NLR-TP-2000-506

A joint European initiative to develop hybrid grid based CFD technology for inviscid and viscous flow computations applicable to geometrically complex aircraft configurations

Publishable Synthesis Report of the FASTFLO project

T. Berglind (FFA), J.W. van der Burg (NLR), K.M.J. de Cock (NLR),

W. Fritz (EADS-M), G. Kretzschmar (IBK), N. Kroll (DLR),

D. Schwamborn (DLR), M. Sillen (SAAB)

Nationaal Lucht- en Ruimtevaartlaboratorium

National Aerospace Laboratory NLR

NLR-TP-2000-506

A joint European initiative to develop hybrid grid based CFD technology for inviscid and viscous flow computations applicable to geometrically complex aircraft configurations

Publishable Synthesis Report of the FASTFLO project

T. Berglind* (FFA), J.W. van der Burg (NLR), K.M.J. de Cock (NLR),

W. Fritz (EADS-M), G. Kretzschmar (IBK), N. Kroll (DLR),

D. Schwamborn (DLR), M. Sillen (SAAB)

* in alphabetical order

Division: Fluid Dynamics
Issued: July 2001
Classification of title: Unclassified

Summary

A joint European initiative to develop hybrid grid based CFD-technology with a short problem-turnaround time applicable to complex aircraft configurations has been undertaken. In the context of the Brite-Euram fourth framework programme two projects have been executed, namely FASTFLO I (1996-1998) and FASTFLO II (1998-2000). The objective of this publishable summary report is to provide an overview of the approach and main achievements in these projects. The practical outcome of the joint development has been CFD technology with inviscid (FASTFLO I) and viscous (FASTFLO II) flow modelling for complex aircraft configurations. The resulting CFD technology is based on the hybrid grid approach (Ref. 7); this approach combines prismatic grid generation in a layer near aerodynamic surfaces with tetrahedral grid generation in the remainder of the flow domain.

It has been concluded that the hybrid grid based CFD technology (both viscous, and inviscid flow models) meets the following two industrial requirements:
1. The CFD problem-turnaround time is within the order of a week for geometrically complex aircraft configurations and,
2. The CFD technology is able to provide a sufficient accuracy of the aerodynamic entities (like pressures, lift, drag and moments).

Through the judicious introduction of hybrid grid generation techniques (that allow a higher level of automation in comparison with the more commonly used classical multi-block grid generation techniques) and the introduction of enhanced physical modelling in the CFD system a large spectrum of fluid flow problems for complex aircraft configurations can now be analysed within a problem-turnaround time of the order of a week. Three examples of industrially relevant aerodynamic problems with complex geometries are:
- Viscous flow analysis to assess the maximum lift capability of high-lift configurations at wind tunnel and flight conditions (at a high Reynolds number) with engines running and flaps and slats deployed.
- Viscous flow analysis to determine stability and control derivatives of real fighter aircraft including engine inlet and exhaust jet with different down-loadings, e.g. pylon(s), pod(s), store(s) and fuel tank(s) at high-loaded flow conditions.
- Viscous flow analysis of civil transport configurations for engine-airframe integration studies to minimise engine/airframe interference for new Ultra-High Bypass Ratio engines. Viscous flow modelling studies for engine inlets and outlets.

The introduction of highly automated grid generation algorithms has enabled viscous flow studies of a wider class of aerodynamic problems that which can not be tackled by standard

CFD technology because time constraints are not met by the more commonly used multi-block grid generation technology.

Economic implications of enabling hybrid grid based CFD-technology in the aerodynamic design process are manifold. Due to the realisation of a short-turnaround time the following cost savings have been realised:

- Enhanced modelling capability to engineer aerodynamic solutions resulting in a minimisation of business risks and costs associated with the introductions of new aircraft models.
- Large reduction in labour costs due to application of highly automated grid generation.
- Reduction of costs due to reduction of the number of intermediate models in an aerodynamic design cycle (first-time-right)

In addition an increase in acceptance level of hybrid grid CFD technology in European aerospace industries (SAAB and EADS-M) can be observed.

It is observed that a high interest at the FASTFLO partners is observed to exploit the results of the FASTFLO I and II projects. A further exploitation of the hybrid grid based CFD technology is foreseen in civil transport projects, military aircraft projects, and in European R&D projects, e.g. Eurolift, Taurus and Hirett.

The FASTFLO projects are an important activity towards the development of an industrial production code. However, the accurate and efficient calculation of viscous high Reynolds number flows is still a subject of current international workshops in general to evaluate multi-block structured, overset, and hybrid grids. An example of such an international workshop is the AIAA CFD Drag Prediction Workshop (Ref. 10).

Another outcome of the research conducted in the FASTFLO projects is that standardisation efforts (such as the CGNS-initiative as an emerging ISO standard) are crucially needed in Europe in order to reduce cost associated to exchange, preparation and modification of geometrical information (as needed for wind tunnel modelling and CFD geometry definition) for application of CFD-technology.

Contents

(34 pages in total

This page is intentionally left blank.

1 Introduction

Aerodynamic design and development of civil and military aircraft is observed to rely on Computational Fluid Dynamics (CFD) analysis tools to an increasing extent as time goes on. These tools are seen as complementary to wind tunnel tests and flight tests. Computational methods can provide better insight into specific aerodynamic features. For example they can give a better insight in installation effects of geometry components, such as engine(s), flaps, slats, flap tip devices, bay doors, weapons, fuel tanks and pods on the flowfield. CFD tools can be used to assess the aerodynamic characteristics of an aircraft in terms of aerodynamic coefficients such as lift, drag and moments, and to examine the behaviour of aircraft under aerodynamic high loaded conditions.

CFD tools can be deployed to optimise aerodynamic performance of aircraft. This is achieved by extending and integrating CFD codes into multidisciplinary processes. Examples are: fluid-structure coupling, aero-elastics coupling with flight mechanics and flight control, coupling with preliminary design methods to allow multidisciplinary optimisation, coupling to aero-acoustic codes to predict aero-acoustic fatigue for example.

Ever improving supercomputer technology has facilitated the development and application of more advanced flow models thereby contributing to an enhanced aerodynamic insight. Nevertheless the scale of application of CFD tools compared to wind tunnel measurements is relatively limited (Ref. 3). Increasing computational resources will allow the investigation of a larger spectrum of aerodynamic configurations in a fixed time frame.

In the area of Computational Fluid Dynamics many initiatives and co-operations have been and are undertaken to improve, develop and integrate new CFD technology for aerodynamic design and analysis.

The objective of this publishable summary report is to provide an overview of the approach and main achievements in the projects FASTFLO I and II.

2   Requirements of aerospace industry

2.1   Motivation and potential of CFD

The European aerospace industry faces a multitude of crucial business and industrial challenges if it is to respond effectively to market opportunities arising from the continuous growth in demand for air transport. Reduced costs, faster aircraft development and reduced design risks are critical factors for competitive advantage in the changing world of aircraft design (Ref. 4).

In addition forecasts as issued by the commercial aerospace industries foresee a steady growth of air traffic and replacement of ageing aircraft over the next 20 years. To remain competitive on the international airliner market aerospace companies are under pressure to change continuously to more cost efficient development of new aircraft and derivatives. This implies that key CFD technologies for improved aerodynamic design and analysis are needed to reduce development costs and by speeding up the aircraft development cycle.

Furthermore, development and incorporation of CFD in an aerodynamic design process should be focussed on reliance (Ref. 5). High reliability is a prerequisite before CFD technology can be embedded in larger processes and high throughput is needed to explore a larger number of design concepts. Important is that assured quality is offered presupposing an efficient verification and validation process and user support.

2.2   Detailed industrial requirements on CFD

For CFD technology to have an impact on the aerodynamic design of aircraft the first requirement to be satisfied is that the CFD-problem-turnaround time should be in the order of a day to a week or less. Aerodynamic analysis is a process of looking at a significant number of flow conditions for more than one geometric variant, so that a large number of flow computations have to be made. If the application of CFD codes does not yield results at this industrial time scale the impact on aerodynamic design will be reduced.

A second requirement that needs to be satisfied by CFD tools for the development of commercial transport aircraft is high accuracy of predicted aerodynamic forces such that the computed aerodynamic coefficients (lift, drag, pitching moment) can be relied upon to reduce risks in aircraft design. This second requirement translates for example into better turbulence models, and extreme grid resolution or automatic adaptive grid generation if the first requirement is also to be satisfied simultaneously.

2.3   Goal of the FASTFLO projects

In view of the above the two projects have been initiated and a joint European development on a common hybrid grid based CFD technology has emerged.

The objective of the research conducted in both projects (FASTFLO I: 1996-1998 and FASTFLO II: 1998-2000) was focussed on the development of a fully automated CFD system based on the Reynolds-averaged Navier-Stokes equations applicable to complete aircraft configurations, e.g. aircraft with engines and high lift systems.

In the first two-year project, FASTFLO I, the inviscid system has been developed. The follow-on project, FASTFLO II, concentrated on extending the system with a viscous capability and to further increase the automation level and performance of the system.

The CFD system is based on hybrid grid technology. This technology, that combines automatic prismatic grid generation near aerodynamic surfaces (to resolve the boundary layer) with an automatically generated tetrahedral volume grid (to represent the remainder of the flow domain), has been selected because it allows a higher level of automation than classical multi-block based CFD (see Ref. 7).

3 Current status of CFD code developments in Europe

Computational Fluid Dynamics has matured to the point at which it is widely accepted as an indispensable tool for aerodynamic design. However, despite all advances achieved over the last years, CFD applications can suffer from deficiencies in accuracy, robustness and efficiency, such as viscous flow predictions of complex aircraft. In recent years considerable research has been devoted world-wide to overcome the key problems of CFD which are insufficient computational resources for advanced physical models such as Direct Numerical Simulation (DNS), long set-up times and high manpower costs.

In Europe each aircraft company and research institute possesses its own proprietary CFD system. Most of the industrial production codes are based on structured multi-block mesh technology. This grid approach has been proven to be well suited for viscous high Reynolds number flows and the numerical algorithms for structured grids have reached a high level of maturity. The block-structured CFD systems are validated for a wide range of applications and they are currently used in the aircraft design phase. The codes are either a result of in-house developments or of projects on national (e.g. FLOWer-Code) or bi-lateral scale (e.g. ENFLOW-Code). An overview of proprietary production codes available at aircraft industries and research institutes within Europe is given in table 1.

Multi-block code	Organisations
CANARI, ELSA	ONERA
ENFLOW	NLR, CIRA, Fokker
EURANUS	FFA, VUB
FLOWer	DLR, Airbus Deutschland, EADS-M
NSMB	CERFACS, Airbus France
MULTNAS	SAAB
RANSMB	Airbus UK
SAUNA	DERA
ZEN	CIRA

Table 1: Overview of proprietary production codes based on multi-block technology available at aircraft industries and research institutes within Europe.

The functionality and basic algorithms of these multi-block based codes are largely the same. Main differences exist in the turbulence models implemented. Currently considerable research

effort is devoted to the development of more accurate and robust turbulence models for industrial applications. The technical limitations of the multi-block approach lies in the generation of structured multi-block grids, which in itself is a major challenge in case of complex aircraft configurations, and is recognised as a very time-consuming process. In the context of structured grids, various techniques (e.g. overlapping grid technique, Cartesian grids) are recently being developed to shorten and automate the grid generation process. However, it should be clearly stated that block-structured methods currently do not satisfy the requirements expressed in section 2.

This has led to strong development efforts of the unstructured hybrid grid CFD technologies. In contrast to structured meshes, this strategy substantially simplifies the treatment of complex geometries. Unstructured grid generation algorithms have a large potential with respect to automation level. They enable the generation of grids for complex aircraft configurations within hours or days. On the other hand, unstructured hybrid grids may complicate the design of accurate and efficient solution algorithms. In the past, most of the research organisations and industries within Europe initiated the development of unstructured CFD algorithms. The development and validation activities have been carried out on national, bi-lateral and European level. An overview over the available codes used for industrial applications is given in table 2.

Unstructured code	Organisations
AETHER	Dassault-Aviation
AIRPLANE	EADS-M, Princeton University
EDGE	FFA, SAAB
FASTFLO	NLR, DLR, FFA, SAAB, EADS-M, IBK
FILTE3D	Airbus UK
TAU	DLR, Airbus Deutschland, EADS-M
SOLAR	Airbus UK
UNS3D	ALENIA

Table 2: Overview of unstructured/hybrid codes available at aircraft industries and research institutes within Europe

For inviscid flows promising and flexible unstructured methodologies have been developed. Their capability to shorten simulation turn around time has been impressively demonstrated for a wide class of complex applications. However, the accurate and efficient calculation of viscous high Reynolds number flows using unstructured meshes requires the introduction of a prismatic

layer to resolve turbulent boundary layers and results in hybrid meshes which are subject of intensive research on national and multi-lateral level. The FASTFLO projects have been important activities to initiate research into the associated CFD technologies and leading to the development of industrial production codes.

4   The outcome of the FASTFLO I and II projects

In the FASTFLO I and II project it has been concluded that the CFD problem-turnaround time for Euler computations and Reynolds-averaged Navier-Stokes computations is within the order of one week starting from a CFD-geometry for a complex aircraft configuration (Refs. 8,9).

For Euler computations it was shown that the CFD problem-turnaround time is in the order of one week for an AS28G wing-body-pylon-nacelle configuration, a F22 fighter aircraft and the ALVAST high-lift model with flaps and slats deployed and an ultra-high-bypass-ratio engine installed.

Turnaround times for the most geometrically complicated aircraft configurations considered in the FASTFLO II project are listed in Table 3.

Aircraft configuration	CFD problem turnaround time
SAAB Gripen fighter	4 days
X31 test aircraft	5 days
A3xx wing-body	7 days
Wing alone, wing-body	1-2 days
X38 re-entry vehicle	within 7 days

Table 3 Measured CFD problem-turnaround time for a viscous flow computation for the selected aircraft configurations in the FASTFLO II project

Due to the introduction of highly automated hybrid grid generation algorithms and viscous flow solver algorithms the major workload for carrying out a viscous flow calculation has been shifted from grid generation towards CFD geometry modelling and aerodynamic post-processing. CFD geometry modelling and CAD repair have become more important and more visible due to a higher level of automation in grid generation enabling CFD application to more complex geometries. This means that sufficient time and care has to be spend to decide on the geometric features and fidelity of an aerodynamic configuration.

For viscous flow computations comparable accuracy to multi-block technology has been demonstrated for: X31-wing, DLR F4 wing-body, ONERA M6 wing, DLR F6 wing-body-pylon-nacelle, RAE 2822 airfoil, L1T2 three element airfoil and an ALVAST high lift configuration. It is concluded that the same accuracy as in multi-block based technology can be achieved provided that a number of requirements are taken into account

- Sufficient grid resolution (grid adaptation) and grid structure (anisotropy in both wall-normal and wall-tangential direction) is adopted. Careful tuning and design of the hybrid grid is still required.
- CFD geometry is modelled with sufficient accuracy.
- The proper turbulence model and transition line(s) have been selected.

Comparable accuracy to experimental data is also demonstrated. See for instance results of M2155 wing, DLR F4 wing-body, Aerospatiale A-airfoil, AS28G-wing-body, RAE 2822 airfoil.

Compared to multi-block technology the hybrid grid approach has the advantage of: a more flexible grid generation process, a high parallel efficiency, due to the application of a load-balanced grid partitioning algorithm that has no restrictions due to block-sizes and automatic local adaptation for suitable grid resolution.

Although viscous flow results have been illustrated for many aircraft configurations, particularly for wind tunnel conditions, it should be made clear that application of hybrid grid technology to for instance high-Reynolds number flows for a complete aircraft configuration remains to be demonstrated.

The accurate computation of high Reynolds number viscous flows for example for a high-lift configuration where the flow topology becomes more complex, e.g. multiple physical phenomena occur such as transition, separation (leading edge, shock wave induced), reattachment, slip lines, requires a large number of grid points. Apart from improvements in flow modelling algorithms, there is also a need to further improve the grid quality.

One way to further improve the accuracy is to efficiently distribute these nodes so that the respective physical phenomena are captured. To resolve the geometrical curvature, the slip lines, the finite trailing edges and the physical dominant effects poses special requirements on the distribution of the grid nodes. For example a slip line could be captured in the grid structure. Isotropic grids lead to an intolerably large number of grid nodes and therefore anisotropic grids would be needed to limit the number of nodes.

In this context it should be mentioned that similarly as for multi-block based CFD systems turbulence modelling remains a major stumbling stone.

5   An overview of the functionality of the common CFD system

To achieve a high level of automation in the flow simulation process, the CFD technology chosen is based on unstructured grid technology. The advantage of using unstructured grid technology compared to structured multi-block is obvious when using a complex geometry or a model with large geometry variations. Comparing numerical results from a structured multi-block grid and an unstructured grid show the same level of accuracy. An often-noticed drawback with flow solvers using unstructured grids is the relative high computational cost. For a given flow condition the cost is estimated to be a factor 2-4 larger compared to the structured multi-block approach. This difference is attributed to the use of indirect addressing and a decreased efficiency due to usage of an explicit residual-smoothing algorithm.   The incorporation of directional implicit residual smoothing could further improve the efficiency of the unstructured CFD technology as shown in (Refs. 12,13).
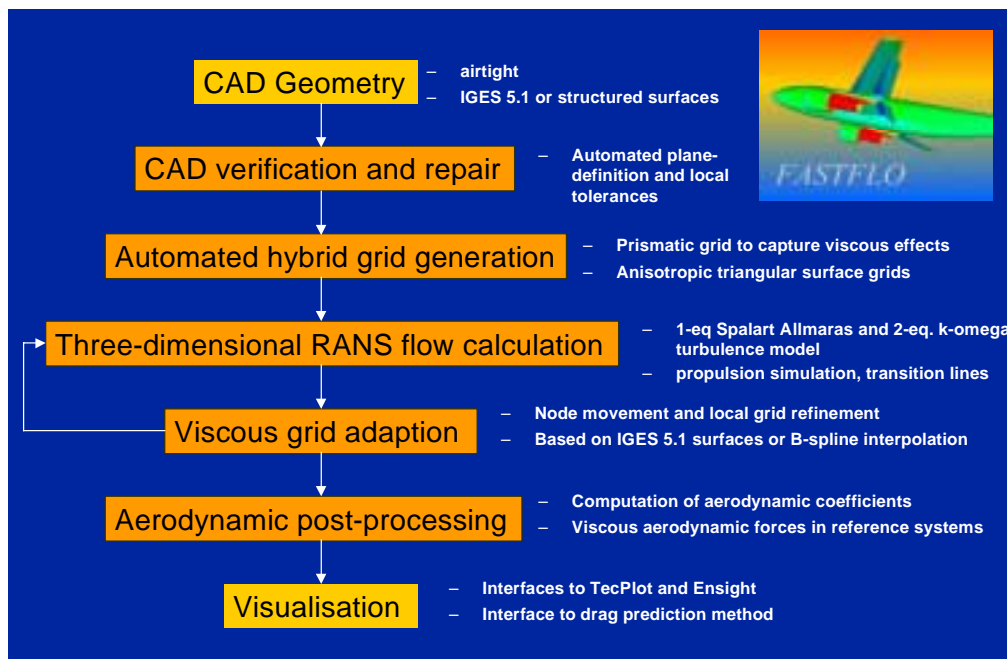

Figure 1: Overview of the algorithmic components in the CFD system.

The CFD system presented here consists of several algorithmic components grouped into a hybrid grid generator and a viscous flow solver. In the following section the main features of the system are described. Figure 1 shows an overview of the algorithmic components in the system.
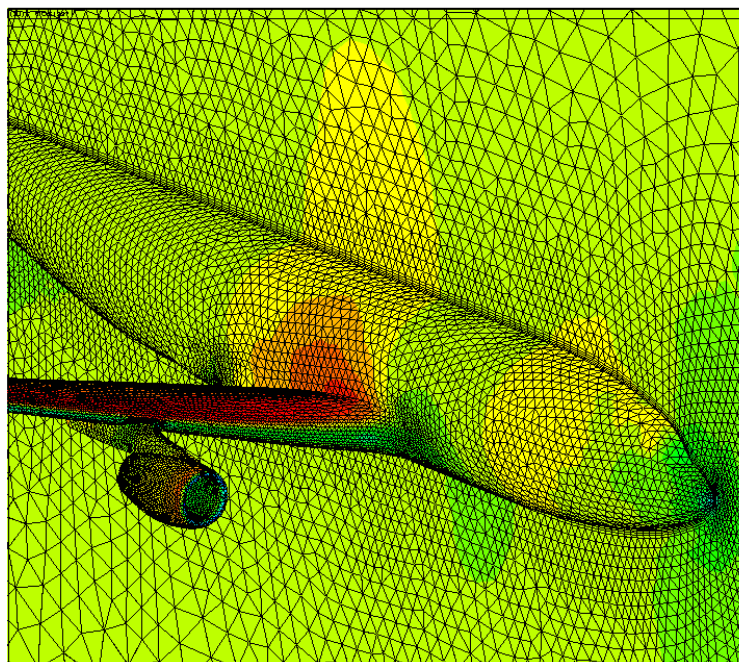
Figure 2 Illustration of a hybrid grid having a layer of prismatic elements near the aerodynamic surface for a civil aircraft.

5.1 Hybrid grid generation process

The starting point for the hybrid grid generation is a CAD model described in an IGES 5.1 data format. In the first step the IGES file is verified and the topology of the model is computed, i.e. how the individual curves and surfaces in the model are connected. This is done based on a tolerance that can be locally adjusted to account for mismatches in the CAD geometry.

The surface triangulation is performed with an advancing front method. The initial grid spacing can be controlled by user-defined point, line and triangle sources or by an automatic curvature-based algorithm.

To resolve the gradients in a (laminar/turbulent) boundary layer the CFD system uses a prismatic grid in the vicinity of solid surfaces. A predefined number of prisms are grown from the solid boundary, based on the surface triangulation and the wall normal direction. Outside the prismatic grid the tetrahedral grid is generated by an algorithm called "positive volume approach". The advantage of this approach compared to the Dalaunay approach is a 30% memory reduction with sustained grid quality and only moderately increased CPU-time. In the transition region between the prismatic grid and the tetrahedral grid additional points can be introduced to improve the mesh quality for highly stretched grids.
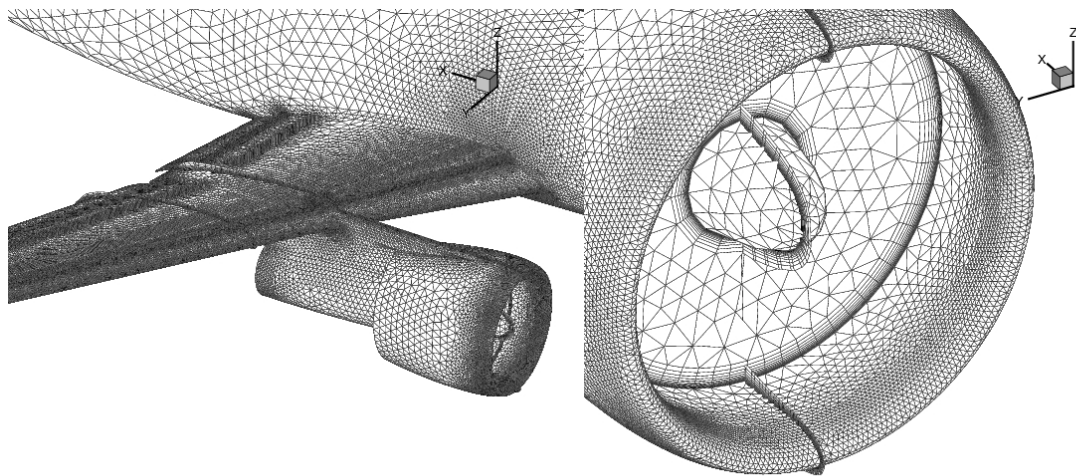
Figure 3: Example of a hybrid grid for a geometrical complex high-lift configuration with flaps and slats deployed and an ultra-high-bypass-ratio engine installed. A number of slices show the layer of prismatic elements representing the boundary layer (left figure). The presence of the layer of prismatic elements can also be observed at the inflow plane of the engine (right figure).

5.2   Reynolds-averaged Navier-Stokes solver

The flow solver is developed based on a finite volume approach, allowing for the use of grids with mixed types of elements, e.g. tetrahedra, prisms, pyramids and hexahedrals. The discretisation of the Reynolds-averaged Navier-Stokes equations is vertex-based. The finite volume dual grid cells are computed in a pre-processing step that also performs the agglomeration for multi-grid, the domain decomposition for parallel execution and the edge colouring for improved processor performance. To account for turbulence different versions of the Spallart-Allmaras and the k-ω turbulence models are implemented.

For increased accuracy in regions with large gradients grid adaptation is employed. The grid adaptation is based on local grid refinement and node movement. New surface nodes introduced in the adaptation step are projected onto the original CAD surface to ensure an accurate representation of the geometry. Grid adaptation will reduce the cost factor of unstructured/hybrid methods compared to structured multi-block codes in which adaptive schemes are quite difficult to realise.

Large viscous flow fields are often difficult to analyse and visualise on desktop computers. A post-processing tool is developed to efficiently compute aerodynamic coefficients, extract data at user-defined sections or parts of the geometry and to provide an interface to commercial visualisation software packages as EnSight and TecPlot.

To provide the aerospace industry with a CFD system that is efficient and has a carefree handling several steps are taken. The different algorithmic components are closely connected and integrated using non-proprietary software. Data transfer is arranged using the netCDF-library that ensures binary compatibility between different computer platforms. One significant workflow improvement introduced during the project is a reduction of input parameters. By using default values, which can easily be changed, the required user input to set up a complete flow simulation is kept to a minimum. The different steps taken to improve the workflow have resulted in a CFD system with a high level of automation, which makes it a powerful tool in the aerodynamic design process.

The software is written in FORTRAN90 and C and runs on several serial and parallel platforms, e.g. SGI, Sun, Cray, NEC and Compaq. For the CFD system to be efficient for large aerospace applications special attention has been paid to memory usage and computational performance. This has led to the introduction of a new algorithm for the generation of the tetrahedral grid that significantly reduces the memory requirement. The often very time-consuming flow simulation part is made efficient through parallellisation by domain decomposition and special grouping of data to ensure high performance on different types of processors. The technique of grouping data is called edge colouring and sorts the edges in the pre-processing step to allow for a high utilisation of the processor performance on both vector processors and cache-based processors.

The CFD system has been applied to a number of different configurations. One example is the simulation of flow around a civil jet aircraft. The scope of the investigation was to evaluate the influence of small geometric changes on the fuselage. Around 140 surface patches describe the aircraft geometry. In figure 4 is the pressure coefficient on the aircraft surface presented for a transonic case in sideslip.
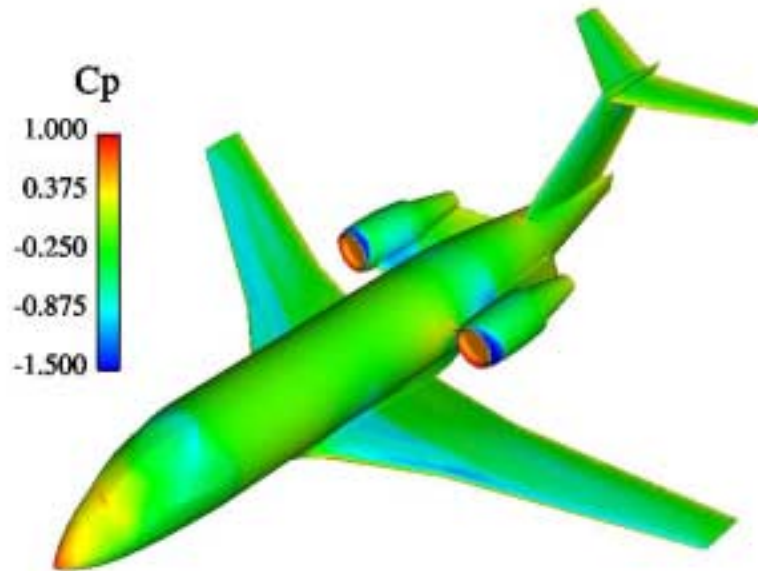
Figure 4: Pressure coefficient at Mach=0.80, angle of attack $\alpha=0^o$ and sideslip angle $\beta=5^o$.

6 Viscous flow analysis

6.1 Viscous flow computations for a X31 test aircraft

During the past 10 years, there have been made a lot of (mainly structured) calculations within different national and international working groups in order to validate CFD tools for different flow types. In case of vortical flow, it turned out, that the primary effects (vortex generation at geometric discontinuities, shock induced vortex generation, vortex break down) are qualitatively correctly predicted by Euler methods. In comparison with the experiments, Euler methods overpredict the leeward suction peaks and because of the missing (viscous) secondary separations they also predict a more inboard position of the primary vortices. By this, a validation of Euler methods by a comparison of the results with experimental data will fail and a validation should better be done by the assessment of the total pressure losses, which can only be generated by shocks or by geometric discontinuities as for example a sharp leading edge. The magnitude of such total pressure losses depends very much on the grid resolution and on the quality of the scheme. In case of vortical flow, the total pressure losses should be limited to the primary vortex and to the thin region of the contact discontinuity starting at the sharp leading edge. In all the other regions, the total pressure losses should be zero or at least as small as possible. Such an assessment of the quality of a FASTFLO Euler-solution, which has been

compared with the solution of the Jameson-type FLOPLANE code for a generic F-22 configuration turned out, that the FASTFLO Euler-solution for such a complex configuration was of a very good quality.
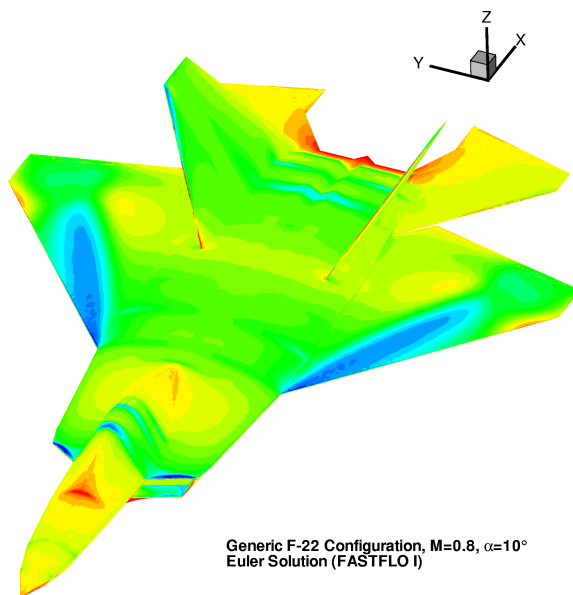

Generic F-22 Configuration, M=0.8, α=10°
Euler Solution (FASTFLO I)

Figure 5 Surface pressure contours for a generic fighter aircraft (F-22) at M=0.8, α=10° (Euler-solution).

Figure 5 shows the surface pressure contours of an Euler-solution about the generic F-22 aircraft at transonic speed. The typical characteristics of this configuration can already be seen in this Euler-solution: the burst of the leading edge vortex, the suction peaks (which are terminated by shocks) at the canopy and also the suction peaks in the channel between the twin tails. By this, Euler methods are meanwhile widely in use in the preliminary design process for the investigation of different configurations. The restriction to Euler methods results in a very short-turnaround time, which is mainly given by the grid generation. In the CFD system, the fully automated grid generation renders such short-turnaround times for Euler calculations.

Reynolds averaged Navier-Stokes (RANS) methods can give a very good agreement with experimental results. Preconditions therefore are high-resolution grids and suitable turbulence models. The first prerequisite implies large computational effort, but by massive parallel computing this is no longer a restriction today. A more severe problem is the accuracy and the reliability of the turbulence model, which concentrates on two main problems: first the tendency to numerical instability of the two equation models because of the large and rapid variations of the production terms and second the physical restriction of the turbulence model itself. It turned

out, that for example the different versions of the standard k-ω models predict an unnatural high production of turbulent kinetic energy in the core of the primary vortex. By this, the primary vortex is smeared out and especially at high angles of attack the effects of vortex break down are predicted completely wrong. Taking into account this effects, RANS methods can be considered as well validated for vortical flow up to moderate angles of attack (α=15° to α=20°, depending on the leading edge sweep).

RANS methods are applied during the final design process in all cases, where viscous effects are no longer negligible (detailed lift and drag prediction, high lift calculations, vortical flow fields about configurations with round leading edges). A second application of RANS methods, which becomes more and more important, is the detailed flow analysis about complex configurations in order to fix clearly interference effects, to find out the reasons for pitch and roll instabilities and to improve the performance of existing fighter aircraft. This requires of course the modelling of complex geometric details like inlet, flaps etc., which can be realised in the most cases only by unstructured methods.
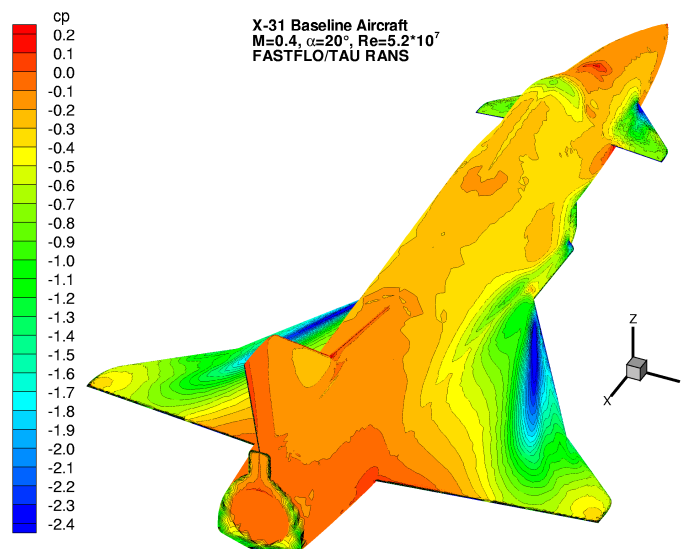

Figure 6 Surface pressure contours for the X-31 test aircraft at M=0.4, α=20° (Turbulent viscous flow solution).

Figure 6 shows a Reynolds-averaged Navier-Stokes solution obtained by the CFD system. At the wing, the pressure contours show very clearly the path of a leading edge vortex. As the configuration has a round, blunt leading edge, this vortex is generated purely by viscous effects and it cannot be predicted by Euler methods. For such flow field analysis, the CFD technology is meanwhile accepted as the only tool, which can provide a detailed insight into complex flow

fields at low cost. Especially in unsteady flow problems, the CFD technology helps very much to understand the physics of unsteady flow fields.

6.2   Verification of turbulence models

Two turbulence models are implemented in the FASTFLO flow solver, the Spallart-Allmaras one equation model, and the k-ω turbulence model. Verification of the turbulence model has been performed in the FASTFLO II project by comparing computational results with other flow solvers and with experimental data. The verification and improvement of turbulence models is still a matter of extensive research.

One of the verification test cases in the FASTFLO II project was turbulent flow around a two-dimensional profile, namely the Aerospatiale A airfoil at Re=2.07$\times$10^6, M=0.15, α=13.3° (see Figure 7). Another verification of the turbulence models may be found in (Ref. 16). The flow was tripped on the pressure side at 30% of the chord from the leading edge. On the suction side the transition was free. The flow is characterised by a separation bubble behind the suction peak, which triggered transition to turbulence. The boundary layer on the suction side is subject to an adverse pressure gradient, leading to separation at approximately 82% of the chord. It has been shown in many investigations that many turbulence models have difficulties to predict this separation.

The flow solvers used to verify the implementation are EURANUS, a code developed by FFA and VUB in Brussels, and the commercial flow solver Fluent.
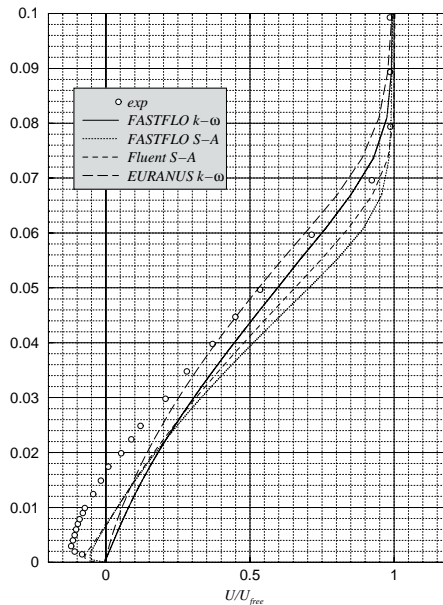
Figure 7 Comparison for different turbulence models of velocity profile at 96% chord for the Aerospatiale A-airfoil.

In the computations it was decided to impose turbulent flow in the complete flow domain, due to difficulties with the treatment of laminar/turbulent regions in some of the codes used in the validation study. Also a vortex correction imposed on the far-field boundary condition was omitted for the same reason.

At 96% chord the flow is separated according to measurements and all computations. The velocity profile using the Spallart-Allmaras model for FASTFLO flow solver and Fluent both show separation. The velocity profiles agree reasonably well. Likewise, the velocity profiles for the k-ω model with the EURANUS code and FASTFLO also agrees well. None of the velocity profiles computed using k-ω turbulence model predict separation. In this case the Spalart-Allmaras model shows a better behaviour than the k-ω model.

The AIAA Applied Aerodynamics Technical Committee sponsored a CFD Drag Prediction Workshop on June 9-10, 2001 in Anaheim, California (Refs. 10,11,15). The objective of the workshop was to assess the state-of-the-art computational methods as practical aerodynamic tools for aircraft force and moment computation with the emphasis placed on prediction of drag polar and drag rise. Participants in this international workshop performed computations over a transport-type wing/body configuration (DLR-F4 model) for a range of Mach numbers and

angles-of-attack. The model geometry had been tested in three European transonic facilities with the results documented in an AGARD publication. Multi-block structured, overset, and unstructured hybrid grids were provided by the organising committee to the participants to perform baseline cruise-point calculations (required cases). Participants were also encouraged to develop their own grids and apply their respective best practices to perform additional computations for a Mach sweep and drag rise matrix (optional cases). A statistical framework, more common to analysis of wind-tunnel results, was used to compare the computational results from all participants.

The workshop has been attended by 51 individuals from the US, Canada, Europe, and Brazil. Solutions computed by the 20 participants used 15 different flow solvers, a variety of turbulence models, and several user-generated grids. At the cruise point, the standard deviation of CFD solutions was approximately 20 drag counts (0.0020), whereas the standard deviation in the wind-tunnel data was approximately 5 drag counts. A similar European-only workshop, held earlier in the 1990's, used the same geometry and resulted in a standard deviation of approximately 50 drag counts. The current workshop and its results generated significant interest among the participants who generally desired follow-on work to this workshop. As a result a special session at Reno 2002 on CFD Drag Prediction and a second workshop is envisioned for June 2003 in conjunction with the APA Conference in Orlando, Florida.

7 Supercomputer technology

The development of computer chips in the past two decades has closely followed Moore's Law, which states that, the circuit density on a chip doubles every 18 months. Thus chip performance has increased by two orders of magnitude per decade. On average we have also seen an almost identical increase in sustained performance of (multi-processor) supercomputer systems, which is remarkable since over the time the top-of-the-line supercomputer systems have developed from shared memory vector systems with at most a few processors to massively parallel systems (MPP) with distributed memory and many – often standard workstation-type – processors.

While MPPs focus on the very high end of performance, the symmetric multiprocessing systems (SMP) of various workstation manufacturers are targeted to the lower and medium market segments and gained great popularity. Their price/performance ratio is better because their design has no overhead for support of the very large configurations and because of economies of scale in their production.

As an additional aspect, it seems that distributed computing might become an alternative to traditional high-performance computing based on central supercomputers. In this case, computing resources of many distributed low or medium performance computers like the above mentioned SMPs are bundled to clusters collaborating to solve a given problem by parallel processing.

Nevertheless, supercomputers are still needed for certain classes of problems. This is also true for CFD, where even the vector-parallel computer with a rather limited number of processors (like those by Fujitsu, NEC and Hitachi) is still indispensable for the near future. While daily problems are often efficiently solved on clustered workstations or on SMPs even in industrial routine, there exist always some leading-edge problems like e.g. unsteady fluid-structure interaction for the complete aircraft, which cannot be solved on these systems at all or in a realistic time frame.

"The future is to parallel processing". This is the belief of the US government and the US hardware companies, which has lead to the ASCI initiative and the development of other highly parallel machines that populate the upper regions of the top-500 list of supercomputer given out by the University of Mannheim twice a year. To get a little more than one Tflop/s of Linpack performance ASCI Red for instance needs more than 9,000 processors. There are, however, only few applications that really run efficiently on these type of massively parallel processors. Most day-to-day parallel applications scale well on a few (perhaps 8 to 64) processors. Especially, large industrial codes as those used for instance in structural analysis or CFD do not

parallelise well beyond that number of processors. While CFD codes based on structured data are often well optimised for vector computer they are often difficult to optimise for a parallel system with many processors. In contrast, codes based on an unstructured approach need more effort to vectorise well, while parallelisation by domain decomposition is easier and very efficient as could be shown in the FASTFLO projects for the flow solver. A linear scaling of the flow solver has been demonstrated in the FASTFLO II project (for viscous flow) for a NEC SX5 supercomputer using 1-8 processors.

Unstructured CFD technology is very suited for massive parallel processing. As for instance was shown in (Ref. 14) an unstructured inviscid flow solver can achieve a sustained floating point rate of 0.227 Tflop/s on an "Asci Red" Intel machine using 3072 processors. Sufficient attention has to be paid to data motion complexity and the reuse of data positioned closest to the processor.

In (Ref 12) a good scalability of unstructured grid technology has been illustrated for a high-lift configuration. A sustained performance of 82 Gflops was demonstrated on a Cray T3E-1200e using 1450 processors for viscous flow computations on an unstructured grid having approximately 25 million grid points.

8   Industrial implications of the FASTFLO results

The main factor behind the better economy of the CFD system, compared to the classical multi-block technology, is the application of a higher level of automation of the grid generation. Economic implications of enabling hybrid grid based CFD-technology in the aerodynamic design process are manifold. Due to the realisation of a short-turnaround time the following cost savings have been realised:

- Enhanced modelling capability to engineer aerodynamic solutions resulting in a minimisation of business risks and costs associated with the introductions of new aircraft models.
- Large reduction in labour costs due to application of highly automated grid generation.
- Reduction of costs due to reduction of the number of intermediate models in an aerodynamic design cycle (first-time-right)

Due to the involvement of aerospace industries in an early stage of the development of the CFD technology the acceptance level of the underlying technology has been increased.

The advantage of the hybrid grid technology is also the flexibility in grid generation that makes it possible to study a number of geometric variants of an aircraft within a reasonable frame of calendar time. Due to the enhanced flow modelling capability of the CFD system this flexibility will also contribute to better designs of aircraft (or components). In such a way new aerodynamic concepts (flap designs, engines) with an improved functionality (reduced noise, fuel-economic) can be developed and integrated in an aircraft.

Another indirect benefit of the CFD system is that due to the high level of automation of the CFD system multidisciplinary design optimisation with enhanced flow physics modelling becomes feasible for geometrically more complex aircraft configurations.

To facilitate the integration in larger processes it is an industrial requirement to further reduce cost associated to the exchange of CFD-related data. A possible solution for a standard exchange format for CFD related data is the CFD General Notation System or CGNS (see Ref. 6). Following parties currently support the initiative to develop the CGNS standard: United Technologies Research Center, Boeing Phantom Works, ICEM CFD Engineering, Rolls Royce/Allison, Boeing Rocketdyne, Boeing Reusable Space Systems. The specific purpose of CGNS is to provide a standard for the exchange of CFD data associated with the numerical solution of the equations of fluid dynamics. The intent is to facilitate the exchange of CFD data between sites, between application codes, across computing platforms, and to stabilise the archiving of CFD data.

9   Application of CFD technology at a SME

Application of CFD technology at a Small Medium sized Enterprise (SME) is, in principle, not very different from its application at huge companies. Nevertheless, due to the different personal structure, the impact of the availability of modern CFD software is perceived in a somewhat different way. In general, engineers in SMEs are in charge of a multitude of problems at the same time. Therefore, they cannot work continuously on a single project for months. The savings in turnaround time produced by up-to-date CFD technology, irrespectively of cost savings, make solutions to problems in fluid dynamics possible that formerly could not be seriously envisaged due to the lack of long-time availability of staff.

To IBK's experience, CFD technology is very seldom applied in producing SMEs. Indeed, its application requires specialised personal whose CFD skills are maintained at high level by permanent use of this technology. This is usually not possible at these enterprises with the consequence that this specialised personal is not available at all. When producing SMEs are occasionally confronted with a problem in fluid dynamics whose solution requires the application of CFD technology, they co-operate with one of the few engineering offices that are specialised in development work in general and in CFD in particular. As a matter of fact, CFD applications at SME level are nearly exclusively performed by engineering offices. Typically, their staff is composed of 5 to 20 persons.

For such small offices it is strictly inconceivable to develop by their own such sophisticated software like a CFD package. They must rely upon commercially available software. A certain number of CFD packages can be found on the market but none of these has been especially designed to compute the whole flow field around aircraft configurations. With the FASTFLO I (EULER) and II (NAVIER-STOKES) packages, this gap is now closed. This fact is of utmost importance to IBK whose activities are devoted to 80 % to Airbus and other European aircraft developments due to the increasing tendency of huge industrial companies to outsource part of development work to small specialised partners. Thus, the availability of the FASTFLO codes opens new horizons to IBK. As a first consequence, IBK could successfully apply for the performance of many CFD applications within the EU-funded VELA project that is partially funded by the European Commission. IBK's partnership in this project will again reinforce its position as a service provider for the European aeronautical industry.

The FASTFLO codes appear at a moment when the necessary hardware is also available at prices that engineering offices of the mentioned size can afford.

There is one more important aspect concerning the application of up-to-date CFD technology at SMEs. It is connected to the development of high performance light aircraft and gliders. The longer the turnaround time to solve a CFD problem, the higher the cost and the longer the time to market. Consequently, until now – to IBK's knowledge – no light aircraft or glider has ever been designed or aerodynamically optimised by computing the whole flow field by means of CFD. In this area also, the availability of the FASTFLO codes opens new horizons for the European manufacturers – all SMEs - of these small machines. This is particularly true for glider manufacturers who are confronted with the permanent necessity to increase the lift to drag ratio and to improve the flight characteristics of their products in order to maintain their competitiveness. It is an extraordinary challenge to increase the L/D ratio over the current value of about 60 and it is to be expected that this will only be possible in the future with the use of CFD assisted optimisation processes. Here again, IBK is in a strong position as a service provider thanks to the availability of the FASTFLO codes. A first co-operation with a glider manufacturer has already started.

With the FASTFLO project an important step forward has been achieved to the very benefit of the whole European aeronautical industry – huge and small manufacturers and small service providers. This effort should be continued in order to extend the current CFD possibilities to new fields like unsteady aerodynamics. These are needed to solve aero-elastic problems and to cope in particular with non-linear problems by solving them in the time domain. In the current state of industrial technology, non linear problems must be linearised and treated in the frequency domain.

In IBK's opinion, the FASTFLO project has started a new era in CFD applications to aircraft design, it must be considered as a progress step that should be followed up.

10 Conclusions

Hybrid grid based CFD technology, combining prismatic grid generation near aerodynamic surfaces with automatic tetrahedral volume grid generation, has emerged as an important tool for aerodynamic analysis and design because it allows a higher level of automation than classical multi-block CFD technology. Main focus of the FASTFLO projects has been:
- To achieve a short CFD problem-turnaround time for inviscid (FASTFLO I) and viscous flow (FASTFLO II) analysis of a complex aircraft configurations and
- To realise a high accuracy of the computed aerodynamic entities such as pressure distributions and lift, drag & moment coefficients.

A short CFD problem-turnaround time for complex aircraft configurations has been realised through the judicious introduction of hybrid grid generation techniques that allow a higher level of automation in comparison with the more commonly used, conventional multi-block grid generation techniques. Through the introduction of enhanced physical modelling in the CFD system a large spectrum of fluid flow problems for complex aircraft configurations can be analysed within a short turnaround time. Such as for example: viscous flow analysis for high-lift configurations at wind tunnel and flight conditions with engines running, and flaps & slats deployed e.g. for the computation of maximum lift.

Through the introduction of highly automated grid generation algorithms it has become possible to apply viscous flow models to solve a wider class of aerodynamic problems which could not be solved by standard CFD technology because time constraints are not met by the more commonly used multi-block CFD technology.

The FASTFLO I and II project have been successful projects in view of the fact that a high interest of the FASTFLO partners can be observed to exploit the outcome of the projects. Even a further and wider exploitation of the hybrid grid based CFD technology can be foreseen in European projects and military projects.

The FASTFLO projects have been an important activity towards the development of an industrial production code. However, the accurate and efficient calculation of viscous high Reynolds number flows is still a subject of current international workshops in general to evaluate multi-block structured, overset, and hybrid grids (Refs. 10,11).

CFD has matured to a level where it is accepted as a key tool for aerodynamic design and analysis. The development of the CFD technology in the FASTFLO I and II projects have contributed to this. An important aspect more in general is that CFD-technology employed in an

aerodynamic design process should have a high throughput allowing to explore a large number of design concepts. This presumes a high automation level. The developed CFD technology (due to an increased level of automation) does provide this opportunity to arrive at a better aerodynamic design in the same period (in comparison to conventional multi-block technology). Nevertheless, the scale of application of CFD tools compared to wind tunnel measurements is still limited (see also Ref. 3).

Large-scale application of CFD will enable aerospace industries to reduce the number of aerodynamic design cycles and create the possibility to develop innovative aero-products. In a future outlook it can be foreseen that CFD technology is truly embedded in the aerodynamic design processes.

Standardisation of the exchange of CFD-related data (such as the CGNS effort as an emerging ISO standard) is crucially needed in Europe in order to reduce cost associated to exchange, preparation and modification of geometrical information as needed for wind tunnel modelling and CFD geometry modelling.

11 References

1. J.W. van der Burg, B. Oskam, "FASTFLO-Automatic CFD system for three-dimensional flow simulations", NLR TP 97556 U, also in: Nouvelle Revue Aeronautique Astronautique, Vol., No. 2, April 1998.

2. J.W. van der Burg, K.M.J. de Cock, S.P. van der Pijl, "Turnaround time and accuracy evaluation of viscous flow computations on hybrid grids", in Proceedings of the European Congress on Computational Methods in Applied Sciences and Engineering, ECCOMAS 2000, Barcelona, September, 2000.

3. Jameson, "Re-engineering the design process through computation", Journal of Aircraft, Vol. 36, No. 1, January-February 1999.

4. P.E. Rubbert, "CFD and the changing world of airplane design", In ICAS Proceedings, 19th Congress of the International Council of the Aeronautical Sciences, September 1994.

5. P.G.C. Herring, "Landscape Document: An assessment of products, business drivers and aerodynamic technologies, issue number 1", Targeted Research Action in Aerospace Aerodynamics TRA3, November 1999.

6. D.M.A. Pourier, R.H. Bush, R.R. Cosner, C.L. Rumsey, D.R. McCarthy, "Advances in CGNS database standard for aerodynamics and CFD", AIAA-2000-0681, Reno, January 2000.

7. J.W. van der Burg, J.E.J. Maseland, B. Oskam, "Development of a fully automated CFD system for three-dimensional flow simulations based on hybrid prismatic-tetrahedral grids", in proceedings of the 5th International Conference on Numerical Grid Generation in Computational Fluid Dynamics, Starkville, Mississippi, April, 1996.

8. Final Technical Report of the FASTFLO II project, "Turnaround time and accuracy evaluation of CFD systems with a visocus flow simulation capability for complex aircraft configurations", compiled by J.W. van der Burg, also issued as: NLR TR-2000-292, February 2001.

9. Final Technical Report of the FASTFLO project, "Turnaround time and accuracy evaluation of prismatic/tetrahedral grid based CFD systems", compiled by J.W. van der Burg, also issued as NLR TR-98197, September 1999.

10. Internet address: http://www.aiaa.org/tc/apa/dragpredworkshop/dpw.html

11. O.J. Boelens, M. Laban, C.M. van Beek and R. van der Leeden, "Accurate Drag Computation for the DLR-F4 wing/body configuration using multi-block, structured-grid CFD technology", NLR TP-2001-321, 2001.

12. D.J. Mavripilis, "Large-scale parallel unstructured mesh computations for 3D high-lift analysis", AIAA-paper 99-0537, 1999.

13. D.J. Mavripilis, "Viscous flow analysis using a parallel unstructured multigrid solver", AIAA Journal, Vol. 38, No. 11, November 2000.

14. W.K. Anderson, W.D. Gropp, D.K. Kaushik, D.E. Keyes, B.F. Smith, "Achieving high sustained performance in an unstructured mesh CFD application", NASA/CR-2000-210080, ICASE Report No. 2000-2, January 2000.

15. M. Rakowitz, D. Schwamborn, M. Sutcliffe, B. Eisfeld, H. Bleecke, J. Fassbender, "Structured and unstructured computations on the DLR F4 wing-body configuration", presented at the AIAA CFD Drag Prediction Workshop, Anaheim, June, 2001; to be presented at the 4th AIAA Aerospace Sciences Meeting, Reno, January 2002.

16. Schwamborn, D.; Gerhold, T.; Hannemann, K.; *On the validation of the DLR-tau-code*, Notes on Numerical Fluid Mechnics, Vol. 72, Vieweg, pp. 426-433, 1999.

12 List of contacts per organisation

The FASTFLO I consortium consisted of 6 partners: three research establishments NLR (co-ordinator), DLR and FFA, two aircraft manufacturing companies EADS-M and SAAB and one engineering company IBK. In the FASTFLO II consortium the University of Delft joined the partnership.

| Org.	Contact person	Tel. No.	Fax No.	E-mail
NLR	J.W. van der Burg	+31 20 511 3696	+31 20 511 3210	vdburg@nlr.nl
DLR	N. Kroll	+49 531 295 2440	+49 531 295 2320	norbert.kroll@dlr.de
FFA	T. Berglind	+46 8555 04326	+46 8555 04000	bdt@ffa.se
EADS-M	W. Fritz	+49 8960724259	+49 89 607 29766	willy.fritz@m.EADS-M.de
SAAB	M. Sillen	+46 1318 5336	+46 1318 5647	mattias.sillen@saab.se
IBK	K. Kretzschmar	+49 911 393560	+49 911 3728299	ibk-nbg@t-online.de
TU-Delft	P. Wesseling	+31 15 278 1833	+31 15 278 7209	p.wesseling@math.tudelft.nl
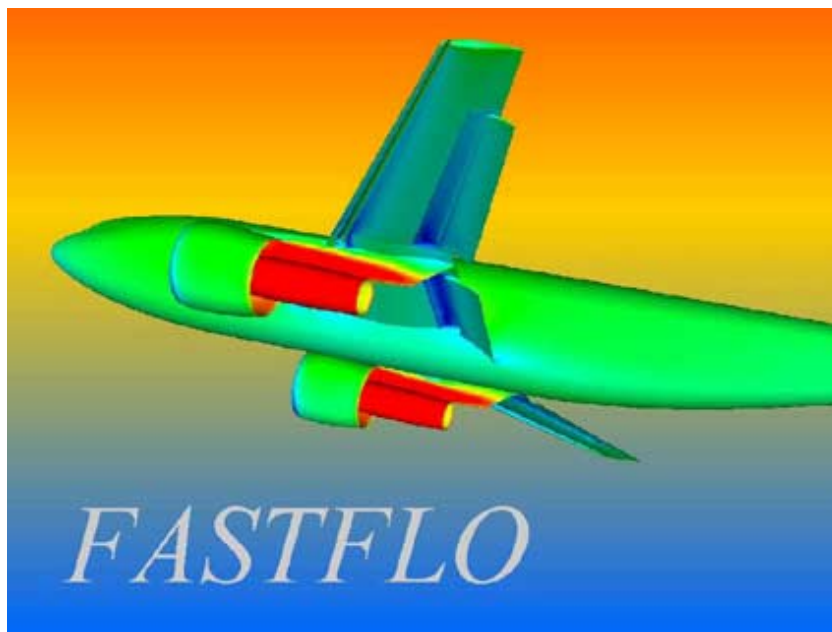
Figure 8: The logo of the FASTFLO I and II projects representing an Euler flow solution for a high lift configuration