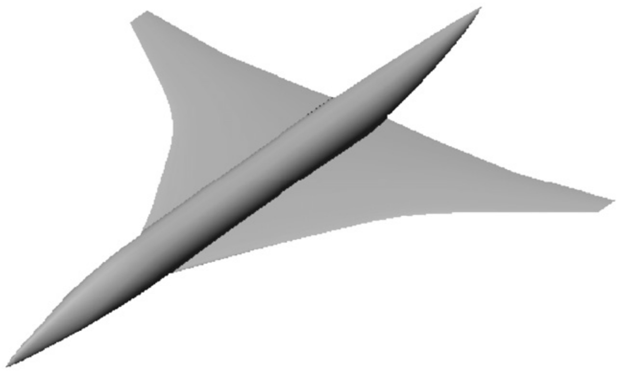

Fig. 14. Geometry of the supersonic transport aircraft.

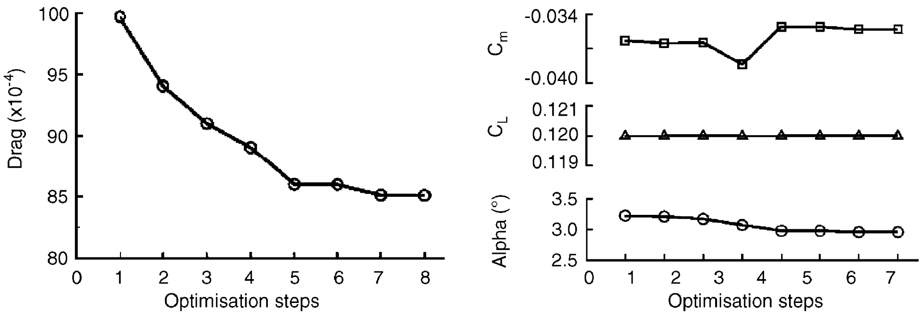

Fig. 15. Evolution of the SCT performance during the optimization.

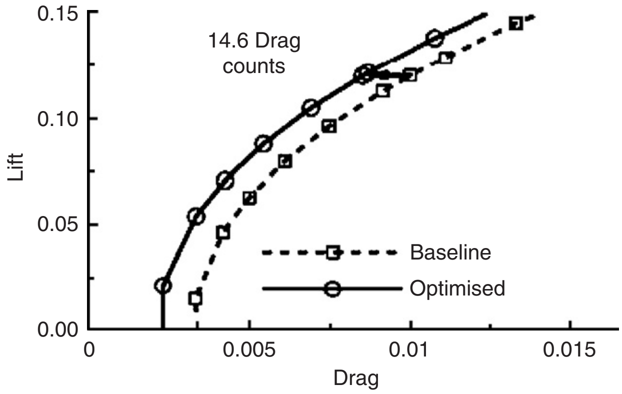

Fig. 16. Polar for the baseline and optimized geometry ($M_\infty = 2.0$).

The constraint on the lift is handled using the target lift mode available in FLOWer which automatically adjusts the angle of attack to reach the desired lift. In the present optimization problem, the unique aerodynamic constraint is the lift, which is handled directly by FLOWer and the geometrical constraints are automatically fulfilled during the parameterization. Therefore, the optimizer deals with an unconstrained problem and a second-order gradient based approach called "variable metric methods" using the Broydon–Fletcher–Goldfarb–Shanno (BFGS) algorithm is used. The required gradients are accurately computed using the adjoint approach.

Fig. 15 shows the evolution of the drag coefficient during the optimization, where an optimization step includes the evaluation of the gradient and the line search. About 8 optimization steps were necessary to achieve the optimum, which represents 54 aerodynamic computations and 8 adjoint flow evaluations: this approach would be more than 11 times faster than using brute force optimization based on finite differences. The optimum configuration has 14.6 less drag counts than the baseline geometry.

It can be seen on the right of Fig. 15 that FLOWer keeps the lift constant during the complete optimization and the angle of attack decreases slightly by about 0.3°. The pitching moment, which is only monitored, decreases by about 2.8%.

At this point, it is interesting to analyze the evolution of the performance around the design point. Fig. 16 shows the polar both for the baseline and the improved geometries. It can clearly be seen that there is an almost constant reduction of the drag for the optimized geometry, not only located at the main design point ($C_L = 0.12$) but all over the polar.

5. Conclusion

In Germany, the co-operate effort MEGADESIGN between aeronautical industry, research institutions and specialized small enterprises has been set up to establish an improved numerical shape optimization capability. The project deals with several critical aspects associated with design optimization. The development activities are driven by industrial applications. Focus is put on geometry parameterization, meshing aspects, improvement of flow solver performance and accuracy, innovative optimization strategies and a flexible optimization environment. The project started at the end of 2003 and will last until mid-2007. It is a continuation of the German MEGAFLOW project, in which the flow solvers FLOWer and TAU have been developed. First encouraging results of the MEGADESIGN project have been presented.

Acknowledgments

The partial funding of the MEGADESIGN project through the German Government in the framework of the aeronautical research program is gratefully acknowledged.

References

[1] J. Brezillon, N.R. Gauger, 2D and 3D aerodynamic shape optimization using adjoint approach, Aerosp. Sci. Technol. 8/8 (2004) 715–727.
[2] O. Brodersen, M. Hepperle, A. Ronzheimer, C.C. Rossow, B. Schoening, The parametric grid generation system MegaCads, numerical grid generation in computational field simulations, Proceedings of the 5th International Conference, Mississippi State University, Mississippi State, 1–5 April 1996, pp. 353–362.
[3] R. Dwight, A comparison of implicit algorithms for the Navier–Stokes equations on unstructured grids, Proceedings of the ICCFD Conference, Toronto, Canada, 2004.
[4] A. Fazzolari, N.R. Gauger, J. Brezillon, Sensitivity evaluation for efficient aerodynamic shape optimization with aeroelastic constraints, Proceedings of ECCOMAS 2004, Jyväskylä, Finland, 2004.
[5] O. Frommann, SynapsPointerPro V2.50, Synaps Ingenieur-Gesellschaft mbH, Bremen, Germany, 2002.
[6] N.R. Gauger, Das Adjungiertenverfahren in der aerodynamischen Formopti-mierung, DLR-Report No. DLR-FB–2003-05 (ISSN 1434-8454), 2003. http://opus.tu-bs.de/opus/volltexte/2004/550/
[7] S.B. Hazra, V. Schulz, Simultaneous pseudo-timestepping for PDE-model based optimization problems, Bit Numer. Math. 44 (3) (2004) 457–472.
[8] S.B. Hazra, V. Schulz, J. Brezillon, N.R. Gauger, Aerodynamic shape optimization using simultaneous pseudo-timestepping, J. Comput. Phys. 204 (1) (2005) 46–64.
[9] R. Heinrich, R. Ahrem, G. Günther, W. Krüger, J. Neumann, Aero-elastische Simulation unter Verwendung des AMANDA Simulationssystems, VDI-Berichte Nr. 1682 (ISBN 3-18-091682-6), 2002.
[10] A. Jameson, Aerodynamic design via control theory, J. Sci. Comput. 3 (1988) 233–260.
[11] A. Jameson, L. Martinelli, N.A. Pierce, Optimum aerodynamic design using the Navier–Stokes equations, Theoret. Comput. Fluid Dynamics 10 (1998) 213–237.
[12] N. Kroll, C.C. Rossow, K. Becker, F. Thiele, The MEGAFLOW project, Aerosp. Sci. Technol. 4 (2000) 223–237.
[13] N. Kroll, C.C. Rossow, D. Schwamborn, K. Becker, G. Heller, MEGAFLOW—A Numerical Flow Simulation Tool for Transport Aircraft Design, ICAS-2002-1105.20, 2002.
[14] D.A. Lovell, Aerodynamic research to support a second generation supersonic transport aircraft—the EUROSUP Project, Eccomas 98, 1998.
[15] H. Luo, J.D. Baum, R. Löhner, A fast matrix-free implicit method for compressible flows on unstructured grids, J. Comput. Phys. 146 (1998) 664–690.
[16] J.R. Martins, J.J. Alonso, J.J. Reuther, High-fidelity aero-structural design optimization of a supersonic business jet, AIAA 2002-1483, 2002.

[17] J.R. Martins, J.J. Alonso, J.J. Reuther, Complete configuration aero-structural optimization using a coupled sensitivity analysis method, AIAA 2002-5402, 2002.
[18] M. Rudolph, Entwicklung eines Programms zur automatischen Adaption blockstrukturierter Netze, DLR Report No. IB-124-2003/03, 2003.
[19] D. Venditti, D. Darmofal, Grid adaptation for functional outputs: application to two-dimensional inviscid flows, J. Comput. Phys. 176 (2002) 40–69.
[20] J.B. Vos, A.W. Rizzi, D. Darracq, E.H. Hirschel, Navier–Stokes solvers in European aircraft industry, Progr. Aerosp. Sci. 38 (2002) 601–697.