

DOCUMENT CONTROL SHEET

	ORIGINATOR'S REF. NLR TP 96286 U		SECURITY CLASS. Unclassified												
ORIGINATOR National Aerospace Laboratory NLR, Amsterdam, The Netherlands															
TITLE ENFLOW, a full-functionality system of CFD codes for industrial Euler/Navier-Stokes flow computations															
PRESENTED AT the 2nd International Symposium on Aeronautical Science and Technology (ISASTI'96), Jakarta, Indonesia, June 24-27, 1996.															
AUTHORS J.W. Boerstoel, A. Kassies, J.C. Kok, S.P. Spekreijse		DATE 960515	pp ref 25 66												
DESCRIPTORS <table style="width: 100%; border: none;"> <tr> <td style="width: 50%;">Adaptation</td> <td style="width: 50%;">Grid generation (mathematics)</td> </tr> <tr> <td>Aircraft configurations</td> <td>Multiblock grids</td> </tr> <tr> <td>Applications programs (computers)</td> <td>Navier-Stokes equation</td> </tr> <tr> <td>Complex systems</td> <td>Numerical flow visualization</td> </tr> <tr> <td>Computational fluid dynamics</td> <td>Turbulence models</td> </tr> <tr> <td>Graphical user interface</td> <td>User requirements</td> </tr> </table>				Adaptation	Grid generation (mathematics)	Aircraft configurations	Multiblock grids	Applications programs (computers)	Navier-Stokes equation	Complex systems	Numerical flow visualization	Computational fluid dynamics	Turbulence models	Graphical user interface	User requirements
Adaptation	Grid generation (mathematics)														
Aircraft configurations	Multiblock grids														
Applications programs (computers)	Navier-Stokes equation														
Complex systems	Numerical flow visualization														
Computational fluid dynamics	Turbulence models														
Graphical user interface	User requirements														
ABSTRACT CFD technology for aerodynamic optimisation and analysis of e.g. transport aircraft and other aerospace configurations in industry is discussed. Industrial CFD technology for aerospace applications should allow 'aircraft-component analyses', to optimise aerodynamically wing/body designs, propulsion-system-airframe designs, tailplane designs, engine inlets, etc. ENFLOW consists of a modern 3D Navier-Stokes flow solver, a multiblock-grid generator, and post-processing codes for visualisation and post-processing of computation results. The ENFLOW Navier-Stokes flow solver is based on well-proven numerical technology. Central-difference methods, multiblock grids, and Runge-Kutta multigrid time-like integration are suitable. the flow solver is provided with turbulence models having a proven applicability record. Criteria for acceptability are a good prediction of pressure distributions, when boundary layers have appreciable cross flow, or when there is (incipient) separation due to strong adverse pressure gradients and shocks. Other acceptability criteria are a good prediction of pressure distributions when there are vortices or turbulent exhaust jets, etc. It is shown that a large variety of 2D and 3D flow domains can be efficiently covered by a multiblock grid. Multiblock grids are nowadays the standard grid technology in industrial CFD. Industrial CFD provides obviously excellent oppurtunities to limit design cycle times and to realise flow design targets precisely, when its possibilities as design tool are fully used.															

NLR TECHNICAL PUBLICATION

TP 96286 U

ENFLOW, A FULL-FUNCTIONALITY SYSTEM
OF CFD CODES FOR INDUSTRIAL
EULER/NAVIER-STOKES FLOW COMPUTATIONS

by

J.W. Boerstael, A. Kassies, J.C. Kok and S.P. Spekreijse

This investigation has been carried out under a contract awarded by the Netherlands Agency for Aerospace Programs (NIVR), contractnumber 01105N.

NIVR has granted NLR permission to publish all or portions of the contents of this report as paper IAST075, presented at the 2nd International Symposium on Aeronautical Science and Technology (ISASTI'96), Jakarta, Indonesia, June 24-27, 1996.

Division : Fluid Dynamics

Prepared : JWB/ ~~AK~~ AK/ ~~AK~~ JCK/ ~~AK~~ SPS/ ~~AK~~ *sp*

Approved : BO/ ~~BO~~ *BO*

Completed : 960515

Order number: 101.123/526.004

Typ. : RC



Contents

Abstract	5
1 Introduction	5
2 Examples of flow computations	7
3 CFD system and computer network	8
4 Navier-Stokes flow-solver technology	9
5 Grid generation	12
6 Conclusions	16
7 References	17
18 Figures	



This page is intentionally left blank



ENFLOW, A FULL-FUNCTIONALITY SYSTEM OF CFD CODES FOR INDUSTRIAL EULER/NAVIER-STOKES FLOW COMPUTATIONS

J.W. Boerstoel, A. Kassies, J.C. Kok, S.P. Spekreijse
National Aerospace Laboratory NLR
P.O. Box 90502, 1006 BM Amsterdam, The Netherlands
phone 31-20-5113698, fax 31-20-5113210 email: boerstl@nlr.nl¹

ABSTRACT

CFD technology for aerodynamic optimisation and analysis of e.g. transport aircraft and other aerospace configurations in industry is discussed. Industrial CFD technology for aerospace applications should allow 'aircraft-component analyses', to optimise aerodynamically wing/body designs, propulsion-system-airframe designs, tailplane designs, engine inlets, etc.

ENFLOW consists of a modern 3D Navier-Stokes flow solver, a multiblock-grid generator, and post-processing codes for visualisation and post-processing of computation results.

The ENFLOW Navier-Stokes flow solver is based on well-proven numerical technology. Central-difference methods, multiblock grids, and Runge-Kutta multigrid time-like integration are suitable. The flow solver is provided with turbulence models having a proven applicability record. Criteria for acceptability are a good prediction of pressure distributions, when boundary layers have appreciable cross flow, or when there is (incipient) separation due to strong adverse pressure gradients and shocks. Other acceptability criteria are a good prediction of pressure distributions when there are vortices or turbulent exhaust jets, etc.

It is shown that a large variety of 2D and 3D flow domains can be efficiently covered by a multiblock grid. Multiblock grids are nowadays the standard grid technology in industrial CFD.

Industrial CFD provides obviously excellent opportunities to limit design cycle times and to realise flow design targets precisely, when its possibilities as design tool are fully used.

1 INTRODUCTION

Modern aerodynamic analyses and optimisations of transport aircraft and other aerospace configurations are also based on an extensive use of CFD (Computational Fluid Dynamics) means. This paper discusses key features of CFD codes suitable for this kind of work.

Industrial CFD codes, when used for analysis and design of flows around aircraft, should allow analysis of differences between computed flows around different configurations of a particular aircraft, obtained by adding/deleting components to/from the aircraft. For jet aircraft, possible added configuration components may be an engine nacelle and/or a nacelle/pylon combination. Lift losses and drag increases due to the flow disturbance effects of these configuration parts may then be computed, and corrected by reverse aerodynamic

¹Results presented were obtained under contract 01105N with the Netherlands Agency for Aerospace Programs (NIVR).

Paper IAST075, presented at the 2nd International Symposium on Aeronautical Science and Technology (ISASTI'96), Jakarta, Indonesia, June 24-27, 1996.

engineering with CFD. Other desired CFD possibilities of industrial CFD codes are: computer analyses of various closely-coupled under wing positions of large-sized jet engines; the computation of the effects of powered and unpowered jet or propeller engines on wing/body flows, to identify and correct engine-installation interference problems; computer analyses of the aerodynamic performance of flap and slat systems on wings in 3D flows; computation of flows around tailplanes, etc.

CFD systems for such work should allow the computation of a broad range of flow features. These include subsonic, transonic and supersonic flows, shock waves, boundary layers, wakes, engine inlet flows, exhaust jets, and propeller slip streams. Turbulence should be computable with various of the better turbulence models. Flow computation results should be accurate enough from industry point of view. Moreover, short turnaround times should be possible, which is of interest to realize short aerodynamic design-cycle times[45].

The major tasks, that are executed in such CFD work, are illustrated in figure 1. This figure shows the layout and the use of a particular CFD system, which has full functionality for industrial CFD[9]. This implies that there is not only a flow solver, but also all the other CFD codes required in industrial CFD work. These are codes for:

- geometry processing of complex shapes of aerodynamic configurations, not only during surface-shape defining work, but also during flow computations and in grid generators;
- generation of flow-computation grids around complex configurations;
- adaptation of grids to flow-computation results; and
- fast visualisation and postprocessing of flow-computation results.

The codes should allow the construction of computation grids, which can be given sufficiently fine resolution in high-gradient regions in the flows, so that flow-computation results are numerically accurate enough. Efficient and accurate mathematical techniques for the representation and manipulation of surfaces of complex aircraft configurations should also be available. It is further desirable that mathematical techniques for 1) grid-generation, 2) for the representation of complex surfaces, and 3) for flow computations, are as much as possible based on common principles, and are tuned to each other, if techniques have to be different.

Nowadays, a key element in CFD work is the construction of grids for flow computations. This situation may be contrasted with the state-of-the-art in CFD about ten years ago, when flow-solver construction was the central issue. However, apart from turbulence modelling (see below), during the last ten years, flow-solver technology has in many respects become more or less mature². As a consequence, attention has shifted to the application of CFD technology to flows around complex configurations (using state-of-the-art CFD technology for turbulence modelling). For complex configurations, multiblock-grid technology has been widely adopted as a basis for flow computations [3].

Through sustained grid-generation technology development over the last eight years, turnaround times for multiblock grid generation around e.g. complete aircraft have been reduced by a factor of 5 to 10. The construction of complex Euler multiblock grids of good quality (according to 1988 standards) required in 1988 three to six months [7]. Today, the generation of good Navier-Stokes multiblock grids allowing accurate flow computations (according to today standards) is a matter of a few hours (only slight configuration-shape changes during design work) to two-four manweeks, when new complex aircraft configurations have to be dealt with.

²At least on structured grids. For unstructured grids, the situation is different.

2 EXAMPLES OF FLOW COMPUTATIONS

In this section, a few typical flow-computation results will be presented, to illustrate various issues of the CFD work.

Figures 2-4 concern a Navier-Stokes computation of the flow on and around the AS28G wing/body of Aerospatiale. The grid had about 10^6 grid cells. Figure 2 shows a layer of blocks around the wing and fuselage. Each block may be seen to have the shape of a deformed cube. Figure 3 gives an impression of the computed transonic pressure distribution on and around the wing/body. There is a low shock at the upper side of the wing, at this flow condition.

Figures 5-6 concern a computation with ENFLOW of a 3D inviscid (Euler) transonic flow around an isolated DLR model jet engine [2]. Purpose of these computations was testing of the proper numerical functioning of the jet-engine facilities in the flow solver (inlet and outlet boundary conditions), and a first analysis of the flow physics in the exhaust jet. Observe that the grid is very fine. Various details of a complex supersonic exhaust-jet flow are discernible on this fine grid: shock waves, slip layers between the cold exterior jet and the inviscid outer flow, a wavy stream-tube structure of the cold jet, with acceleration and deceleration regions in the cold jet, etc.

Figures 7-10 are typical ENFLOW flow-computation results for a wing-body-pylon-jet-engine in transonic cruise flow. The configuration and grid were provided by DLR[44]. The jet engine is of large size, and is placed closely under the wing. The computed flow has shocks.

Figures 11-13 concern two-dimensional Navier-Stokes computations of the flow around the RAE2822 airfoil, which is a well-known CFD test case [14]. Purpose of these computations was testing of the accuracy of various turbulence models for boundary layers in transonic flows, with shock-wave/boundary-layer interaction present. Computation results were thereto compared to results of wind tunnel experiments [14].

For the testing of turbulence models against experimental results, it is necessary to reduce numerical errors to such low levels, that differences in computation results for various turbulence models can be attributed with absolute certainty to differences in the turbulence-model equations. This imposes very demanding requirements on the fineness and smoothness of grids. The above figures illustrate this.

Figures 21-24 give results of 3D Navier-Stokes test computations of the flow around the ONERA M6 wing, with the NLR version of the Johnson-King turbulence model[33]. The pressure-coefficient distribution on the upper side of the wing shows a λ -shock. A skin-friction vector plot (not shown here) shows that this computed boundary layer separates under the shock in the tip region. This flow is also a well-know test case in the literature, see [1]. The generation of a good Navier-Stokes grid for this configuration is also not trivial, as it requires due attention to avoid grid folding in the boundary-layer grid around the wing tip. There are elliptic methods which do avoid grid folding in this region when the grid is fine enough [53][54][52].

The ENFLOW system has been validated, by applying it also to many other aerodynamic problems. A few of them are:

- Wing, wing/body, wing/body/nacelle, and wing/body/nacelle/ pylon configurations in transonic flows: "numerical-wind-tunnel testing" with various aircraft configurations. Optimisation of wing shapes in the presence of jet-nacelle/pylon interference effects.



- Fokker F100 wing-body in transonic flows: comparison of computed wing loadings with free-flight test results [35]. ALENIA G222 propeller aircraft in sonic and transonic flow [8]: computational analysis of propeller-slipstream effects on wing and tail-plane flows.
- Estimation of aerodynamic performances of space capsules in transonic- and supersonic flows. For a multiblock grid see figures 17-20.
- The usual test cases of the CFD literature: ONERA M6 wing, DLR F4 wing, NACA0012 and RAE2822 airfoils, flat-plate flows, etc., see [1][2].

3 CFD SYSTEM AND COMPUTER NETWORK

The ENFLOW system consists of five major CFD codes. These are (see figure 1):

- A commercial software package (ICEM-CFD), for the CAD manipulation of aerodynamic surfaces of complex aerospace configurations.
- A flow solver, ENSOLV, for Navier-Stokes flow computations on multiblock grids. This flow solver is a batch code, which can be executed on workstations, as well as on conventional and vector computers. Supercomputer capacity is required for grids with more than 10^6 grid cells, if reasonable problem cycle times (3-5 hours for one accurate 3D Navier-Stokes computation) are required, and computed results should have sufficient numerical accuracy.
- Two grid generation codes, ENDOMO and ENGRID, for the graphical-interactive construction of multiblock grids.
- A grid adaptation code, ENADAP, for numerical-accuracy improvement by multiblock grid adaptation.
- Various codes for the visualisation of computation results and for aerodynamic postprocessing (like the commercial software package DATA VISUALISER).

The CFD codes should be operational on a computer network having characteristics, which make this network (also) suitable for industrial CFD work. Major technical requirements to be imposed on computer networks from the point of view of industrial CFD are as follows.

- The network should contain a modern vector supercomputer, such as a NEC-SX4 parallel computer³, to achieve acceptable short cycle times and large throughput rates for the computation of Navier-Stokes and Euler flows around complete aircraft [37].
- The computer network should also have good workstations for CAD processing of surfaces, for grid generation, and for the visualisation and postprocessing of CFD results.
- Computers and workstations should be integrated in one balanced high-speed computer network, to obtain acceptable cycle times for the transfer of files with computation results between computers, workstations, and fast, large-volume storage media.
- Remote access to the computer network for engineering CFD work should be possible.

For a recent discussion of views about the role of parallel super computing in CFD by aerospace industries, see Loeve [37]. He discusses that modern parallel vector supercomputers can supply the sustained computing power for large-volume application-oriented flow computations; multi-processor workstations and workstation clusters cannot. Aerospace industries might consider the use of remote large parallel super computing power, if they wish to do large-volumes of CFD computations, to bring the quality of aerodynamic designs and aerodynamic design-cycle times to up-to-date levels.

³Installation at NLR planned in June 1996. 16 processors, 4 Gigabyte= $500 \cdot 10^6$ real numbers of about 13 decimal places

4 NAVIER-STOKES FLOW-SOLVER TECHNOLOGY

The CFD technology in the NLR Navier-Stokes flow solver has the following specifications.

- Suitable for arbitrary, complex aerospace configurations.
- Flow equations: thin-layer or full Navier-Stokes and/or Euler flow equations.
- Turbulence models: algebraic models of Baldwin-Lomax [5], Johnson-King [22][23][24], and Cebeci-Smith [12]. Two-equation turbulence models of $k-\omega/\epsilon$ type [40][41][42][43][27] are being implemented. The turbulence models of Baldwin-Lomax and Johnson-King, as described in the literature, have been improved with more robust algorithms for the estimation of the turbulent length scales [33]. For the Johnson-King turbulence model, second-order fully-upwind discretisation methods were designed which take proper account of regions of influence and dependence of dependent variables in this turbulence model, see Kok e.a. in [33]. These regions are trivial in airfoil boundary layers, but are not trivial near attachment and separation lines in boundary layers at e.g. wing tips.
- Special boundary-condition options: jet-engine inlets, jet-engine outlets, and propellers, see Kassies and Oskam in [30][31]. A disk model for a propeller may give useful wing/nacelle flows.
- Multi-zone capabilities, based on 3D-flow-domain decomposition with blocks: Navier-Stokes equations or Euler equations defined per block; boundary conditions defined per block-subface.
- Grids: smooth structured multiblock grids, suitable for complex aerospace configurations, and for accurate, fast and reliable computations. See Spekrijse en Boerstoel in [53][54].
- Spatial discretisation: cell-centred, finite-volume, central-differencing [19][20]. Shock capturing and algorithm stabilisation: blended 2nd- and 4th-order explicit artificial dissipation [19][20]. For improved numerical accuracy of boundary-layer computations with Navier-Stokes methods, an adapted form of matrix artificial dissipation is applied, see Kok e.a. in [33]. The combination of central-difference technology with matrix dissipation is not standard. In the literature, matrix dissipation is usually combined with upwind CFD technology.
- Time iteration: multistage Runge-Kutta time stepping, accelerated by local time stepping, residual smoothing, enthalpy damping [19][20], and high cell aspect-ratio scaling of Martinelli [39]. FAS multigrid with automatic partial coarsening when required [29][32][33], and computation start-up with grid-sequencing.
- Block-coupling at block faces in the flow: special coupling algorithms, providing 2nd-order spatial accuracy, sharp numerically captured shocks, and stable time integrations, when the grid is only C^0 -continuous over block faces [28][10].

It is interesting to compare this specification, as far as the major numerical techniques are concerned, to the specifications of 86 other CFD flow-solver codes in [3]. The codes may be classified according to the kind of grids used, the kind of time integration techniques, and the kind of convergence-acceleration techniques. The results may be summarised as follows.

- Grids used by the codes (as a % of the total number of codes):
 - 45 % multiblock grids
 - 27 % monoblock grids
 - 17 % unstructured grids
 - 2 % overset grids
- Time-integration or iteration techniques in the codes:
 - 49 % have one or more explicit Runge-Kutta time integration methods

62 % have one or more implicit techniques (approximate factorisation, Gauss-Seidel, implicit Euler, Lerat, LU, etc.)

- Convergence acceleration techniques in the codes:
 - 93 % have local time stepping
 - 30 % have explicit or/and implicit residual averaging
 - 29 % have multigrid convergence acceleration
 - 17 % have enthalpy damping.

It may be concluded that multiblock-grid technology is very popular and obviously generally considered necessary for CFD production codes. Among the time-integration techniques, Runge-Kutta methods are a winner, as half of the codes may be run with this time-integration method. This is probably due to the algorithmic simplicity and the low memory requirements of Runge-Kutta methods, compared to the alternative methods. Many codes have also one or more forms of convergence-acceleration techniques, but only 29% of the codes have multigrid as convergence accelerator. Because, in Navier-Stokes flow solvers, multigrid is required if one wants good convergence characteristics, this leaves open questions in this respect. Only 22% of the codes has some form of combination of multiblock and multigrid technology. The ENSOLV code has all useful major numerical techniques incorporated.

Flow-computation results are acceptable for industrial use, if these results are sufficiently accurate. There are two pacing items to the achievement of this requirement: turbulence modelling (see below) and grid generation (see section 5).

It is interesting to address the question which turbulence models in the literature are best for use in industrial CFD codes. The list of published candidate turbulence models is very long, and includes the following ones.

- The Baldwin-Lomax [5] eddy-viscosity model. The model was developed in 1978, is very popular in CFD codes, and serves in fact as a standard turbulence model for boundary layer and wake flows. The turbulence model has been derived from that of Cebeci-Smith [12], by replacing the boundary layer thickness δ of a turbulent boundary layer by another variable that, in attached 2D boundary layers at least, can be easier computed than δ .
- The Cebeci-Smith [12] eddy-viscosity model. This model is well founded in the older turbulent boundary-layer literature.
- The Johnson-King [21][22][24], and Johnson-Coakley [23] eddy viscosity models. These models have an auxiliary differential equation, to model history effects in turbulent boundary layers due to shocks and strong adverse pressure gradients (1/2-equation models).
- The Baldwin-Barth [6] and the Spalart-Allmaras [49] one-equation turbulence models.
- The $k-\epsilon$ two-equation standard model of Jones, Launder, and Spalding [26][36]. There is a large number (much more than 10) of different improved-modified-augmented versions of the standard $k-\epsilon$ model. For recent publications, in which a larger number of different $k-\epsilon$ and similar turbulence models were analysed, see Taulbee [55] and Kalitzin, Gould, and Benton [27]. A few of these other models are the $k-\epsilon$ models with the Jones-Launder low-Reynolds-number correction [26], the Chien low-Reynolds-number correction [27], the Launder-Sharma low-Reynolds-number correction [27], and the Wolfshtein 1-equation sub-layer model [27]; the $k-\omega$ two-equation models of Wilcox [61][62][63][64][65][66] and of Menter-Wilcox (the BSL and the SST-model variants) [40][41][42][43], and the $g-\omega$ model of Kalitzin, Gould, and Benton [27].

Other classes of turbulence modelling techniques are not considered here, as they are not (yet) interesting for industrial applications. (This does not imply that they are scientifically not interesting.) Examples of such techniques are RSM (Reynolds-stress transport model) equations [17], DNS (direct numerical simulation) and LES (large-eddy simulation)

methods, and RNG (renormalisation-group) methods. A main bottleneck for progress with these turbulence models in industrial CFD codes is insufficient computing power (speed and memory).

Implementation in a particular flow solver of many of the algebraic, 1/2-, 1-, or 2-equation turbulence models would be a major effort. However, this is not necessary, because requirements to be satisfied by good turbulence models for industrial computations of subsonic and transonic flows rule out many models. These requirements are discussed here.

If a turbulence model in a flow solver should only properly predict pressure distributions on swept transport-aircraft wings, when:

- the flows are subsonic (no shocks), and
- the boundary layers are attached and moderately loaded by the pressure distributions, so that the boundary layer has a mainly two-dimensional structure (without much cross flow),

then many turbulence models will produce results of acceptable accuracy. This may already be concluded from the two-dimensional benchmark results published in 1987 by Terry Holst [18] (flows around airfoils). It is then efficient to choose for implementation in CFD codes one of the simplest models available.

However, when for example engine-airframe configurations in transonic or subsonic flows have to be aerodynamically analysed and/or designed with CFD means, it is necessary to impose more demanding requirements.

- Requirement (R): turbulence models should also allow reasonably accurate computations of pressure distributions and wing circulations, when there is cross flow or incipient separation in boundary layers, which are due to higher adverse pressure-distribution loadings, shock-wave/boundary-layer interactions, etc. Only then wing-circulation losses due to viscous effects are sufficiently accurately computed, and leading-edge flows of not only wings, but also of pylons and engine inlets have proper computed local angles of attack. Then it is also possible to reliably analyse the interference effects (which are partly strongly nonlinear) of these aircraft components, and to design, with CFD means, optimal shapes and positions of these components with respect to each other.
- Turbulence models should not contain distances to walls or wake-centre surfaces as a variable defining turbulence levels. Further, turbulence models should also not be defined with variables or scales, which depend on the idea of a boundary-layer with finite thickness. Such variables are a source of troubles in, for example, corner flows, confluent wakes, exhaust jets, and in neighbourhoods of shear layers. The computation of these distances or scales is in these cases often at least a serious organisational problem in codes, or the idea of a boundary layer is not applicable.

Other requirements which are of interest for industrial CFD work may also be formulated if desired, for example requirements which address flow separation phenomena at buffet boundaries, and flows in jets.

It is unfortunate that, under requirement (R), many turbulence models cannot be classified as useful, because the turbulence models were not tested to flows with shock waves, or they were shown to fail. See e.g. again Holst [18] (16 Navier-Stokes methods, and 7 other methods). The literature allows the following conclusions. a. From 1987 on, Johnson, King and Coakley [21][22][23] developed their 1/2-equation models. Under requirement (R), these perform better than the Baldwin-Lomax model. b. Rumsey and Vatsa apply in [46] six different turbulence models to transonic flows with shocks. The configurations were the RAE2822 airfoil, the isolated ONERA M6 wing, and the Lockheed wing C. Taking requirement (R)



as criterion, we concluded from their results that, the Johnson-King-Coakley and Spalart-Allmaras models must be preferred over the Baldwin-Lomax and Baldwin-Barth models. c. Londenberg applies in [38] four recent turbulence models (Baldwin-Lomax, Johnson-King, Baldwin-Barth, Spalart-Allmaras), applied to transonic flows with a strong shock around an airfoil, with aileron flap deflected and retracted. Londenberg concluded that the Baldwin-Barth model performed best (perhaps it is better to say the least badly). We concluded also that the Johnson-King model performed only slightly worse. d. Menter presents in a series of papers [40][41][42][43] results about the application of the Baldwin-Lomax, Johnson-King models, the $k-\epsilon$ model (Jones-Launder version) and two models of himself, which are a combination of the $k-\epsilon$ and $k-\omega$ models: the BSL (baseline) and the SST (shear-stress transport) models. The turbulence models are applied to more than ten very different flow test cases with experimental results, with very different kinds of turbulence. Among the test cases there are a few with shock waves, see [43]. The last publication allows the conclusion that, under requirement (R), Menter's models are among the better models now available⁴. e. Vivek, Papadakis and Greathouse [57] find that the $k-\omega$ and Baldwin-Barth models produce the better results, when a RAE2822 airfoil with a moderate shock in transonic flow is considered. (They also consider two other flow cases, with a flap a deflected. These are not so relevant here.) f. Kral, Mani, and Ladd [34] apply ten different turbulence models (six of them are $k-\epsilon$ or $k-\omega$ models) to two transonic flows with (strong) shocks around the RAE2822 airfoil, and to two other flows (ejector nozzle, 3D diffuser). Their results support that the Menter SST model, together with the Spalart-Allmaras model, belong to the best models, when requirement (R) is taken as criterion. g. Kalitzin, Gould, and Benton [27] present a two-equation model, which they derive from the $k-\omega$ equations by a transformation of a dependent variable. The purpose of this transformation is replacement of the $\omega \approx y^{-2} \uparrow \infty$ behaviour near solid walls by an $g \approx y \downarrow 0$ behaviour (y is the normal distance to the wall). For the two tested flow cases with shock waves, their results have at least similar quality as the results obtained with the Menter-Wilcox model.

From the recent literature it may thus be concluded, that new turbulence models seem to become available, which promise an improvement in accuracy of computed pressure distributions on wings, when there are stronger adverse pressure gradients and/or shocks. As discussed, such improvements are very welcome, to enhance the accuracy of procedures for propulsion-system/airframe designs, for example.

5 GRID GENERATION

For CFD computations of 3D flows, grids are required. CFD computations may be based on either multiblock structured grids, or on unstructured grids.

A multiblock grid may be described as a collection of blocks, with in each block a grid.

- Each *block* has the form of a deformed cube, and covers a part of the flow domain. A block has the topology of a unit cube, and has thus six block-face surfaces, twelve block-edge curves, and eight block vertex points. The blocks together should cover the flow domain completely. Moreover, a part of the block-face surfaces should cover the flow-domain boundaries completely. The covering of the flow domain by blocks may be arranged in various ways, see below for more details.

⁴It is interesting to observe that both Johnson and Menter rely on an old result of Bradshaw [11]. Menter eliminates the Boussinesque hypothesis with Bradshaw's result, when strong adverse pressure gradients are present. Johnson uses Bradshaw's result to derive his extra ordinary differential equation for 2D boundary layers. The Boussinesque hypothesis is well known to be questionable in adverse-pressure-gradient boundary layers, where strong history effects are present.

- Each block is completely covered by *hexahedral* grid cells. These grid cells are, in general, packed cell-face-to-cell-face, and are further required to form a smooth, well-ordered, three-dimensional covering of the complete block. The coordinate vector of each cell vertex is a grid point. The collection of all grid points in a block is the grid in the block. Because the grid cells in a block form a well-ordered three-dimensional covering of the block, these grid points may be stored in simple data structures, 3D arrays. The smoothness of the grid-point distribution, together with the simplicity of its data structure, are a sound basis for the construction of efficient numerical approximation and solution methods in flow solvers, which execute efficiently on parallel vector computers.
- It is ideal when a complete flow domain can be covered by one block, because then it is not necessary to construct a covering of the flow domain by blocks. One then has a *monoblock* grid. Unfortunately, a monoblock grid is only possible when the edge curves and vertex points in the boundary surface of the flow domain have the same topological structure as a unit cube. This is only the case for very simple flow domains, like that around a wing-body, see the geometry in figure 2 (not: the grid). Hence, in general, multiblock grids (or unstructured grids) are required.

An unstructured grid may be described as a covering of the 3D flow domain by tetrahedral cells. Each cell has four cell-face surfaces, and four cell-vertex points. These tetrahedral cells are usually packed cell-face-to-cell-face, and should cover the flow domain completely. The collection of all cell vertex points defines the grid. Because here the grid cells are not required to cover the flow domain in a smooth, well-ordered way, the grid points cannot in general be stored in simple data structures like 3D arrays. Sometimes, other cells than tetrahedra are used.

There are various types of multiblock grids, depending on the way of covering of the flow domain by the blocks and the grid cells.

- In *overlapped (overset)* multiblock grids, blocks, which are adjacent to each other, have a volume part of the 3D flow domain as intersection. Each intersection or overlap region is common to at least two blocks.
- In *patched* grids, adjacent blocks have a surface as intersection. Each interface surface lies in the block-face surfaces of two blocks, one at either side of the interface surface. The two grids in the two blocks induce in general two different subgrids in the common interface surface. When this is true, the grid is a patched grid.
- In C^0 -*continuous* grids, blocks with a common interface surface as intersection, have also the grid in that interface surface common.

Overlapped grids, patched grids, and C^0 -continuous grids form a hierarchy of multiblock structured grids of increasing structure. The covering of the flow domain by blocks is, with overlapped grids, in principle completely unstructured. This is, to a large extent, also true for patched and C^0 -continuous grids. However, C^0 -continuous grids have much extra structure, because such grids are continuous from block to block over common block interface surfaces, in which the subgrids are required to be common.

Multiblock grids may be considered to provide a fair compromise between the advantages and disadvantages of structured and unstructured grids. The structured grids within the blocks allow high computational efficiency and numerical accuracy of computation results, while the unstructuredness of block coverings of flow domains allow the treatment of complex aerospace configurations. These advantages of multiblock grids come with the price of increased man-hour effort, compared to the situation about ten years ago, when monoblock grids were usual. The success of block-structured-grid methods for solving viscous flows over complex aerodynamic configurations is demonstrated in various recent conferences on grid generation [60],[4],[13],[48], [59],[58]. From [3] it is also clear, that many aerospace



organisations have some type of multiblock-grid capability in their CFD codes. It may be prudently concluded, that the man-hour efforts for the construction of multiblock grids are generally accepted, and thus in fact considered acceptable. This does not mean, however, that there do not exist wishes to reduce this effort by better grid-generation technology.

It is interesting to observe from [3], that only 2% of the 98 CFD codes, reviewed in that report, can apply overlapped multiblock grids. We conclude that, for much CFD work, the added complexities in Navier-Stokes flow solvers due to the overlapped blocks, are in fact not considered acceptable, at least nowadays.

We will further consider C^0 -continuous multiblock grids only.

As shown in figure 1, the generation of a C^0 -continuous grid requires the execution of three major tasks:

- 1) CFD modelling of the surfaces of the flow domain boundaries,
- 2) domain decomposition i.e. the construction of the blocks within the flow domain, and
- 3) grid generation i.e. the construction of a structured grid within each block.

Grid generation processes start often with surface modelling of flow boundaries, using CAD-CAM software. The purpose of surface modelling is to bring the geometric definition of configuration surfaces in a form, which is acceptable for CFD work. CAD geometries have usually to be edited and/or trimmed, to correct geometric defects which are not acceptable in CFD work (i.e. gaps, or non-physical edges of surface elements, overlaps and doubly defined regions, non-physical protrusions, or other dirty matters in the surface definitions), or to modify the true geometry for the purpose of CFD analysis (i.e. remove parts of the geometry that are not relevant for CFD analysis, or which removal is required to obtain required insights about the effect of removed geometrical components). The creation of for CFD acceptable surface patches can be a time-consuming effort, requiring mastery of large, complex CAD-based systems.

When the surface modelling task is completed, the domain-decomposition task has to be executed, to obtain a block decomposition of the 3D flow domain. The flow domain is then subdivided into an unstructured collection of blocks.

Domain decomposition is an highly interactive task. An interactive block-decomposition code with an advanced Graphical User Interface (GUI), and a good workstation are needed to support a user during domain decomposition. The GUI should provide a user with a small set of powerful functions for domain decomposition. Nowadays, the interactive generation of blocks is still a somewhat time consuming and sometimes difficult part of the multiblock grid generation process. This is the case when the aircraft or aerospace configurations have complex shapes. It may be expected that this situation will improve considerably in the next few years.

Subsequently, the grid generation step can be executed, to obtain a structured grid in each block. Grid generation is also a highly interactive task, but is much more simple and less time consuming than domain decomposition. The result of a grid generation is a multiblock grid, which may be used in a flow solver to compute solutions of Euler and/or Navier-Stokes equations.

Multiblock grid generation codes should be designed such that a user is allowed to concentrate on the handling of the topology and geometry of block decompositions, and on providing sufficient grid quality. To that end, the tools for surface manipulations, block subdivision and grid generation should be automated as much as possible. The basis of high degrees of automation are good mathematical theories, which reduce information flows in codes to essentials, eliminate redundancies in definitions, combine various kinds of informa-

tion in as few topology and geometry tables as possible, while numerical methods for surface representation (interpolation methods) and for grid generation must be so robust that they require (practically) no user control for their proper functioning.

A full account of a theory is given by Spekreijse and Boerstoeel in [52][53][54].

The ENFLOW system is now being extended with a facility for grid adaptation of multiblock structured grids. This facility is needed to automatically adapt boundary-layer grids, so that it will not be necessary to construct interactively a new Navier-Stokes grid for every Reynolds number, Mach number, or incidence. Observe, that boundary-layer properties may vary greatly with Reynolds numbers, Mach numbers, and incidences, and that therefore boundary-layer grids in Navier-Stokes computations may have to be (correspondingly tuned) to obtain acceptable numerical accuracies. The grid adaptation facility will be based on the application of mesh movement, see Hagmeijer [15][16].

The figures show various examples of results of 3D multiblock grid generation. From these figures it may be concluded that, for a very wide class of aerospace CFD problems, multiblock grids of high quality (measured in terms of smoothness and well-orderedness of data), can be successfully constructed. Multiblock grid generation is obviously applicable to a wide variety of CFD problems, with very different flow-domain shapes.

6 CONCLUSIONS

Aerodynamic analyses and design optimisations of transport aircraft and other aerospace configurations may, to a large extent, also be based on the use of CFD (Computational Fluid Dynamics), because this is efficient to reduce aircraft design cycle times, and meet precisely technical design requirements (section 1). As far as transport aircraft are concerned, one may think of computational 'aircraft-component analyses', to optimise aerodynamically wing/body designs, propulsion-system/airframe integrations, tailplane designs, etc.

Systems of CFD codes for such industrial CFD work should not only contain a Navier-Stokes flow solver with a few of the better turbulence models, but also codes for aircraft surface tuning for CFD work, grid generators, flow visualisers and postprocessing. Such a system is described (sections 1, 3). Examples illustrate various aspects of this kind of CFD work (section 2).

Good Navier-Stokes flow solver technology, based on central-difference methods, multiblock grids, Runge-Kutta/multigrid time-like integration, and standard turbulence models, is available (section 4).

Modern parallel vector supercomputers can provide the sustained computing power for large-volume CFD computations, which are required to improve the precision with which aerodynamic-designs targets are realized, while at the same time aerodynamic design-cycle times are reduced, see section 3. Aerospace industries might consider the use of remote large parallel super computing power, if they wish to achieve these goals.

The turbulence models of Baldwin-Lomax, Johnson-King, and Cebeci-Smith are widely used in industrial CFD codes. They are suitable for the computation of attached, three-dimensional boundary layers, with not too much cross flow. Only very few of the many other turbulence models that are around, seem to offer real progress over these three models, if industry requirements are taken as criterion (section 4). Flow phenomena, which were found hard to predict very accurately, are pressure distributions and wing loadings, when turbulent boundary layers have appreciable cross flows, or have incipient separation due to adverse



pressure gradients (rear loading, shocks, wing trailing-edge flows) (section 4). However, better turbulence models have recently become available.

It is shown that a large variety of 2D and 3D flow domains can be efficiently covered by a multiblock grid (see figures in section 5). This includes grids around complex aircraft configurations with engines and/or tailplanes. About 80% of the CFD codes are based on the use of multiblock grids (section 4). Multiblock-grid technology is obviously widely accepted.

Industrial CFD provides obviously excellent opportunities, if it is used as tool for the optimisation of flows, around aircraft and aerospace vehicles. The benefits are, in particular, reduction of design cycle times, and precise realization of design targets.

References

- [1] AGARD, Experimental Data Base for Computer Program Assessment, AGARD Fluid Dynamics Panel Advisory Report 138 (1979)
- [2] AGARD, A Selection of Experimental Test Cases for the Validation of CFD Codes, AGARD Fluid Dynamics Panel Advisory Report no 303, Vol 1 and Vol 2 (1994)
- [3] AGARD, Computational Aerodynamics Based on the Euler Equations, AGARDograph 325, editors J.W. Slooff, W. Schmidt (1994)
- [4] Arcilla, A.S., Hauser, J., Eiseman, P.R., Thompson, J.F. (eds), Numerical Grid Generation in Computational Fluid Dynamics and Related Fields, Proceedings of the 3th International Conference, Barcelona, Spain, Elsevier Science Publishers (1991)
- [5] Baldwin, B.S., Lomax, H., Thin-Layer Approximation and Algebraic Model for Separated Turbulent Flows, AIAA-78-257 (Jan. 1978)
- [6] Baldwin, B.S., Barth, T.J., A One-Equation Turbulence Transport Model for High Reynolds Number Wall-Bounded Flows, NASA TM-102847 (1990)
- [7] Boerstael, J.W., Jacobs, J.M.J.W., Kassies, A., Amendola, A., Design and Testing of a Multiblock Grid Generation Procedure for Aircraft Design and Research, AGARD FDP Conf. on Appl. Mesh Generation to Complex 3D Configurations, Loen, Norway (May 1989), NLR TP 89146 L (1989)
- [8] Boerstael, J.W., Conceptual Design of a Computer Code System for the Calculation of Flows Around Transport Aircraft, NLR TP 92065 (1992)
- [9] Boerstael, J.W., Spekreijse, S.P., Vitagliano, P.L., The Design of a System of Codes for Industrial Calculations of Flows around Aircraft and other Complex Aerodynamic Configurations, NLR TP 92190, AIAA 92-2619 (1992)
- [10] Boerstael, J.W., ENFLOW, A System of CFD Codes for Industrial CFD Analysis of Flows Around Aircraft Including Propulsion Systems Modelling, NLR CR 93519 (1993)
- [11] Bradshaw, P., Ferriss, D.H., Atwell, N.P., Calculation of Boundary Layer Development Using the Turbulent Energy Equation, Journal of Fluid Mech., Vol 28, (1967) 593-616
- [12] Cebeci, T., Smith, A.M.O., Analysis of turbulent boundary layers, Academic Press, New York (1974)
- [13] Choo, Y.K.(ed), Surface Modelling, Grid Generation, and Related Issues in Computational Fluid Dynamic (CFD) Solutions, NASA CP 3291, Proc. of a workshop held at NASA Lewis, Cleveland, Ohio (1995)
- [14] Cook, P., McDonald, M., Firmin, M., Airfoil RAE 2822 - Pressure Distributions and Boundary Layer Wake Measurements, AGARD AR-138, A6 (1979)
- [15] Hagmeijer, R., Grid Adaptation based on Modified Anisotropic Diffusion Equations formulated in the Parametric Domain, Journal of Computational Physics, 115, 169-183 (1994)
- [16] Hagmeijer, R., Adaptive generation of structured grids. VKI Course on Computational Fluid Dynamics, March 25-29, (1996), NLR TP to be published
- [17] Hytopoulos, E., Simpson, R.L., Critical Evaluation of Recent Second-Order Turbulence Models, AIAA 84-0175 (1984)
- [18] Holst, T.L., Viscous Transonic Airfoil Workshop Compendium of Results, AIAA-87-1460 (1987)

- [19] Jameson, A., A Non-Oscillatory Shock Capturing Scheme Using Flux-Limited Dissipation, MAE Report 1653 (1984)
- [20] Jameson, A., Multigrid Algorithms for Compressible Flow Computations, MAE Report 1743 (1986)
- [21] Johnson, D.A., King, L.S., A New Turbulence Closure Model for Boundary Layer Flows with Strong Adverse Pressure Gradients and Separation, AIAA-84-0175 (1984)
- [22] Johnson, D.A., Predictions of Transonic Separated Flow with an Eddy-Viscosity / Reynolds-Shear-Stress Closure Model, AIAA-85-1683 (1985)
- [23] Johnson, D.A., Coakley, T.J., Improvements to a Non-Equilibrium Algebraic Turbulence Model, AIAA J., Vol 28 (Nov. 1990) 2000-2003
- [24] Johnson, D.A., Nonequilibrium Algebraic Turbulence Modeling Considerations for Transonic Airfoils and Wings, AIAA-92-0026 (1992)
- [25] Johnson, D.A., Menter, F.R., Rumsey, C.L., The Status of Turbulence Modeling for External Aerodynamics, AIAA-94-2226 (1994)
- [26] Jones, W.P., Launder, B.E., The Calculation of Low-Reynolds-Number Phenomena with a Two-Equation Model of Turbulence, Int. J. Heat Mass. Transfer, Vol 16 (1973) 1119-1130
- [27] Kalitzin, G., Gould, A.R.B., Benton, J.J., Application of Two-Equation Turbulence Models in Aircraft Design, AIAA 96-0327 (1996)
- [28] Kassies, A., Tognaccini, R., Boundary Conditions for Euler Equations at Internal Block Faces of Multiblock Domains, using Local Grid Refinement, NLR TP 90134, AIAA 90-1590 (1990)
- [29] Kassies, A., Amato, M., Kok, J.C., Bosse, S., Software Design of ENSOLV, a Flow Solver for 3D Euler/Navier-Stokes Equations in Arbitrary Multiblock Domains, NLR CR 93190 L (1993)
- [30] Kassies, A., Oskam, B., Simulation of Propellers in Euler/Navier-Stokes Flow Solvers, NLR CR 93537 L (1993)
- [31] Kassies, A., Inlet and Outlet Boundary Conditions for the Simulation of Engines in the ENSOLV Flow Solver, NLR CR 95088 L (1995)
- [32] Kok, J.C., Amato, M., Bosse, S., Kassies, A., Numerical Design of ENSOLV; A code for the numerical simulation of 3D flows using the thin-layer Navier-Stokes and Euler equations, NLR CR 93152 L (1993)
- [33] Kok, J.C., Boerstoel, J.W., Kassies, A., Spekrijse, S.P., A Robust Multiblock Navier-Stokes Flow Solver for Industrial Applications, paper for ECCOMAS'96 Conf., Section Computational Fluid Dynamics, Paris (Sept. 1996), NLR report to be published
- [34] Kral, L.D., Mani, M., Ladd, J.A., On the Application of Turbulence Models for Aerodynamic and Propulsion Flow Fields, AIAA 96-0564 (1996)
- [35] Kuijvenhoven, J.L. (Fokker), Boerstoel, J.W., 3D Euler flows around modern airplanes, 8th GAMM Conf. Num. Methods in Fluid Mech.s, Delft, the Netherlands (1989), NLR TP 89343 U (1989)
- [36] Launder, B.E., Spalding, D.B., The Numerical Computation of Turbulent Flows, Comp. Meth. in Appl. Mech. and Eng., Vol 3 (1974) 269-289
- [37] Loeve, W., From R&D in Parallel CFD to a Tool for Computer Aided Engineering, Invited paper in Parallel Computational Fluid Dynamics: Implementations and Results using Parallel Computers, pp 51-58, editors A. Ecer, J. Periaux, N. Satofuka, S. Taylor, Elsevier Science BV (1966) NLR TP 95382 L (1995)
- [38] Londenberg, W.K.: Turbulence Model Evaluation for the Prediction of Flows Over a Supercritical Airfoil With Deflected Aileron at High Reynolds Number, AIAA 93-0191 (1993)
- [39] Martinelli, L., Calculations of Viscous Flows With a Multigrid Method, Ph.D Thesis, Princeton Univ. (1987)
- [40] Menter, F.R., Performance of Popular Turbulence Models for Attached and Separated Adverse Pressure Gradient Flows, AIAA-91-1784 (1991)
- [41] Menter, F.R., Influence of Free-stream Values on $k-\omega$ Turbulence Model Predictions, AIAA Journal, Vol 30, No 6 (1991) 1657-1659
- [42] Menter, F.R., Zonal Two Equation $k-\omega$ Turbulence Models for Aerodynamic Flows, AIAA-93-2906 (1993)
- [43] Menter, F.R., Assessment of Two-Equation Turbulence Models for Transonic Flows, AIAA-94-2343 (1994)

- [44] Rossow, C.C., Ronzheimer, A., Multiblock Grid Generation Around Wing-Body-Pylon-Engine Configurations, in: Numerical Grid Generation in Computational Fluid Dynamics and Related Fields, ed. Arcilla, A.S., Hauser, J., Eiseman, P.R., Thompson, J.F., North-Holland (1991)
- [45] Rubbert, P.E., CFD Research in the Changing U.S. Aeronautical Industry (1995)
- [46] Rumsey, C.L., Vatsa, V.N., A Comparison of the Predictive Capabilities of Several Turbulence Models Using Upwind and Central-Difference Computer Codes, AIAA-93-0192 (1993)
- [47] Saini, S., Bailey, D.H.: NAS Parallel Benchmark Results 12-95, Report NAS-95-021 (Dec. 95)
- [48] Smith, R.E. (ed), Software Systems for Surface Modeling and Grid Generation, NASA CP 3143, Proceedings of a workshop held at NASA Langley, Hampton, VA (1992)
- [49] Spalart, P.R., Allmaras, S.R., A One-Equation Turbulence Model for Aerodynamic Flows, AIAA-92-0439 (1992)
- [50] Spekrijse, S.P., Boerstoeel, J.W., New Concepts for Multi-Block Grid Generation for Flow Domains around Complex Aerodynamic Configurations, NLR-TP-91046, in: Numerical Grid Generation in Computational Fluid Dynamics and Related Fields, Proceedings of the 3th International Conference, Barcelona, Spain, Elsevier Science Publishers (1991)
- [51] Spekrijse, S.P., Boerstoeel, J.W., Kuijvenhoven, J.L., van der Marel, M.J., Surface Grid Generation for Multi-block Structured Grids, NLR-TP-92267, in: Proceedings of the First European Computational Fluid Dynamics Conference 1992, Ch. Hirsch et al (eds), Vol 2, pp. 937-944, Elsevier Science Publishers (1992)
- [52] Spekrijse, S.P., Elliptic Grid Generation Based on Laplace Equations and Algebraic Transformations, J. Comp. Phys. 118 (1995) 38-61
- [53] Spekrijse, S.P., Boerstoeel, J.W., Multiblock Grid Generation, Part 1: Elliptic Grid Generation Methods for Structured Grids, VKI Course on Computational Fluid Dynamics, March 25-29, (1996), NLR TP to be published
- [54] Spekrijse, S.P., Boerstoeel, J.W., Multiblock Grid Generation, Part 2: Multiblock Aspects, VKI Course on Computational Fluid Dynamics, March 25-29, (1996), NLR TP to be published
- [55] Taulbee, D.B., Engineering Turbulence Models, in George, W.K., Arndt, R. (eds): Advances in Turbulence, Hemisphere Publ. Corp., New York (1989)
- [56] Vegt, J.J.W. v.d., Turbulence Models for Complex Wall Bounded Flows, NLR CR 94187 C (1994)
- [57] Vivek, L., Papadakis, M., Greathouse, J.S., Performance of an RNG Based Turbulence Model for Single and Multi-Element Airfoil Computations, AIAA 96-0164 (1996)
- [58] Weatherill, N.P.(ed), Numerical Grid Generation, VKI Lecture Series 1990-06 (1990)
- [59] Weatherill, N.P. (ed), Grid Generation, VKI Lecture Series 1994-02 (1994)
- [60] Weatherill, N.P., Eiseman, P.R., Hauser, J., Thompson J.F. (eds), Numerical Grid Generation in Computational Fluid Dynamics and Related Fields, Proceedings of the 4th International Conference, Swansea, Wales, Pineridge Press (1994)
- [61] Wilcox, D.C., Reassessment of the Scale-Determining Equation for Advanced Turbulence Models, AIAA Journal, Vol 26, No 11 (1988) 1299-1310
- [62] Wilcox, D.C., Multiscale Model for Turbulent Flows, AIAA Journal, Vol 26, No 11 (1988) 1311-1320
- [63] Wilcox, D.C., Supersonic Compression-Corner Applications of a Multiscale Model for Turbulent Flows, AIAA Journal, Vol 28, No 7 (1989) 1194-1189
- [64] Wilcox, D.C., Turbulence Modeling for CFD, DCW Industries Inc., California (1993)
- [65] Wilcox, D.C., A Two-Equation Turbulence Model for Wall-Bounded and Free-Shear Flows, AIAA-93-2905 (1993)
- [66] Wilcox, D.C., Comparison of Two-Equation Turbulence Models for Boundary Layers with Pressure Gradient, AIAA Journal, vol 31, No 8 (1993) 1414-1421

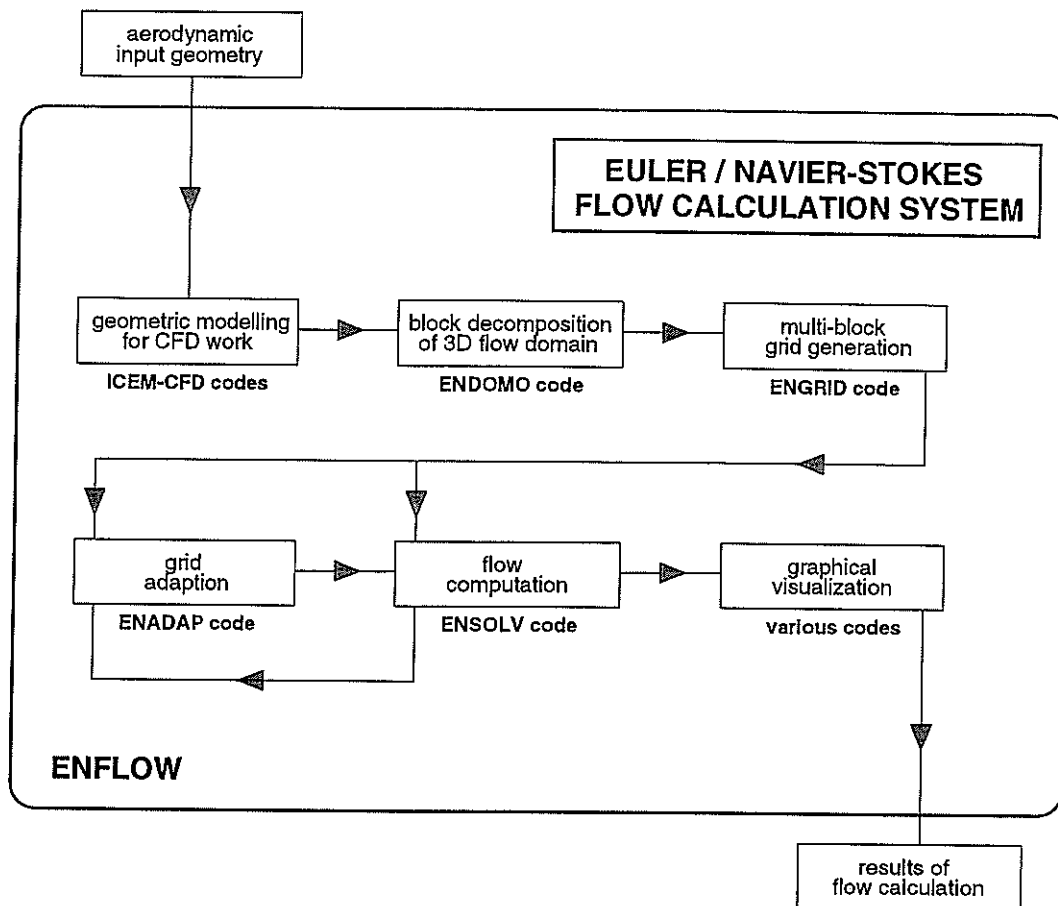


Figure 1: A CFD system with full functionality.

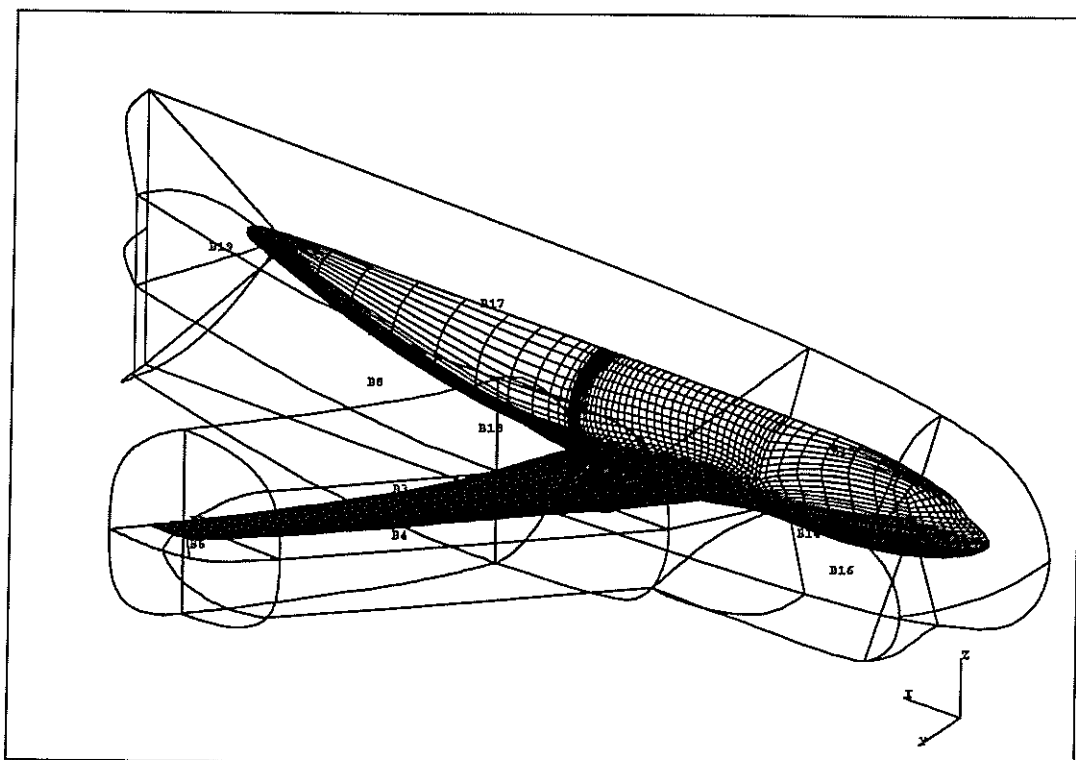


Figure 2: Block decomposition around AS28G wing/body, produced with ENDOMO.

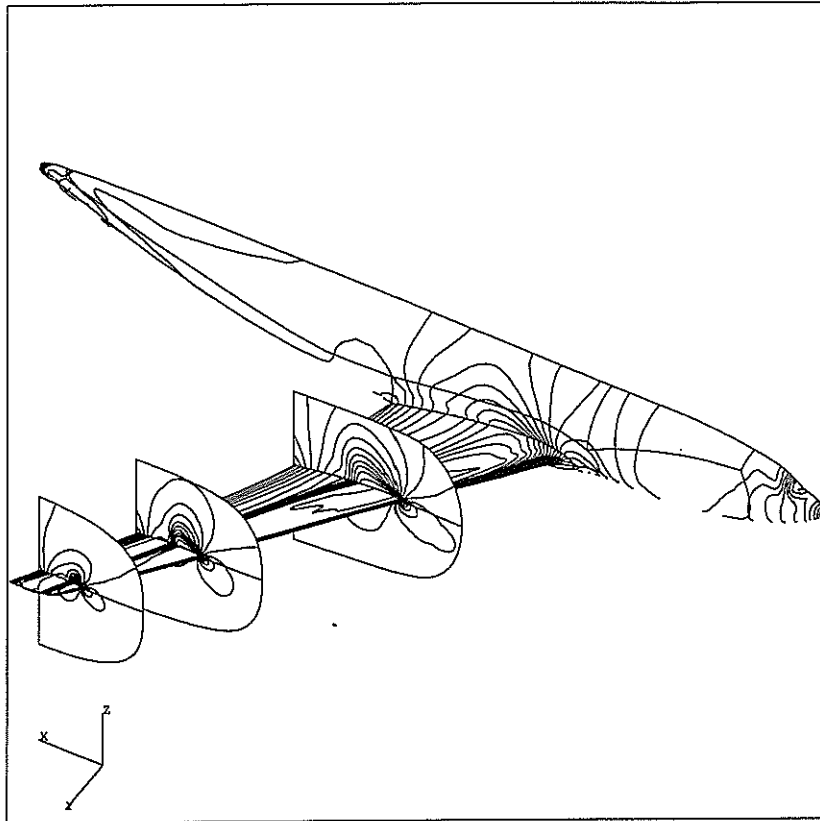


Figure 3: Navier-Stokes pressure distribution on and around AS28G wing/body. Computed with ENSOLV.

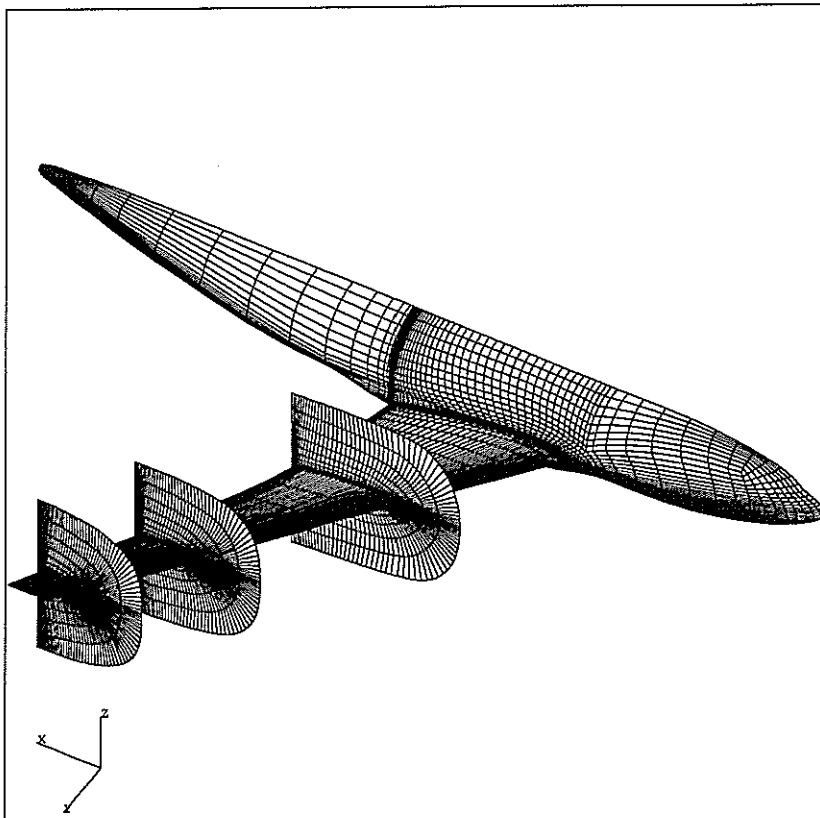


Figure 4: Medium grid on and around AS28G wing/body. Fine grid has all mesh sizes halved. Computed with ENGRID and ENDOMO.

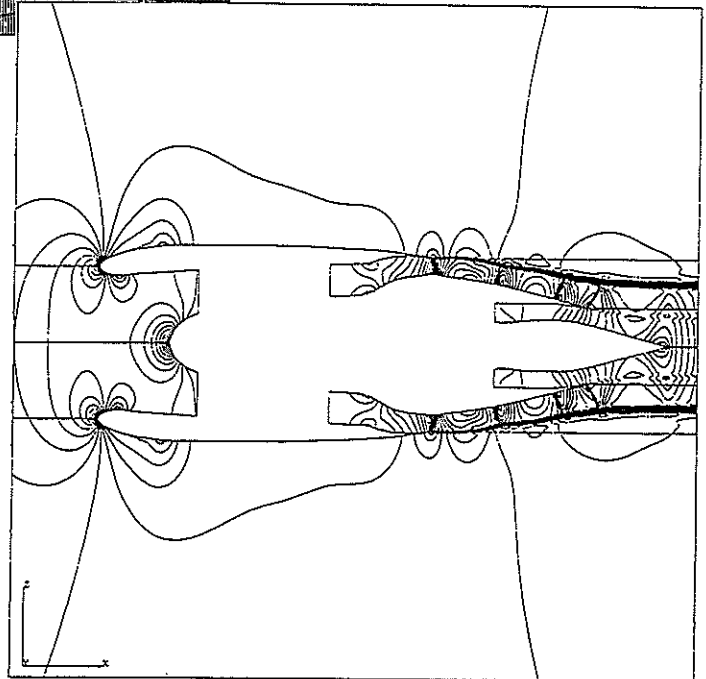
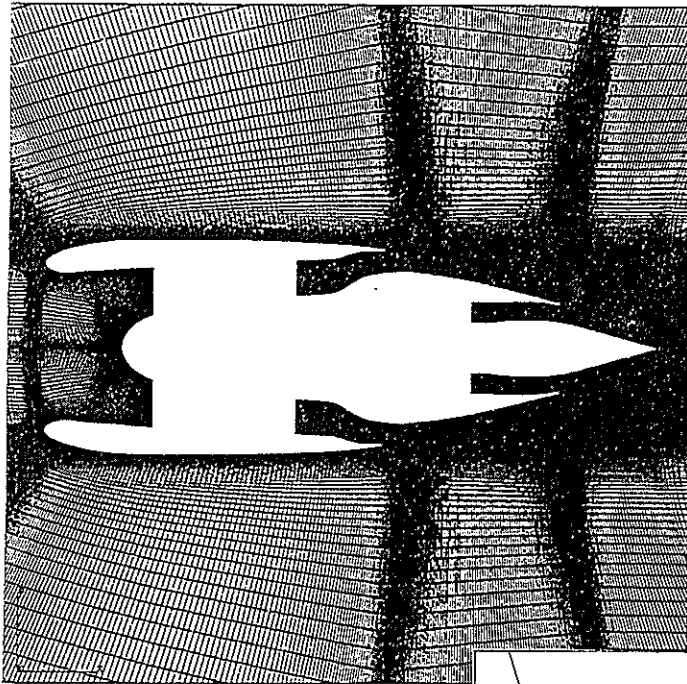


Figure 5: Part of fine NLR multiblock grid in vertical symmetry plane of DLR model jet engine.

Figure 6: Lines of constant Mach number in vertical symmetry plane of DLR model jet engine in transonic inviscid flow. Shock waves in supersonic cold jet. Slip layer between supersonic cold jet and inviscid exterior flow.

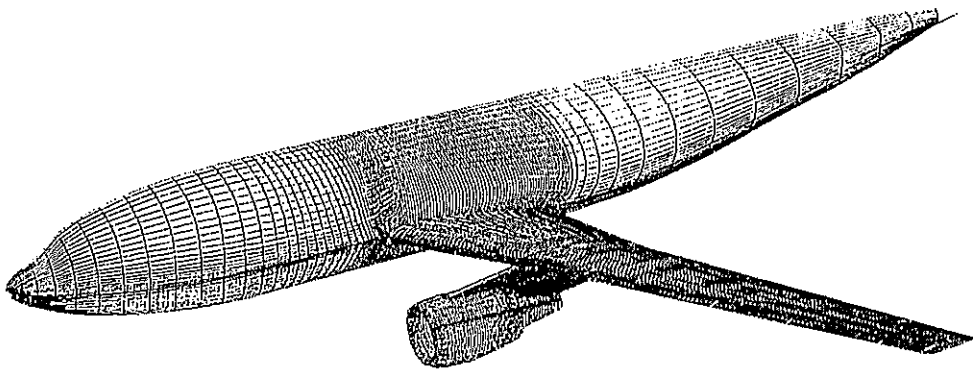
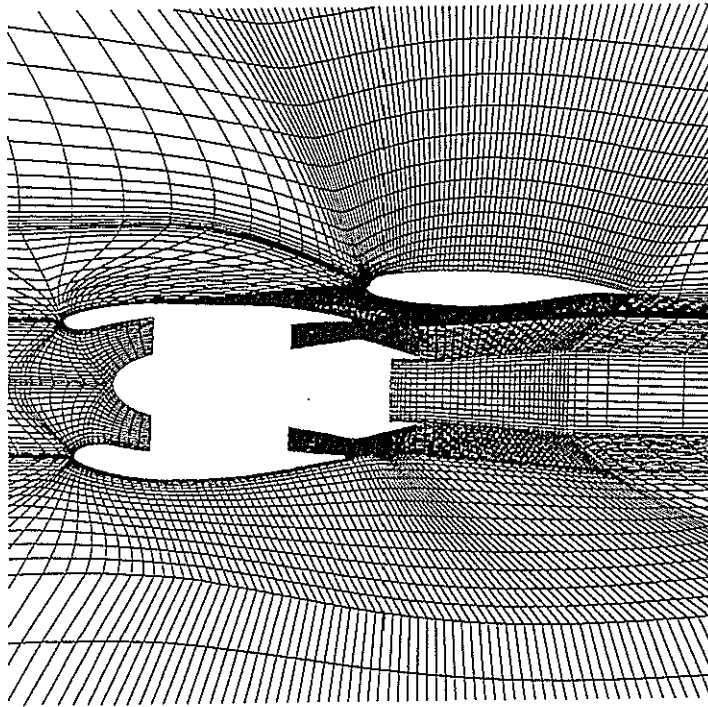


Figure 7: Multiblock grid in approximately vertical surface through VHBR-engine center line of a DLR WBPE (wing-body-pylon-engine) configuration. Multiblock grid provided by DLR.

Figure 8: DLR WBPE (wing-body-pylon-engine) configuration. VHBR (very high bypass ratio) engine. Close coupling to wing.

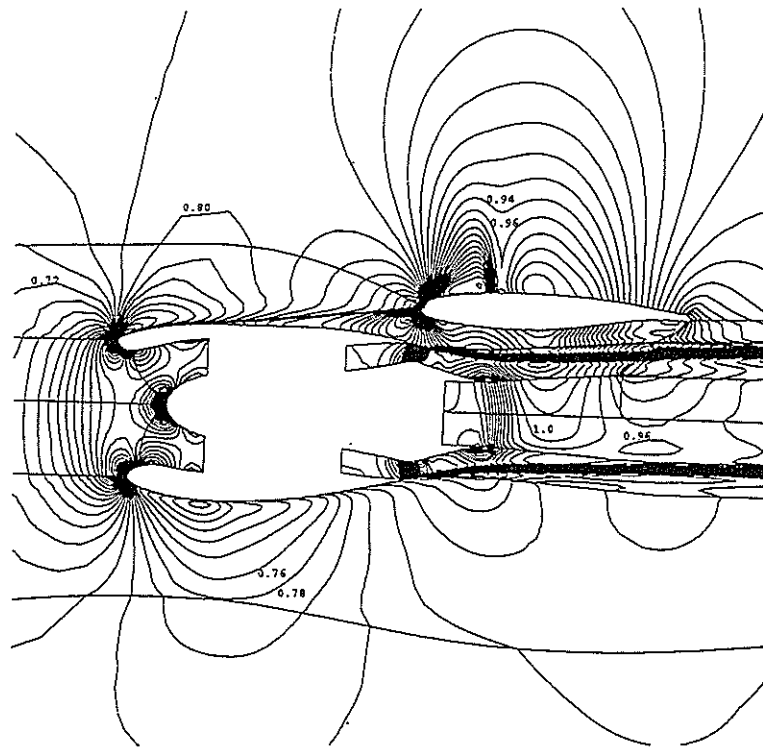


Figure 9: Lines of constant Mach number in approximately vertical surface through VHBR-engine center line of DLR WBPE configuration. NLR ENSOLV flow computation on DLR multiblock grid.

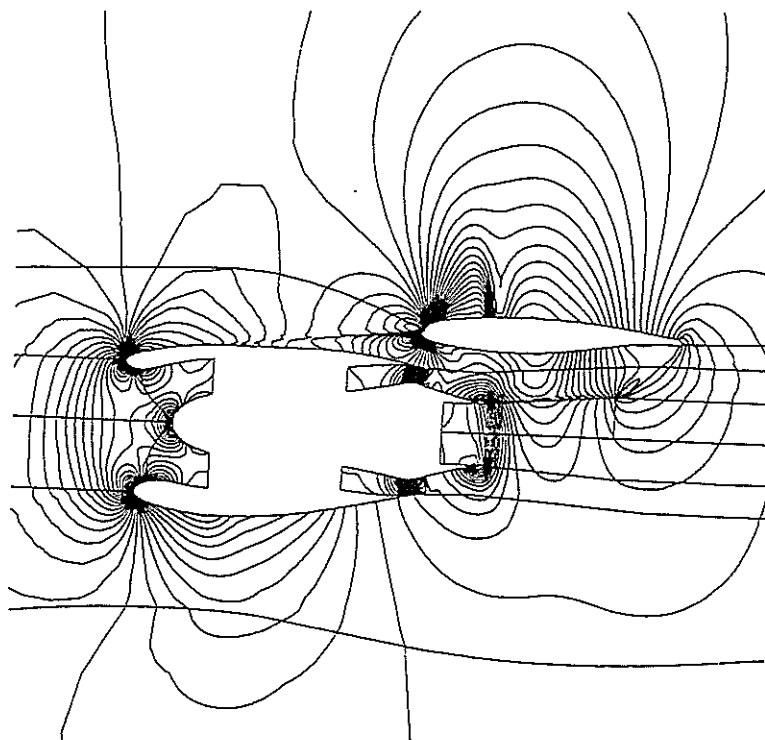


Figure 10: Lines of constant pressure coefficient in approximately vertical surface through VHBR-engine center line of DLR WBPE configuration. NLR ENSOLV flow computation on DLR multiblock grid.

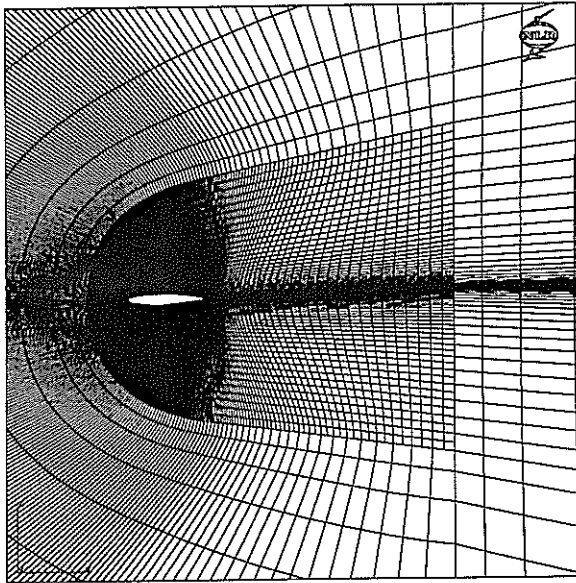


Figure 11: Euler far-field blocks around Navier-Stokes blocks around RAE2822 airfoil and in wake. N.B. local grid refinement, to reduce total number of grid cells

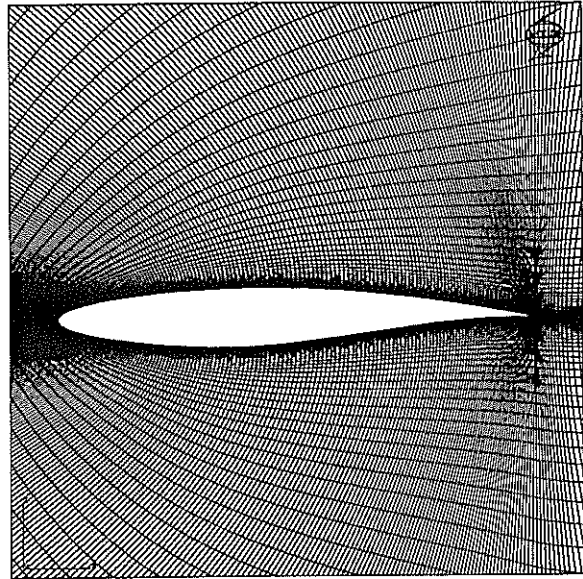


Figure 12: Navier-Stokes grid near RAE2822 airfoil

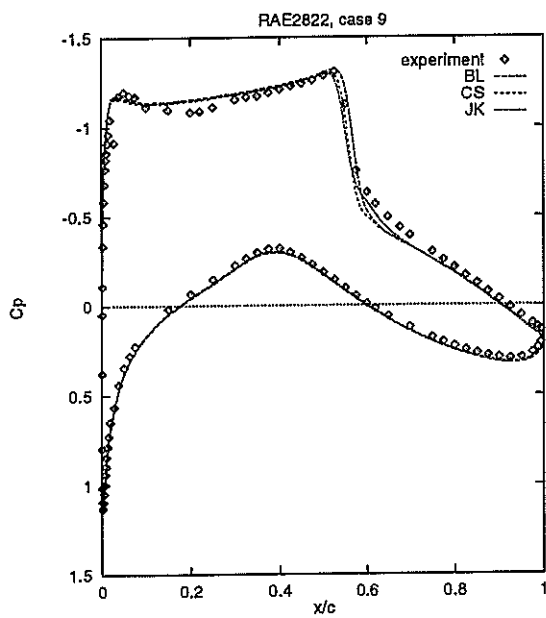


Figure 13: Pressure coefficient

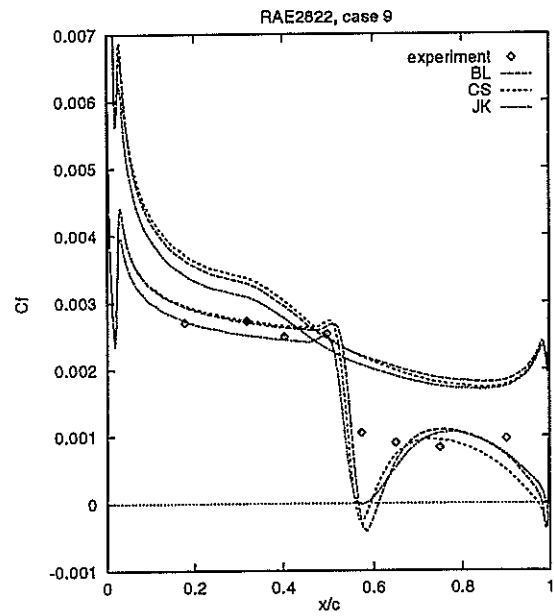


Figure 14: Skin-friction coefficient

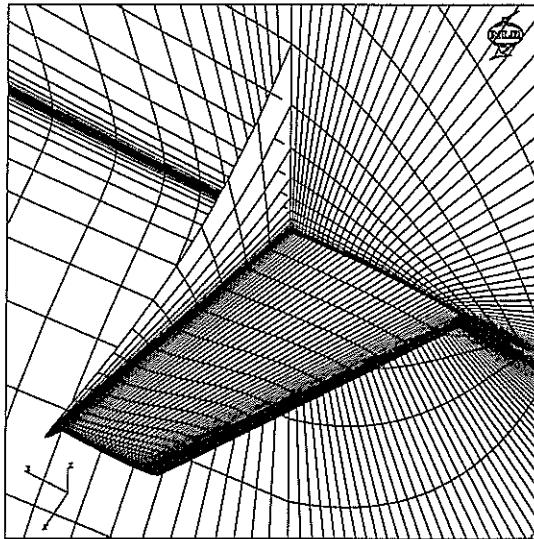


Figure 15: Navier-Stokes grid around ONERA M6 wing. 256 * 64 * 48 CO grid with about 790K grid cells.

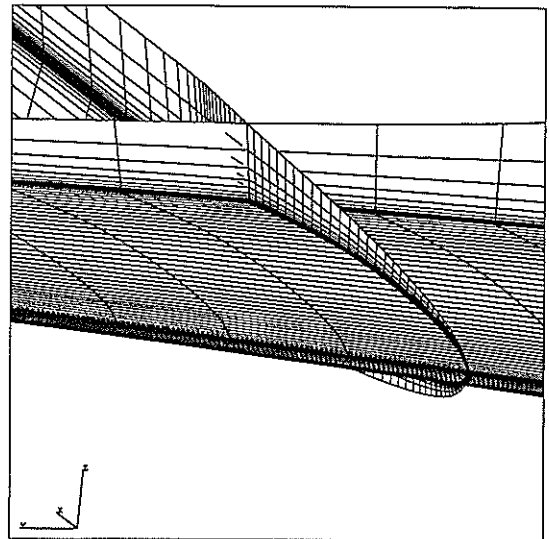


Figure 16: Nas MB grid around ONERA M6 wing: detail boundary-layer grid

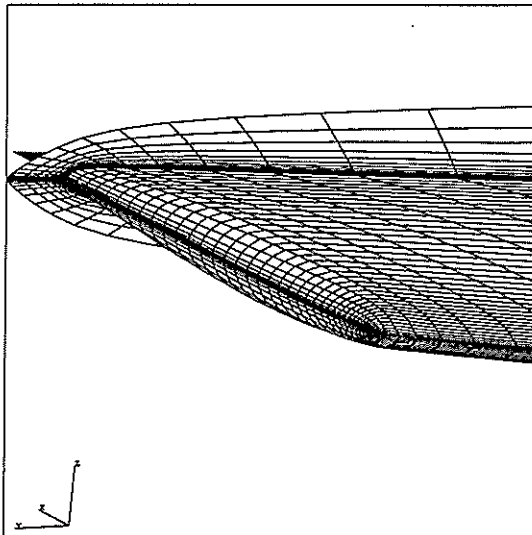


Figure 17: Nas MB grid around ONERA M6 wing: detail wing-tip grid

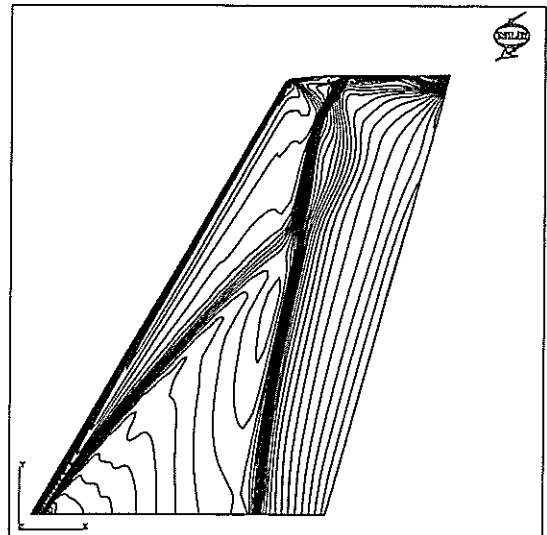


Figure 18: Pressure coefficient on ONERA M6 wing, with boundary layer. $M_\infty = 0.8447$, $Re_\infty = 11.78 \cdot 10^6$, $\alpha = 5.06$. Johnson-King turbulence model.