TRANSONIC AIRFOIL DESIGN WITH EXPERT SYSTEMS

Robert E. Zores
DLR German Aerospace Research Establishment
Institute for Fluid Mechanics
Dep.: High Speed Aerodynamics
37073 Göttingen, Bunsenstr. 10
Germany

Abstract

The combination of new technologies with classical methods for designing transonic airfoils is described in this paper. While designing supercritical airfoils the skilled aerodynamicist is usually completely aware of the laws and criteria that guide the designing process. An expert system can help inexperienced engineers reduce the time of learning and exploring new generated airfoils. It speeds up the process of generating, calculating and analyzing the airfoil. Proven programs, such as inverse CFD solvers, beside new technologies from computer science, can be used to build new tools for aerodynamic design on workstations. A selection of improved workstation tools to design transonic airfoils is described.

The capability of the expert system is shown by two examples. The first example raises a more academic question on airfoils which can be concave on their upper side. The second example shows how an expert system can be used to support wind tunnel tests.

1 Introduction

Beside traditional technologies for designing new aircraft configurations one has to look at other fields of research.

CFD research in the past few years was heavily based on supercomputers. But with a new generation of programs it is possible to use modern workstations for solving the equations. At this time we are still limited to solving the steady Euler or potential equations, but in the future it will surely be possible to compute the Navier Stokes equations with workstations. This progress will have a great impact on the airfoil designing process. Also, the limitation to 2D methods will disappear in the future. 3D design will become a common method that needs powerful tools for parametric generated surfaces or computational grids.

In the past, there were many attempts to speed up the design process. Also, there was success in rewriting new programs, such as expert systems which were heavily based on artificial intelligence systems. The general idea behind expert systems is to keep and store proofed or heuristic rules of doing something. Keeping and storing data (or knowledge) is usually done with conventional databases. In the case of analyzing airfoils there are a lot of rules to learn. But not only rules have to be stored; also geometry data as well as physical boundary conditions such as Mach number, angle of attack, etc. must be stored. Some systems like `Enginous` [7] [11] are based on large commercial expert shells (KEE™). These shells are based on 4th generation programming languages like `LISP` or `SCHEME.` Expert shells need to have links or interfaces to pre- and postprocessing programs.

Rule based systems like `IDEA` [3] focus on formulating and applying rules to the expert system.

Our goal is to achieve this speed up in the design process based on the idea of saving and using validated CFD codes with modern techniques of computer science. In this specific area the airfoil design problem can be faced using two different approaches:

- Direct Design, using geometry tools and only analysis code.
- Optimization, using inverse methods coupled with analysis code.

Combining these approaches with other pre- and postprocessing tools leads to a new expert system for airfoil design.

2 Aim

In this paper a definition of a new expert system which combines traditional CFD work and new methods of computer science is described. As an example of how it can be used, a redesign of a transonic airfoil will be presented.

Supercritical wing technology is expected to have a significant influence on the next generation of commercial aircraft. The use of supercritical wings could economize fuel consumption by reducing drag. An effective approach to the design of supercritical wings is through the development of shockfree airfoils.

The design of shockfree airfoils (or at least tolerable weak recompression shocks) was based on different methods which required a large amount of trial and error work. Some computer programs represent a more advanced approach, which makes it possible to assign the pressure as a function of arc length and to obtain a shockfree airfoil that nearly achieves the desired pressure distribution[10]. This inverse method reduces the amount of time for the design process enormously.

An expert system should support CFD (inverse)-solvers with interactive geometry and curve manipulating programs, which were often in the past very basic. Also, access to previous computed or measured data of airfoils is needed. Consequently a mechanism for importing data to the expert system is necessary.

Another aspect of using expert systems is the fact of storing data or knowledge. Even during code development for the CFD-solver it can be useful to feed experience to the database. Usually this knowledge is put directly into the solver, but it might be necessary to have access to it.

Summarizing all this information, one big problem could also be solved for the future: namely, not losing knowledge while losing experts.

As shown, an expert system does not consist of only one program. It is a whole toolset which must be expandable in the future. With these tools an engineer should be able to design families of transonic airfoils for use as aircraft wings.

This expert system is used at DLR Göttingen for case studies for new configurations and as a research development system for 2D airfoil and 3D wing design.

Other specific expert systems for various other problems related to CFD work (e.g. cascade aerodynamics, complex configuration design) may follow.

3 Toolset

To make a system usable for engineers with different tasks of aerodynamic work there must be a toolset that we call an “Aerodynamic Workbench” or expert system. This toolset consists of a large number of tools to manipulate geometries such as airfoils or complex configurations in 2D and 3D. Programs for editing or generating geometries are called preprocessing tools. This can include conventional CAD programs as well as geometry generators based on specific functions.

There should also exist various CFD solvers for different tasks and with different performances. To use these solvers on workstations they have to be fast and easily satisfied with a small amount of main memory. Therefore, this part is called “fast-processing”. Using this part means using CPU power of the workstation. Because a lot of time is consumed for calculating polars (Lift / Drag) or other parameter variations there are various approaches to get the result fast.

Finally, there must be a system to explore and visualize the CFD data. This post-processing stage is becoming a more and more standard task for visualization programs. But there appear many highly sophisticated evaluations of CFD data, so that this will be an ongoing project of research.

One main design goal for this expert system is to keep clean interfaces between these three tasks of work. Therefore, all tools write and read to the same database. This database should be standardized and available for all computer types. Then in future, it will be easy to replace or add new components to the system. Right now, there are two solvers implemented in the system.

To keep the system modular and portable as possible was also an objective during implementation and design.

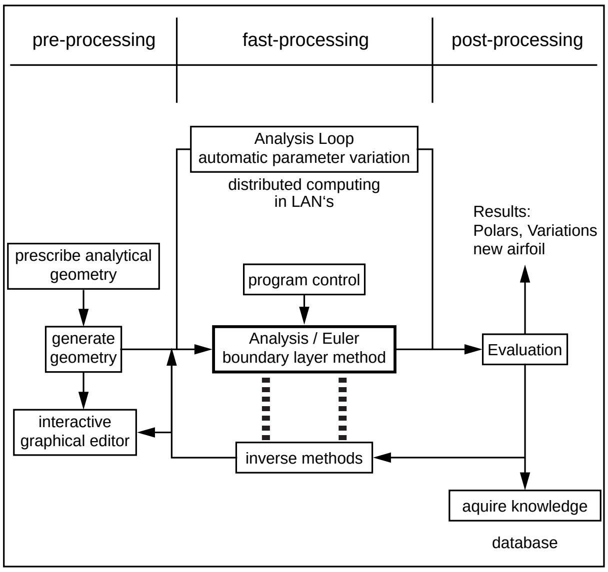


Figure 1: Diagram of the dataflow in the expert system

The above shown diagram shows the conventional way of designing and analyzing airfoils. To represent this way of working the user interface is designed to follow the dataflow.

4 Data Format

One of the problems, which occurred while using these codes, is the use of complex input and output formats. There is no convention on how to use these programs. We discovered this as a main problem with code which was mainly written in `FORTRAN` and `C`. If the developer of a specific code cannot be asked what to do with special parameters, the program is useless. With preserving the knowledge of the developer in databases, this problem can be solved. The question now is how to save and formalize this knowledge. First of all one has to find a database system which deals with different computer systems and different programming languages. While in the CFD world mostly `FORTRAN` and `C` code is used, we decided to implement all I/O data with the `netCDF` [13] database format. Currently, it seems to be the easiest way of storing numerical data beside object oriented information, such as valid ranges or further items. `netCDF` (Network Common Data Format) is based on `XDR` (external data representation) which is the most widely used binary data representation. (e.g.: This is also used with `NFS`, which is available for all workstations). Also, almost any supercomputer can read and write this binary format. It has become the European standard exchange format [5] for scientific data. `netCDF` can also be used to generate `FORTRAN-` or `C-` code from a given dataformat. Once the input or output format is described in a specific file, either the database or the I/O routines can be generated. This automatic generated code can now easily be added to given CFD codes. This makes it comfortable to adapt given data formats to `netCDF`.

Beside storing numerical input or output data this format also allows us to save text to specific variables. This can be text in any form and length. This leads to the possibility of formulating rules or writing information to the database. Also valid ranges for variables can be prescribed and be checked before running a CFD program. These features have made this format a widely used standard in many visualization systems.

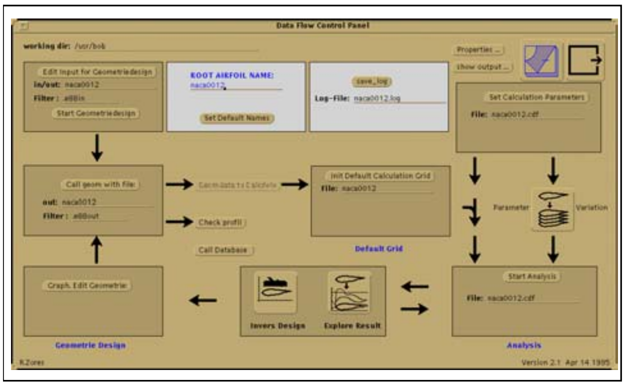
4.1 Implementation

To support such an expert system it has to run on a modern workstation, with an interactive and powerful window system. To use it as an interactive tool the computing time for flow solvers should be less then 15 minutes.

Because all codes are based on some kind of input data, it is important to use a simple and easy to understand user interface.

We choose a `UNIX` workstation running the X-Windows system. The supported window managers are currently `XView` and `Motif`.

Since the expert system needs these graphical user interfaces (GUIs) it would take a great amount of time to develop these interfaces from scratch. CASE-Tools (Computer Aided Software Engineering) are used for rapid prototyping and testing. Even for porting the GUI-code to different platforms, they are very useful. With the new generation of these CASE-Tools it is even possible to build the final graphical user interface. The typical dataflow in this expert system is also represented in the main control program. (Figure 2)

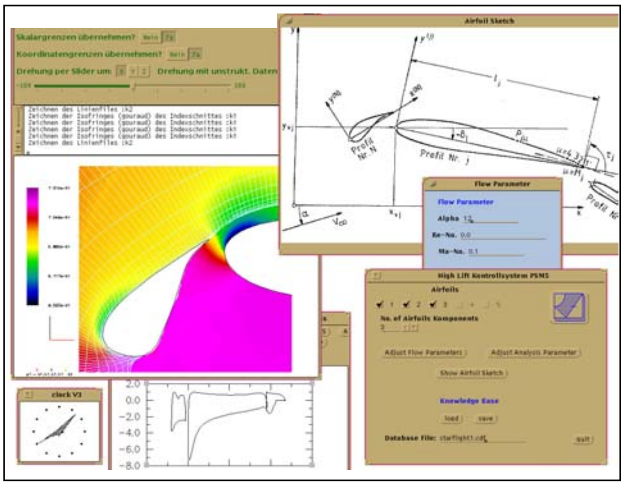


working dir: /usr/bob
in/out: naca0012
Filter: a88in
ROOT AIRFOIL NAME: naca0012
Log-File: naca0012.log
File: naca0012.cdf
out: naca0012
Filter: a88out
File: naca0012
File: naca0012.cdf

Figure 2: Dataflow Control Panel

During the first implementation a sophisticated interface (written in `LISP`) was used to map rules given by CFD codes or geometry tools. But it shows that in using this method portability to other systems suffers. Thus, all rule checking and evaluation of meaningful input is done in the user interface. This keeps the GUI portable, but needs a close relation to the GUI builder. All information about the look and feel and connectivity rules are stored in so-called meta-files. This guaranties portability and continued existence.

Summarizing the implementation, one most important feature is to keep the graphical user interface as portable as possible. There is no real standard yet to use as a reasonable base. It is very hard to predict which GUI will become standard in the future.



```
Zeichnen des Linienfiles :k2
Zeichnen der Isofringes (gouraud) des Indexschnittes :k1
Zeichnen der Isofringes (gouraud) des Indexschnittes :k1
Zeichnen der Isofringes (gouraud) des Indexschnittes :k1
Zeichnen der Isofringes (gouraud) des Indexschnittes :k1
Zeichnen des Linienfiles :k2
```

Alpha 12
Re-No. 0.0
Ma-No. 0.1
No. of Airfoils Komponents 3
Database File: starflight1.cdf

Figure 3: Screenshot while working with the visualization part of the expert system

To take advantage of new networking capabilities the system can distribute numerical computation tasks across a local area network (LAN). This is done using methods like `RPC` (remote procedure calls) or via the `Network Queuing System (NQS)`.

With this method the computing time of the local workstation can be reduced. This feature is often used with calculating exact polars or automated multiple parameter studies of an airfoil.

4.2 Geometry

On top of the previous mentioned platform we implemented geometry manipulation codes by Sobieczky [9] with new interactive tools. They allow us to describe geometry using parameterized functions. Thus, it is possible to generate as many (or as few) points on a curve as needed. Also, it is possible to modify the airfoil automatically during the design process with noninteractive tools. (e.g. redistribute points, flatten, blow up, etc.).

For the interactive modification of geometries a special object oriented tool was developed. This can also be used to define pressure distribution functions along airfoils.

All 2D interactive editing known from CAD systems can be done within this program. A special feature is the ability to add new programs (filters) as input or output to this tool. Therefore, almost any geometry manipulation can be done while using a unique graphical user interface. Since the development of these “Computer Aided Geometric Design” (CAGD) tools in `C` or `C++` which are based on methods by Farin [2], they can easily be ported to other systems. Even converting geometry data to very specific CAD data formats can be done with these tools.

Beside these programs, a simple airfoil builder is available. NACA 4 digit and other common airfoils can be generated to help starting to work with the expert system.

It is important to mention that all geometry tools can also be used in a noninteractive environment. With this feature they can also be implemented in other program environments such as grid generators or CAD programs.

4.3 Flow Solver

In addition to powerful geometry tools another key application in this expert system is a flow solver. Most of the information and results found while analyzing airfoils are based on the flow solver.

In the described expert system the transonic airfoil design analysis is currently based on a 2D-flow solver. Here, Drela´s [1] `mses`-code is used, which contains a 2D Euler method with a boundary layer extension. It features also the previous mentioned inverse method. Drela’s code is a well known 2D solver which is a fast and powerful tool for analysis and inverse design. With this solver it is possible to analyze airfoils with up to 4 components.

Some modifications and enhancement had to be done to use `mses` in this expert system. First, all I/O functionality has to be changed to `netCDF`. Since Drela‘s solver actually consists of a streamline grid generation tool, this was also added to the expert system. To make some extended evaluations of the flow data such as hodograph mappings [10], new routines had to be added.

Another flow solver from Jacob (`psm5`) [4] was added, which is used in addition to `mses`. This solver investigates flow around airfoils with flaps and slats. It is based on a potential flow method (also coupled with boundary layers) which is a very fast scheme to get the coefficient for lift, drag and momentum. It is useful for fast 2D and 3D high-lift prediction. It also contains methods for considering ground effects.

The use of various codes in an expert system gives the engineer the possibility of choosing which method is appropriate for the problem. With ongoing research there will be more flow solvers added, depending on their usability for an interactive expert system.

4.4 Data Reduction

There are actually two databases used with the expert system. One is used to work with one family of airfoils (temporary database), the other to store the final data or knowledge. During a design process for one airfoil all data are kept in the temporary database file.

Using this method keeps the final database small. Unnecessary or redundant information which is sometimes needed is avoided in the final database.

During the design process the database consists of three parts:

- program control part: information e.g. for solvers, geometry-tools or visualization programs.
- flow values: e.g. Mach number, angle of attack, etc.
- numerical information: e.g. airfoil data, computational grids, internal variables, etc.

For each item in this database additional information can be available in the form of constraints or just text.

To explore the large amount of numerical data produced by flow solvers we treat these data with special tools to use them in various graphics system. This tool can handle different databases simultaneously, so different cases can be compared very easily.

To separate numerical, physical or program control data a special program is needed. It is used to process data to be stored in a database as well as data to be visualized.

4.5 Visualization

Visualization of flow data has become an important feature while using CFD codes. Many flow phenomena can only be shown with qualified visualization tools. In addition to standard flowfield visualization pictures, like pressure or entropy distribution, often unsteady solutions have to be visualized. At this time there is no solver available for unsteady flow for this expert system, but showing the quasi unsteady flow solution given by parameter or airfoil variations, helps to investigate flow phenomena.

Animation techniques can be used to visualize multiple parameter studies.

Enhanced features, which are only needed for specific airfoil design tasks must be added to the visualization system. Often used examinations such as curvature plots or more sophisticated investigations of the transonic flow phenomena with classical hodograph mappings are included to the expert system.

To visualize CFD field data (2D, 3D), which were extracted from the expert system database, we use the `HighEnd` [8] system which was developed at DLR Göttingen.

For typical 2D plots there are many public domain programs or commercial programs available. The program ACE/gr [12] (`xvgr`, `xmgr`) satisfies the need for reading *x-y* graphs as well as reading the `netCDF` format. It can be used with templates to prescribe often used plots.

To keep the visualization part as modular as possible many different file formats can be written. Changing to other scientific visualization formats or systems could be necessary.

4.6 Knowledge Databases

Keeping and storing the results of the design process is also an aim of the expert system. For example, important coefficients, airfoils or c_p-distributions are saved as well as knowledge which is provided by the engineers or experts.

While using the system, this database grows with more and more data, which can be easily extracted with user friendly graphical interfaces.

Before starting a new case study the database can be searched to retrieve information about similar airfoils or other flow conditions.

The database technology is based on `netCDF`. Once information is put into the database, airfoil information can be provided to other programs (e.g. visualization tools). The knowledge base is not limited to specific expert systems. Together with other specific expert systems this may become a new way to keep knowledge available. Converting data to other common database formats may be necessary in the future. So probably this database can be used as a server for e.g. airfoils, specific pressure distributions or airfoils descriptions.

5 Investigating new airfoils

In this section examples of using the expert system at DLR Göttingen are shown. In addition to these examples the system is used to investigate the shock-boundary layer interaction on turbulent airfoils. Within this work local contour modifications with “bumps” are made.

The first example shows what can be learned while redesigning an airfoil with the inverse method.

Wind tunnel support is another aim of the expert system. The second example indicates that an expert system can serve as a control mechanism for wind tunnel tests. Furthermore, predesign stages for planned wind tunnel experiments can be tested.

5.1 Airfoil redesign

Efficient supercritical airfoils typically have a flattened upper surface.

Designing a shockfree airfoil is typically the main goal. To achieve this goal it might be necessary to adapt supercritical airfoils on the upper side. There are many problems to be solved with adapting airfoils. It seems very difficult to build airfoils for wind tunnel tests. Therefore, it is less expensive to do investigations with “workstation aerodynamics”. In our case the expert system can help to expand the knowledge of such airfoils.

In a first test case we started with a given airfoil (CAST 7). As shown in Figure 4 in comparison with wind tunnel experiments, this airfoil is not shockfree at these flow conditions.

What happens to the airfoil while prescribing a shockfree c_p-distribution with the same lift c_l?

A segment which is affected by the supersonic region from $x/c = 0.1$ to $x/c = 0.75$ on the upper side of the airfoil is chosen to be modified. The remaining airfoil coordinates are fixed. Starting from the initial c_p-distribution only the specified segment is modified. To avoid curvature problems at the segment edges one has to be very careful while prescribing the new c_p-distribution. Some improvements to the original inverse method have been implemented.

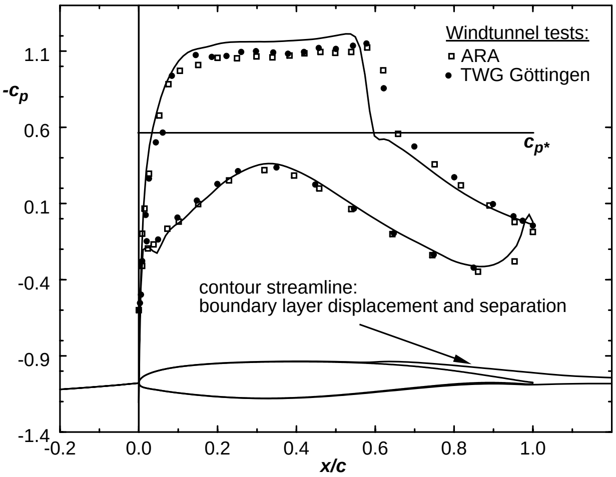


Figure 4: CAST 7 original design case with wind tunnel results

Redesigning the airfoil at $M_\infty = 0.775$ and $c_l = 0.63$ with the inverse method leads to a modified upper side which is shown in Figure 5. The result is a flattened airfoil shape as expected. With this 12% thick airfoil, a reduction of ~0.6% thickness is reached.

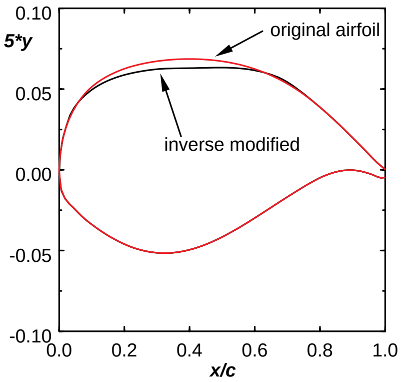


Figure 5: Original CAST 7 and modified airfoil

After recalculating with the modified airfoil the c_l / c_d - polar indicates a better performance within the design area $c_l = 0.6$. A typical point optimized airfoil is created.

For designing new wings many airfoil investigations have to be done. This includes evaluating polars at different Mach numbers or angles of attack. Redesign or modifications like the above shown example can be done within minutes on modern workstations.

New design case studies are verified by comparing airfoil polars, drag rise boundaries and *L/D* (M, c_l) distributions. Furthermore, new design concepts like [6] will relatively easily be implemented in 2D and future 3D versions of this system, since it is the goal of an expert system to convert experts‘ applied mechanics knowledge to “black box” type computer codes.

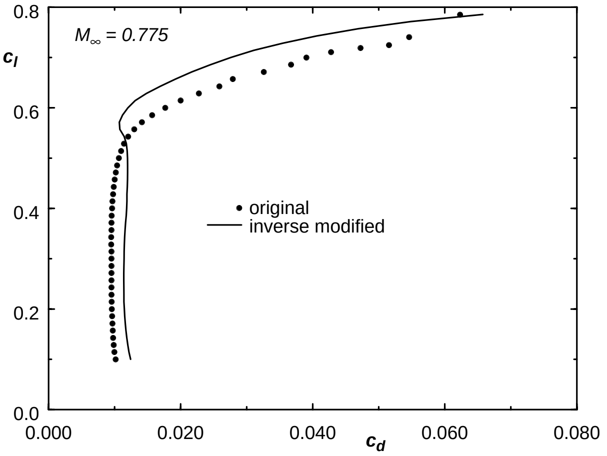


Figure 6: Lift / Drag comparison

Further, during investigating the CAST 7 airfoil at higher Mach numbers we found an interesting phenomena.

While increasing M_∞ and fixing c_l the upper airfoil side became more and more flat (flattened by ~1% of chord). Such “Hanging Shock Airfoils” have a slightly concave portion of the upper surface.

Investigating the flowfield with visualization tools shows a shock appearing in the flow above the airfoil. Figure 7 shows the entropy distribution in the flow field which is produced by the shock.

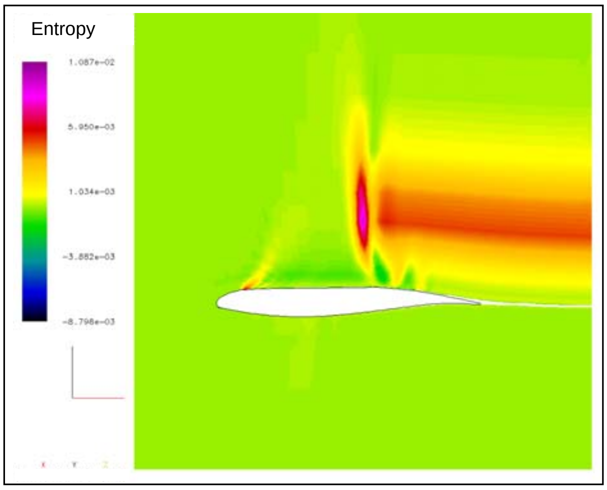


Figure 7: Entropy distribution on hanging shock airfoil

Comparing the overall drag coefficients c_d to the original airfoil shows nearly the same values. A difference can only be found in the drag components (wave-drag, viscous-drag). We found an increased wave drag, and due to less boundary layer interference with the shock a decreased viscous drag. Due to the fact that strong separation has been avoided this phenomenon could lead to more investigations in this direction. Comparing the CFD-results with wind tunnel tests of this very special family of airfoils may or may not confirm this phenomenon. In any way, results like this are worth considering to be stored in the knowledge base of an expert system.

5.2 Wind tunnel Experiment Support

The traditional way of investigating airfoils is to build wind tunnel models. In the past many attempts were made to improve wind tunnel techniques. The DLR Göttingen operates a modern cryogenic wind tunnel, where new laminar airfoils can be tested. This wind tunnel has two different test sections. One is a conventional test section with slots, the other one has adaptive walls on the upper and lower side. The expert system can help to adapt wind tunnel walls as shown in Figure 8. This is possible since the flow solver operates on streamlines. Therefore the walls could just be adjusted along such streamlines.

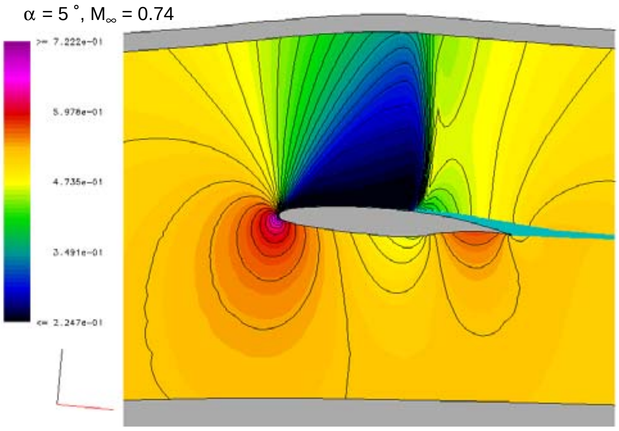


Figure 8: Simulated adaptive wind tunnel walls

Another task of the expert system can be the on-line comparison of wind tunnel tests. It can be used by the wind tunnel crew through a graphical user interface. They can put their feedback and remarks to the database, also.

Wind tunnel tests allow series of measurements, suitably depicted as “carpet diagram”. On-line computation with the system allows a comparison of such data like the lift-drag ratio. Figure 10 shows the *L/D* plot with isolines for a specific range of c_l and Mach numbers. In this case 400 adjustments were computed. The plot shows the offdesign area (at $M_\infty > 0.74$ and $c_l > 0.9$) as well as the optimal range at $c_l = 0.7$ and $M_\infty = 0.71$. This can help to plan more precise experiments by choosing the range of Mach numbers and angle of attack.

Supporting the wind tunnel staff with this information can help to detect errors during measurement and may save wind tunnel time.

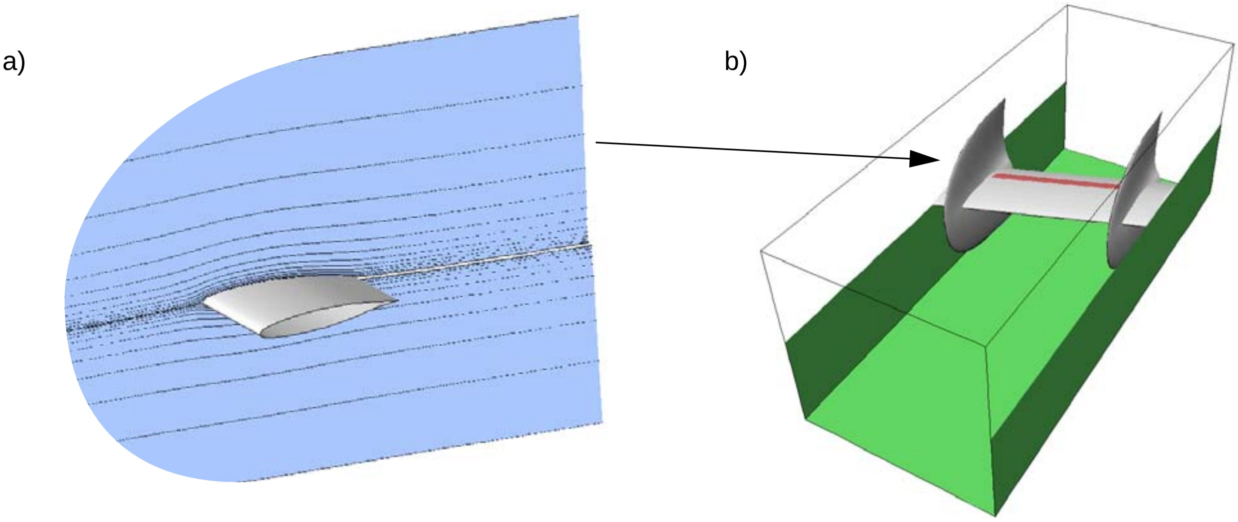


Figure 9: Stream surface used for splitter blades in wind tunnel experiments

L/D

M_∞

c_l

tions on different blades while changing flow and geometry parameters.

6 Conclusions

This paper describes a new workstation-based tool for analyzing and designing transonic airfoils.

Based on proven flow solvers and powerful geometry modification tools combined with new information technologies this tool will help aerodynamicists develop better airfoils.

This paper shows in short how to analyze the design process of transonic airfoils and which tools are needed. Combining traditional CFD methods with new computer science technology such as database systems or advanced visualization can be very successful. New airfoils can be generated, knowledge is saved in databases for future use.

So, expert systems can be installed as education or production systems with a given database.

Operating areas for the transonic expert system lie also in supporting wind tunnel experiments as well as in case studies for new configurations.

With the ongoing development of computer hardware and the availability of desktop workstations an expert system can be a platform for aerospace research.

The considered expert system is a prototype for a series of knowledge based systems which will be implemented at DLR.

7 References

[1] Drela, M., Giles M.B.
ISES: A Two-Dimensional Viscous Aerodynamic Design and Analysis Code.
AIAA-87-0424 (Jan. 1987)

[2] Farin, G.
Curves and Surfaces for Computer Aided Geometric Design: *A Practical Guide (sec. edition)*
Academic Press, Inc. ISBN 0-12-249051-7

[3] Ghielmi, Marazzi, Baron
A Tool for Automatic Design of Airfoils in Different Operating Conditions
Computational Methods for Aerodynamic Design (Invers) an Optimization, AGARD CP-463, (1990)

[4] Jacob, K., Steinbach, D.
A Method for Prediction of Lift for Multi-Element Airfoil Systems with Separation.
AGARD CP-143 (April 1974)

[5] Lang, U.
A Software Environment for Cooperative Simulation and Visualization in the Aerospace Field
High-Performance Computing and Networking (HPCN) *Int. Conference Munich, Germany, April 1994, Lecture Notes in Computer Science 797*
Proceedings, Vol.II Springer-Verlag

[6] Li, P; Sobieczky, H.; Seebass, R.
A Design Method for Supersonic Transport Wings
AIAA 95-1819

[7] Nicklaus D., Tong S., Russo C.
Engineous: A Knowledge Directed Computer Aided Design Shell
Third conference on artificial applications
IEEE (1987):

[8] Pagendarm, H.-G.
HIGHEND, A Visualization System for 3d Data with Special Support for Postprocessing of Fluid Dynamics Data
M. Grave, Y. LeLous, W.T.Hewitt (ed.) "Visualization in Scientific Computing", Springer, Heidelberg (1993)

[9] Sobieczky, H.
The Combination of a Geometry Generator with Transonic Design and Analysis
Algorithms.
AIAA 83-1862, (1983)

[10] Sobieczky, H.
Progress in Inverse Design and Optimization in Aerodynamics
Conf. on Comput. Methods for Aerodynamic Design (Inverse) and Optimization, AGARD CP 463, 1989, pp. 1.1-1.10.

[11] Tong, S.S.
Design of Aerodynamic Bodies using Artificial Intelligence/Expert System Technique
AIAA 85-0112 (1985)

[12] Turner, P.
ACE/gr Graphics for Exploratory Data Analysis.
Oregon Graduate Institute, USA (1992)

[13] Unidata Program Center
netCDF User's Guide
An Interface for Data Access.
Unidata Program Center, Boulder, CO, USA (1990)