Aerodynamic Applications using MegaCads

O. Brodersen
A. Ronzheimer
R. Ziegler
T. Kunert
J. Wild
M. Hepperle

DLR, Institute of Design Aerodynamics
Lilienthalplatz 7
D-38108 Braunschweig, Germany

E-mail: megacads@dlr.de

Abstract

The application of the interactive grid generation system MegaCads for transport aircraft configurations in cruise and high-lift flight conditions is presented. Navier-Stokes and Euler calculations are compared to windtunnel data. In addition the possibility to use the batch version of MegaCads for the optimization of multi-element airfoils is described.

Introduction

One key aspect of numerical aerodynamics in aircraft design is the accurate computation of pressure distributions and the lift and drag coefficients. These important values have to be predicted with an error of less than 2% for the complete flight envelope, compared to windtunnel measurements. This accuracy can be reached using structured grids of high quality and two-equation turbulence models.

Difficulties arise from the fact that turbulence modeling may not be sufficient for many industrial applications and from a lack of robustness for complex configurations. Furthermore, the generation of multiblock structured grids for high Reynolds-number flows and configurations with high-lift devices require a lot of expertise to ensure that discretisation errors can be kept at a minimum. For industrial applications it is also of great importance to validate the CFD software for different configurations and flow conditions so that results for new designs can be trusted.

In 1996 the DLR, Daimler-Benz Aerospace Airbus and the universities of Braunschweig, Berlin, München and Darmstadt started the project MEGAFLOW [1]. It has the objective to develop a reliable, efficient software system with high quality standards for the aerodynamic simulation of aircrafts in cruise and take-off / landing configuration. The project is supported by the German Ministry for Education, Science, Research and Technology (BMBF). The software system consists of the flow solver FLOWer and the grid generation system MegaCads [2,3]. For the validation of the MEGAFLOW system detailed results from windtunnel experiments are available for transport aircraft configurations in cruise and high-lift conditions.

Development of MegaCads

MegaCads is an interactive as well as an non-interactive system for the parametric generation of structured multiblock grids. Compared to the version which was presented in 1996, significant enhancements have been introduced. New functions for the creation of grids, an extension of the data structure and a modified graphical user interface are the key advantages of the current release.

An elliptic smoothing algorithm with an advanced control of the source terms [4] and a biharmonic technique with multigrid acceleration [5] have been integrated to offer a simplified control of the grid quality. Using so called „offset functions" grid blocks close to the contour of the geometry can be defined to fill boundary layers. Other functions, e.g. surface/surface intersection, have been improved concerning their robustness and accuracy. Another important development is the implementation of a simplified „Grid Topology Model" [6,7] which enables the user to make use of the growing block topology while generating the grid. Point distributions on edges can be propagated with respect to the arclength of corresponding edges. Faces, consisting of four edges, and blocks, consisting of six faces, can be detected and generated automatically for non degenerated cases.

In agreement with different users the graphical user interface has been completely revised. Graphic functionality is extended and supports OpenGL. The number of dialog boxes has been reduced significantly to use the drawing area more efficiently. Figure 1 gives an impression of the current design.

Another important feature that is finished is the non-interactive mode of MegaCads. This mode can be used on Unix workstations as well as on high performance computers like CRAY/SGI and NEC-SX. The user can easily select between the two modes during the configuration of the software. MegaCads can now be used in a close coupling with flow solvers to perform automatic grid generation in an iteration process for design and optimization purposes.

Aerodynamic Applications

In the following all grids presented have been generated with MegaCads. The flow solutions have been obtained using the FLOWer software. This code solves the compressible, three-dimensional Reynolds-averaged Navier-Stokes equations with a cell vertex finite volume approach. It is derived from the DLR CEVCATS code [8] and includes algebraic and low-Reynoldsnumber two-equation turbulence models.

DLR-F6 Aircraft Configuration

The DLR-F6 windtunnel model is a generic twin engine aircraft configuration for transonic flight conditions. It has CFM56 like through-flow nacelles and is used for the study of engine-airframe interference phenomena for different types of engines and pylons. The design Mach number is $M_\infty = 0.785$ at $C_L = 0.5$. The position of the nacelles is defined so that strong interference effects can be recognized. Since 1992 experimental and numerical investigations concerning different engine sizes and positions have been performed together with ONERA [9,10]. In the past, calculations were restricted to Euler and Navier-Stokes calculations resolving the wing boundary layer only because of the limitations of our „batch" grid generation system [11].

Grid Generation

The resolution of the boundary layers can be done adequately using C-topologies in streamwise and O-topologies in spanwise direction. The fuselage, the wing and the pylon are included in these C-O blocks. For the through flow nacelle a circumferential C-grid is used. The inner part of the nacelle is filled with an H-type grid. The wakes of all components are extended to the end of the far field outflow boundary. All C-O blocks are included in a H-O topology which defines the far field boundary. Figure 2 gives an overview of the coarse surface grid and selected planes. A difficult part of the block edge definition process occurs at the intersections of the C-O blocks at the nacelle-pylon-wing connection. Figure 3 shows surface grids and planes of the C-grids for the nacelle, the pylon and the wing. The user has to ensure that all blocks fit well together and that cell spacings and angles do not vary significantly at the block interfaces. Otherwise discretization errors increase. Another challenging detail can be found on the bottom of the pylon. Due to the fact that the pylon extends behind the nacelle, a block fillet has been defined with a shape like a wedge with a rhombus at one side. Figure 4 shows this block.

Usually a Navier-Stokes grid is generated in two steps. First all blocks are defined with a point distribution suitable for Euler calculations. Next the distributions are adapted by the user to fill the boundary layers. For small grids this can be done quickly by changing the variables for the grid density in the MegaCads script file and performing a restart of the grid generation process. A point insertion technique is also available to transform Euler into Navier-Stokes grids semi-automatically. For this grid both methods have been used. The current grid consists of 55 blocks with $3.2 \cdot 10^6$ cells. The wing surface is discretized by 68 cells in streamwise and 92 in spanwise direction. All spacings are a percentage of the chord length of the wing root section. The spacing at the leading and trailing edges are 0.5%. The first spacing normal to the wing is approximately $1 \cdot 10^{-5}$ so that a y^+ between 1.5 and 4 can be computed.

Computational and Experimental Results

Computational and experimental data will be compared for $M_\infty = 0.75$, $\alpha = 1.0^0$ and $Re = 3 \cdot 10^6$ because quite strong interference effects occur on the wing lower side close to the pylon. All computations have been performed using the Baldwin-Lomax turbulence model. Figure 5 shows the pressure distribution at a wing section inboard of the pylon and the polar. The difference in drag for this grid density is approximately 5% [12].

DLR-ALVAST High-Lift Configuration

The DLR-ALVAST fuselage and wing design is very similar to DLR-F6. The wing has an additional slat along the full wing span and a Fowler flap. The flap is separated into an inboard and outboard part. Due to the step-by-step approach in the

MEGAFLOW project the validation of the high-lift configuration is started without an engine and pylon. Figure 6 gives an impression of the geometry.

Grid Generation

The application of C-grids for slat, wing and flaps is more difficult than for a configuration in cruise flight condition, when all wakes were extended to the outflow farfield boundary. Another point to mention is, that the large amount of wake cells is not necessary. Therefore a topology in streamwise direction was chosen, where all elements are supplied with small wake blocks, since the configuration has blunt trailing edges. The flap together with its wake block is then surrounded with an O-grid. In case of the other elements slat and wing, the wake-blocks are bent and end either on the contour or they are extended to the lower farfield boundary. Figures 8 and 9 present parts of the grid for the slat and flap region.

To keep the amount of grid generation processes small, the MegaCads script-subroutine technique has been used to generate local grids at different spanwise stations around flap, wing and slat sections. Sections without a flap were filled with volume grids having a virtual flap surface geometry. Therefore nearly the same subroutine was applied to all spanwise flap sections. After the generation of local 2d grids around the complex parts of the geometry by using subroutines, 3d volume grids are generated using transfinite interpolation which enclose all parts of the wing. Beside the wing tip volume grids are generated by a simple extrusion technique. At that state of work the further amount of grid generation reduces to the problem of filling the volume around a wing-body configuration with an infinite wing. Here a standard C-O topology can be applied. The current grid for Euler computations consists of 53 blocks and $3.3 \cdot 10^6$ grid cells. The grid for Navier-Stokes computations has $7.6 \cdot 10^6$ grid cells and was generated by simply changing script variables which control the cell spacing and number of cells normal to the surface of each component.

Computational and Experimental Results

Figure 9 shows the comparison of the pressure distribution for the take-off flow conditions $M_\infty = 0.22$, $\alpha = 12.03^0$ at $\eta = 0.265$. The main features of the flow can be computed using the Euler mode of FLOWer. Due to the fact that viscous effects are not considered, the differences are clearly visible.

Optimization of 2D High-Lift Devices

In addition to the aerodynamic analysis of existing configurations it is of increasing importance to design and optimize new geometries. Therefore a geometry modeler, a grid generator, a flow solver and an optimizer have to be connected closely to form an optimization system. All components of the system have to work in an efficient and robust way to ensure that optimization cycles run automatically. Because the complete grid generation process interactively defined with MegaCads is stored in a script file, it is possible to use it for an automatic grid generation if the geometry changes. A first application of the system is the minimization of the drag coefficient at constant lift for a 3-element high-lift airfoil (NHLP L1T2).

The topology is designed to handle flaps and slats. Although the grid generation process offers high flexibility, some constraints are necessary for the deflections because block connections and point mappings cannot be changed during the optimization process. The reason is, that the topology information of the flow solver is fixed during the solution process. To retain grid quality, the partial extended positions of the moving elements are limited by the increasing spacing on the downstream element. The full extended positions are constrained by large cell aspect ratios in the element coves. Previous investigations have shown that these limits are not reached during the optimization process. Figure 10 shows the grid and the possible range of the flap movement.

Although the grid is smoothed with an elliptic technique at each optimization cycle, the time needed for grid generation is less than 0.2% of the overall solution time.

Conclusions

The current status of development of the grid generation system MegaCads and the flow solver FLOWer allows to compute the aerodynamic of aircraft configurations in cruise and high-lift flight conditions. The accuracy for the prediction of drag still has to be improved. Suitable structured grids can be generated. The objective to repeat the grid generation process automatically for design and optimization purposes has been demonstrated for a multi-element airfoil.

It is still necessary to speed up the time-consuming process to generate high quality Navier-Stokes grids for new configurations. In addition, future developments will also focus on the enhancement of the robustness of MegaCads for three-dimensional optimization. The validation of the MEGAFLOW system will be continued with a strong focus on the application of a k-ω turbulence model.

Acknowledgment

The authors highly appreciate the work of R. Rudnik and E. Monsen who are responsible for the numerical results.

References

[1] N. Kroll, W. Bartelheimer: *Projektbeschreibung MEGAFLOW.* DLR-IB 129-96/8, http://www.bs.dlr.de/sm/ea/Proj_MEGAFLOW/proj_megaflow_e.html, DLR, Institute of Design Aerodynamics, Braunschweig, 1996

[2] Installation and User Handbook for the Project FLOWer. Doc.-Nr. QS-FLOWer-3008, DLR, Institute of Design Aerodynamics, Braunschweig, 1996

[3] O. Brodersen, M. Hepperle, A. Ronzheimer, C.-C. Rossow, B. Schöning: *The Parametric Grid Generation System MegaCads.* Proc. of 5th Intern. Conf. on Numerical Grid Generation in Comp. Field Sim. Ed. B.K. Soni, J.F. Thompson, J. Häuser, P. Eiseman, NSF Eng. Center, Mississippi, 1996, pp.353-362

[4] P. Niederdrenk: *On the Control of Elliptic Grid Generation.* To be published at the 6th Intern. Conf. on Numerical Grid Generation in Comp. Field Sim., 1998

[5] J. Yan, L. Xue, F. Thiele: *Biharmonic Grid Generation Using a Multigrid Method.* Hermann-Föttinger-Institute, Technical University of Berlin, 1997

[6] K.J. Weiler: *Topological Structures for Geometric Modeling.* Ph.d. Dissertation, Rensselear Polytechnic Institute, Troy, New York, 1986

[7] A. Gaither: *An Efficient Block Detection Algorithm for Structured Grid Generation.* Proc. of 5th Intern. Conf. on Numerical Grid Generation in Computational Field Simulation. Ed. B.K. Soni, J.F. Thompson, J. Häuser, P. Eiseman, NSF Eng. Center, Mississippi, 1996, pp. 443-450

[8] R. Radespiel, C.-C. Rossow, R.C. Swanson: *An Efficient Cell-Vertex Multigrid Scheme for Three-Dimensional Navier-Stokes Equations.* AIAA Journal, Vol. 28, No. 8, 1990, pp. 1464-1472

[9] C.-C. Rossow, J.L. Godard, H. Hoheisel, V. Schmitt: *Investigations of Propulsion Integration Effects an a Transport Aircraft Configuration.* AIAA Paper 92-3097, 1992

[10] O. Brodersen, C.-C. Rossow: *Calculation of Interference Phenomena for a Transport Aircraft Configuration Considering Viscous Effects.* Proc. of Conf. Recent Developm. and Appl. in Aeron. CFD, Bristol, UK, 1993, pp. 6.1-6.13

[11] C.-C. Rossow, A. Ronzheimer: *Multiblock Grid Generation Around Wing-Body-Engine-Pylon Configurations.* Proc. of Conf. on Numerical Grid Generation in Computational Fluid Dynamics and Related Fields. Ed. A.S. Archilla, J. Häuser, P.R. Eiseman, J.F. Thompson, North-Holland, 1991, pp. 357-368

[12] E. Monsen: *Validation of FLOWer.* Presentation at the MEGAFLOW workshop, DLR-Göttingen, November 1997

Figures

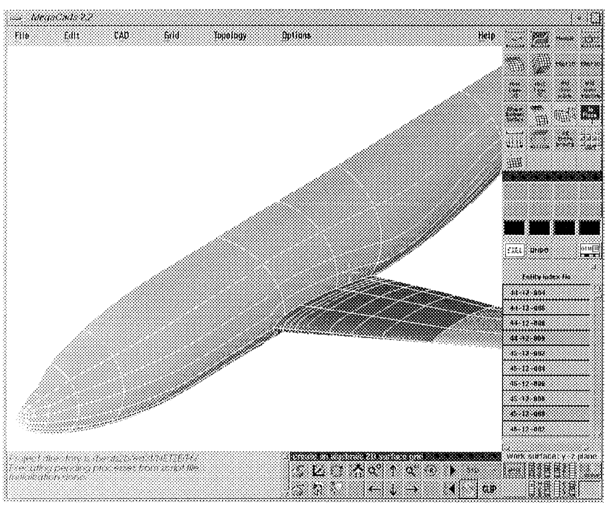

Figure 1: Redesigned MegaCads graphical user interface

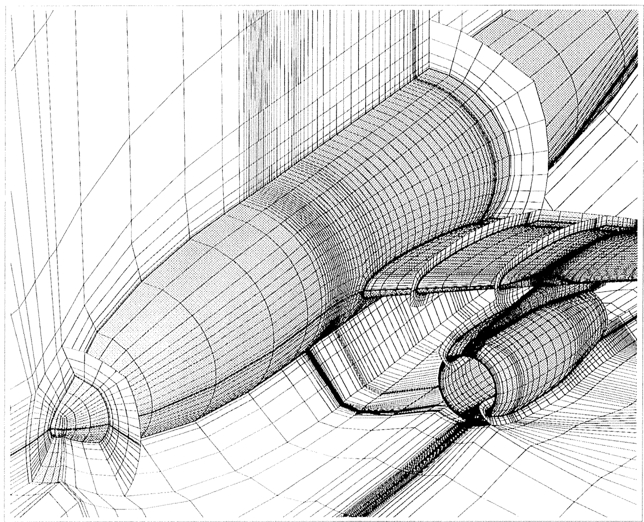

Figure 2: DLR-F6 Navier-Stokes grid (coarse), C-O grids, embedded in H-O grids

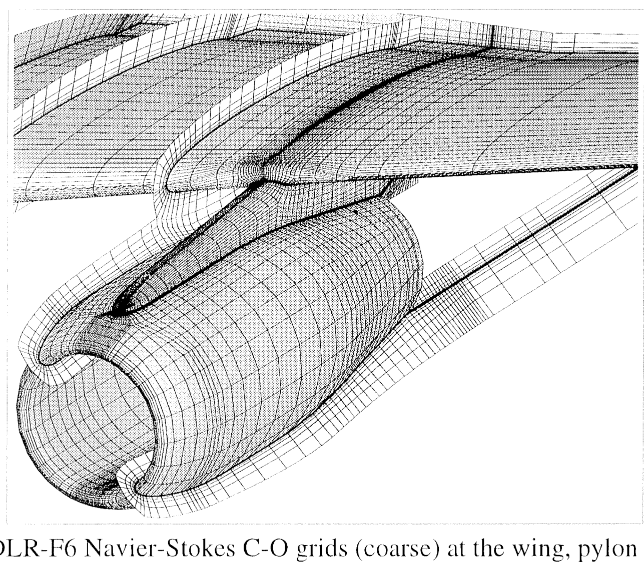

Figure 3: DLR-F6 Navier-Stokes C-O grids (coarse) at the wing, pylon and nacelle

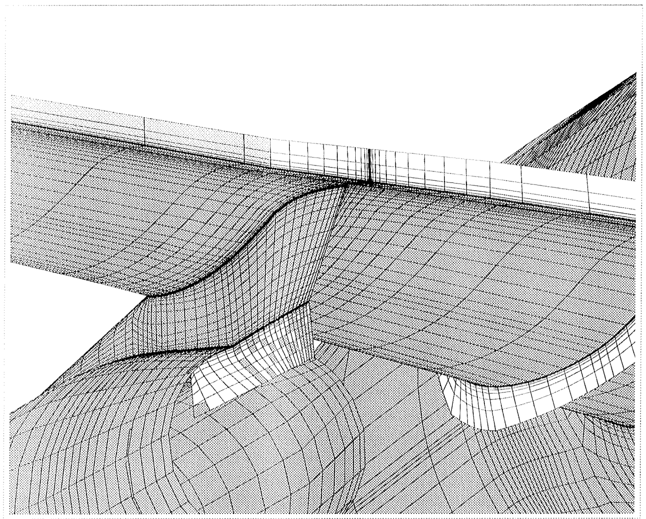

Figure 4: DLR-F6 Navier-Stokes grid (coarse), special block below the pylon

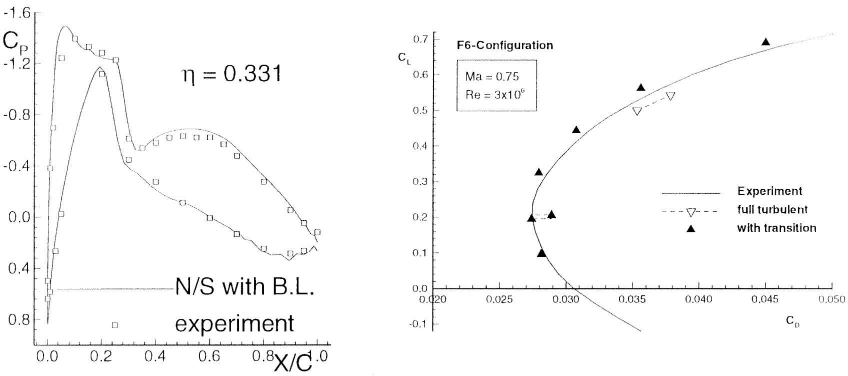

Figure 5: DLR-F6, comparison of numerical and experimental results

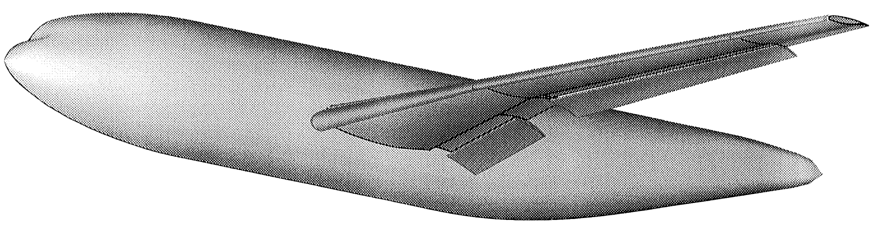

Figure 6: DLR-ALVAST high-lift configuration

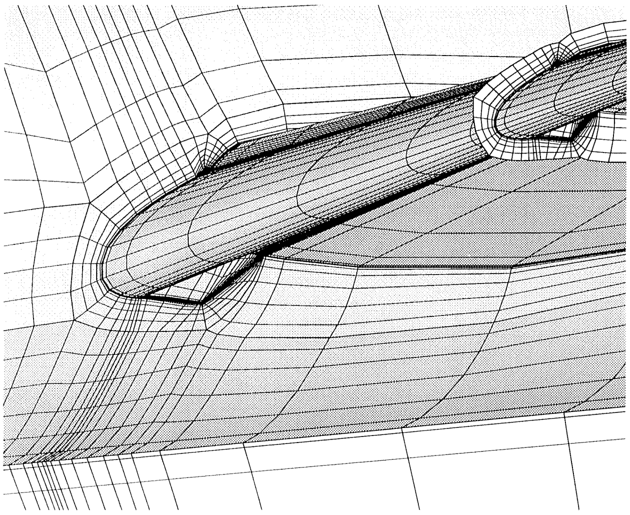

Figure 7: DLR-ALVAST Navier-Stokes grid (coarse), slat region

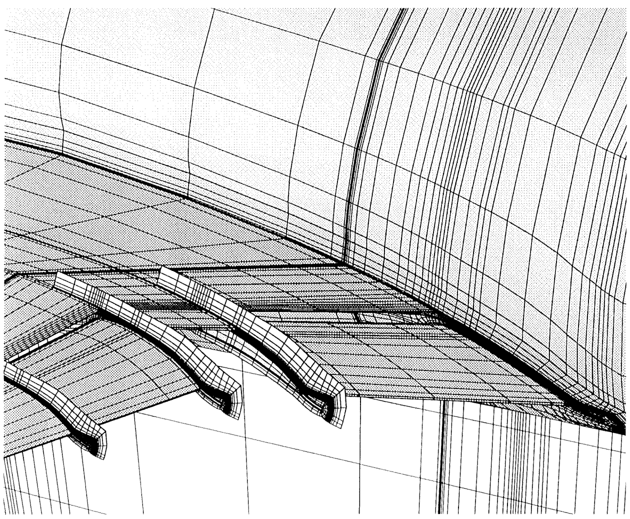

Figure 8: DLR-ALVAST Navier-Stokes grid (coarse), flap region

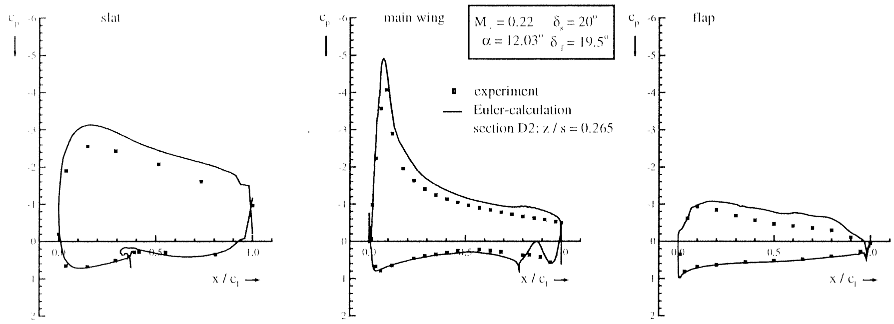

Figure 9: DLR-ALVAST, comparison of numerical (Euler) and exp. results

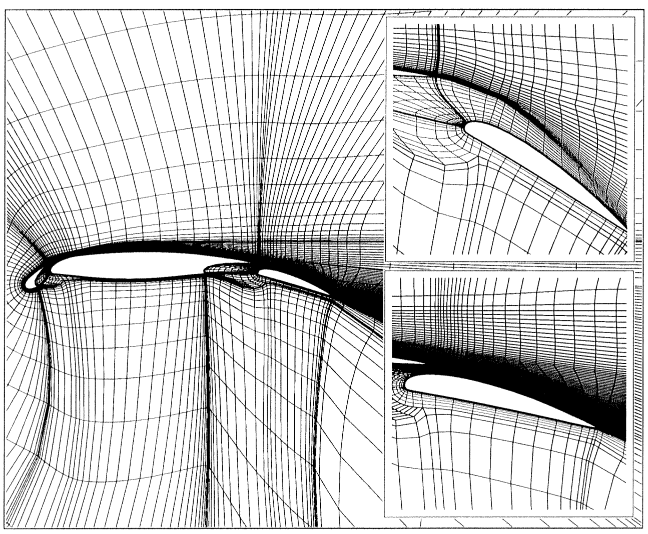

Figure 10: Multi-element airfoil grid (coarse),
Variation of the grid during the optimization process