Prof. Wittenberg

NLR MP 86074 U

NATIONAAL LUCHT- EN RUIMTEVAARTLABORATORIUM

NATIONAL AEROSPACE LABORATORY NLR

THE NETHERLANDS

NLR MP 86074 U

TRENDS IN CFD FOR AERONAUTICAL 3–D STEADY APPLICATIONS: THE DUTCH SITUATION

BY

J.W. BOERSTOEL, A.E.P. VELDMAN, J. VAN DER VOOREN AND A.J. VAN DER WEES

NLR MP 86074 U

TRENDS IN CFD FOR AERONAUTICAL 3-D STEADY APPLICATIONS: THE DUTCH SITUATION

by

J.W. Boerstoel, A.E.P. Veldman, J. van der Vooren and A.J. van der Wees

Paper presented at the 25th meeting of the "Contactgroep Numerieke Stromingsleer" (Workinggroup Computational Fluid Dynamics) on October 20th, 1986, Delft, The Netherlands

(18 pages in total)

Division: Informatics
Prepared: JWB/ AEPV/ JvdV/ AJvdW/
Approved: FJH/

Completed : 28-XII-1986
Ordernumber: 0540.22
Typ. : MM

TRENDS IN CFD FOR AERONAUTICAL 3-D STEADY APPLICATIONS: THE DUTCH SITUATION

J.W. Boerstoel, A.E.P. Veldman, J. van der Vooren and A.J. van der Wees
National Aerospace Laboratory NLR, Informatics Division,
Anthony Fokkerweg 2, 1059 CM Amsterdam, The Netherlands

SUMMARY

Current and mid-term developments in computational 3-D steady aerodynamics software at NLR, focusing on the efficient aerodynamic design of the next generation of transport aircraft, are surveyed on a global level. Following a brief review of the various levels of sophistication in physical flow modelling, and their relation to the aerodynamic design process in general, the major aerodynamic problem areas that are at present accessible to computational aerodynamics are discussed. The coherence in computational methods development is subsequently explained by showing how the methods cover a growing part of the aircraft operating range. Subsequently, the approach taken towards the development of the most advanced methods, based on the Euler and Reynolds-averaged Navier-Stokes equations, is discussed. Here the accents are on proven technology, uniformity of approach, block-structured boundary conforming grids, flexibility, robustness, and adaptive local grid refinement for physical relevance. It is shown that the developments discussed presuppose access to the computing power offered by the present and upcoming generation of modern vectorcomputers. Finally, the informatics aspects are discussed. It is explained, that the steadily growing amount of computational aerodynamics software needs definite measures to keep things under control. The general technical concept, which is currently being developed at NLR to stay in control, is briefly surveyed. This involves the management of methods as well as data, and the interaction with the user. Computers/workstations are embedded in an efficient communication network.

INTRODUCTION

The purpose of the present paper is to explain the current and mid-term developments at NLR in steady aerodynamics computations. Since the Dutch aircraft industry focuses on transport aircraft, the contents of the paper is restricted to applications for this type of aircraft.

The development of computational aerodynamics software to day has four aspects that are of prime importance. The first aspect is the decision which aerodynamic problems are going to be solved. This involves the aerodynamic configuration, the flight conditions, and the purpose why the problem must be solved. These factors then lead to the choice of a level of physical flow modelling, dependent on the technology available.
The second aspects is the choice of a numerical approach, which eventually must lead to the construction of a numerical algorithm that solves the problem. Also here the choice must be carefully made, because computational aerodynamics methods must be robust, fast, and above all flexible (in the sense that their range of application can grow evolutionary). Wrong choices can easily frustrate the delicate balance that usually exists between these factors.

The result of the above discussions and choices is discussed in the chapter on "current and mid-term developments". Occasionally it is possible to acquire the necessary software from elsewhere (e.g. by software exchange), but in most cases in-house development is mandatory. Such developments require as a rule large investments over comparatively long periods of time (number of years). Therefore decision making must be thorough.
Subsequently, the approach taken towards computational aerodynamics methods based on the Euler and the Reynolds-averaged Navier-Stokes equations will be discussed in a special chapter. This subject has been chosen, because Euler and Reynolds-averaged Navier-Stokes based methods are most advanced, and their development towards applications of real engineering interest has in fact only just begun.

The third aspect is access to the computing power required. This aspect is discussed in some detail in the chapter on "computing power required".

The fourth aspect concerns the efficient development of the necessary software (i.e. fast, and at low cost), the implementation of the software in the computational aerodynamics infrastructure for efficient usage, and finally the efficiency of further evolutionary developments and maintenance. Under the title "informatics aspects", a final chapter discusses this problem area from a number of angles, and explains the general technical concept that is being developed at NLR to handle the management of methods as well as data, and the user interaction; computers/workstations are embedded in an efficient communication network.

CURRENT AND MID-TERM DEVELOPMENTS

Steady flow calculations can be based on various levels of sophistication of the physical flow model. Also, two basic types of computation are generally required, viz. direct and inverse computation. Before discussing the current and mid-term developments in more detail, these two aspects will be reviewed briefly. The choice for a certain level of physical flow modelling, or the type of computation, generally depends on the application (purpose, and physical relevance required) and the technology available (numerical techniques, computing power). In a design environment, computational speed very often prevails over completeness in the modelling of the physics.

In aircraft aerodynamics the most sophisticated level of physical flow modelling is the Navier-Stokes equations, where only the small subgrid scale turbulent eddies are covered by (isotropic) turbulence modelling (level V). These equations model on a continuum basis all relevant flow phenomena, including large turbulent eddies. These equations are essentially time dependent (because of turbulence), and at the present stage of technology still out of reach. One level lower (level IV) are the Reynolds-averaged Navier-Stokes equations (and subsets of these like the thin layer equations and the so-called parabolized Navier-Stokes equations). In these equations turbulence is modelled completely, partly on a theoretical and partly on an empirical basis, whence steady processes exist and the steady equations have meaning. Further simplifications of the flow model require, that the flow domain be decomposed in subdomains where the viscous effects are important (boundary layers, wakes), and subdomains where the viscous effects are negligible (inviscid outer flow). Naturally, the flow solutions in the viscous and inviscid subdomain must be coupled. Three levels of coupling are usually distinguished, viz. no coupling, weak coupling, and strong coupling. No coupling means, that the inviscid flow is computed without taking the boundary layers and the wakes into account; boundary layers/wakes are computed afterwards using inviscid pressure distributions. Weak coupling means, that the inviscid flow and the boundary layer/wakes are computed alternatively in an iterative fashion. Strong coupling means, that the inviscid flow and the boundary layers/wakes are computed simultaneously. Strong coupling is mandatory if the boundary layer separates (i.e. reverse flow occurs).

In inviscid subdomains, three further levels of simplification of the flow model are of interest. Level III is the so-called Euler equations. These equations constitute the most complete inviscid flow model. One level lower, level II, is potential theory. Here the flow is assumed isentropic and irrotational, in order to allow the introduction of a velocity potential. In this case, the five conservation laws of physics (mass, 3*momentum, energy), reduce to the law of mass conservation. The assumptions are correct for subsonic flow, and reasonably accurate for transonic flow as long as shockwaves are weak. The advantage is the reduction of the number of dependent variables from five to one. The lowest level of sophistication, level I, is the so-called Prandtl-Glauert equation, which is the fully-linearized small-disturbance version of potential theory. As the equation is linear, the representation of shock waves is no longer possible.

In viscous subdomains, various subsets of the Reynolds-averaged Navier-Stokes equations, boundary layer equations (laminar, turbulent), thin layer equations, parabolized equations, or even the full equations, are of interest.

In aerodynamic aircraft design, two basic types of computations are required. The first type is the direct computation, in which the aircraft geometry is given and the flow is required. From the flow data, the aerodynamic characteristics required are derived. At present, such computations are feasible on the levels I (Prandtl/Glauert), II (potential theory), and III (Euler) of inviscid flow modelling, and the level of boundary layer equations (often in integral form). However, level IV (Reynolds-averaged Navier-Stokes, and subsets) is coming rapidly within reach. The second type is the inverse computation, in which some characteristics of the flow are given and the aircraft geometry is required within certain constraints. Such inverse computations are of great value in wing design. Then a favourable pressure distribution for cruise conditions is prescribed by an experienced aerodynamic designer, and the geometry of the wing is sought under such constraints as given planform, and minimum allowed thickness. Such computations put a high demand on computational speed, and therefore are at present only feasible on the levels I (Prandtl/Glauert) and II (potential theory), and the level of boundary layer equations (often in integral form).

Current and mid-term developments aim at the efficient aerodynamic design of the next generation of transport aircraft.

Methods are required for subsonic and transonic cruise conditions, as well as for take-off and landing conditions. The revived interest in propeller propulsion (in view of its prospects for lower fuel consumption) then leads to the requirement that these methods must be applicable to not only jet-aircraft, but also to propeller-aircraft. Since the aerodynamic integration of the propulsion system is an important aspect in the aerodynamic design considerations, it is necessary to have methods that can handle complex aircraft configurations, including wing, body, tail, nacelles, pylons, winglets, propeller slipstreams, jet exhaust plumes, etc. (Fig. 1). In case of take-off and landing, the complexity is even greater, because control surfaces must be simulated as well, and vorticity shedding plays an important role (Fig. 2).

An important goal in the aerodynamic design of transport aircraft today is to improve upon their aerodynamic effectivity. This involves both take-off and landing, as well as cruise conditions. The opportunities to improve aircraft performance under cruise conditions are constrained by necessary characteristics at low speed during take-off and landing. If better devices can be developed to increase the lift during take-off and landing, it is possible to reduce the drag under cruise conditions.

For subsonic cruise conditions, as well as take-off and landing conditions, a higher-order accurate panel method is currently being developed on the basis of the Prandtl/Glauert equation (level I). Mid-term developments aim at extending the higher-order panel method to a field panel method on the basis of potential theory (level II), and at the incorporation of turbulent boundary layer and wake effects (mainly on the wing) in such a way that separation is allowed (strong interaction).

For transonic cruise conditions, finite-volume methodology based on potential theory (level II) is currently being developed for direct [1], as well as inverse, computations. In the latter case, emphasis is on wing design. Also in this case, mid-term developments aim at the incorporation of turbulent boundary layer and wake effects (mainly on the wing) in such a way that separation is allowed (strong interaction).

For installation effects of propellers and jet engines, finite-volume methodology based on the Euler equations (level III) is currently being developed [2]. Here the first goal is the interaction of a wing/nacelle with a propeller-slipstream (Fig. 3). Mid-term developments will involve jet engine inlet and exhaust flows (Fig. 4), requiring extension to the Reynolds-averaged Navier-Stokes equations (or subsets, level IV).

Regarding the improvement of aerodynamic effectivity, the first step is the development of devices to increase the lift during take-off and landing by studying two-dimensional airfoil/slats/flaps configurations (Fig. 5). Currently, a higher order field panel method (based on potential theory, level II, [3]) is being extended with a turbulent boundary layer in the strong-interaction sense. Mid-term developments will involve methodology on the basis of the Reynolds-averaged Navier-Stokes equations (or subsets, level IV).

The above discussed current and mid-term development of methods is the consequence of a pertinent policy to cover the aircraft operating range to the best possible extent in view of the technological possibilities.

Consider figure 6, where a qualitative picture is given of the range of applicability of methods, based on the various levels (I through IV) of physical flow modelling, in the different parts of the operating range of a subsonic transport aircraft. Most aircraft flying today have been designed using mainly Prandtl/Glauert methods (level I), and of course the windtunnel. At best, also some early potential methods (level II) were used incidentally. For the design of the next generation of transport aircraft it is necessary to have sufficiently powerful methods on level I and levels II, III. These are basically the methods that must handle complex aircraft configurations in take-off, cruise and landing, and must facilitate the aerodynamic integration of the propulsion system. However, since these flight conditions cannot be investigated in sufficient detail without taking into account boundary layer separation effects to some extent (e.g. on the wing), it is also necessary to extend these methods onto the strong interaction levels IV/I and IV/II,III for specific applications. Also, e.g. jet engine exhaust flows can only be handled adequately by the Reynolds-averaged Navier-Stokes equations (or subsets thereof), level IV. Similarly, the development of devices to increase the lift during take-off and landing requires the Reynolds-averaged Navier- Stokes equations to model the physics adequately (level IV).

APPROACH TO EULER AND REYNOLDS-AVERAGED NAVIER-STOKES METHODOLOGY

The timely development of the comparatively new and complicated methodology associated with the Euler and Reynolds-averaged Navier-Stokes equations, at a reasonable cost, requires careful consideration of all aspects involved. In this chapter, some ideas that exist today at NLR, and also some decisions that have already been taken, will be briefly discussed.

The approach taken is first of all to use, whenever possible, proven technology, in order to cut down the development time and cost. New basic research will therefore be carried out only if the aerodynamic goals set cannot be reached on the basis of the proven technology available.

A second important issue is the awareness of the fact, that the Euler equations, and e.g. the thin-layer Navier-Stokes equations, or the parabolized Navier-Stokes equations, are all subsets of the full Reynolds-averaged Navier-Stokes equations. This leads to the general strategy, that a flow solver for the full Reynolds-averaged Navier-Stokes equations must function properly for all subsets. Naturally, this strategy sets a requirement for the development of an Euler flow solver which generally precedes the development of flow solvers for the viscous subsets of the Reynolds-averaged Navier-Stokes equations. It should be realized that a high demand for computational efficiency can easily lead to developments which depart from the above strategy. Though such developments cannot always be avoided, there occurrence should be minimized.

Thirdly, all software development is preferably directed to three-dimensional flow, right from the beginning. This point of view is taken, because the successfull generalization of a two-dimensional approach requires three-dimensional considerations anyway, and experience has taught that two-dimensional pilot versions of the software as a rule do not provide sufficient insight in the informatics aspects of three-dimensional flow simulation.

Finally, an integrated uniform approach is taken towards the problem areas of gridgeneration, flow solving, and visualization. In particular, the areas gridgeneration and flow solving will be discussed below in more detail.

The first choice in gridgeneration is always between fully boundary conforming, not boundary confirming at all, or a mixed form of these two extremals. Mainly based on two arguments, here the choice is made for fully boundary conforming grids. The first, and most important, argument has to do with the fact that the most difficult part of doing Euler and Reynolds-averaged Navier-Stokes calculations is not the development of a stable method, but rather the achievement of a physically relevant solution. Then, of course, high accuracy of boundary-condition implementation is a prerequisite, and it is firmly believed that such high accuracy can only be obtained using fully boundary conforming grids. The second, and subsidiary, argument is that, only in a fully boundary conforming grid, control can be exercised over the coordinate directions in the vicinity of the boundary. Such control can be important in using algebraic turbulence models such as mixing length models and eddy viscosity models. But also the "thin-layer" and "parabolized" versions of the Reynolds-averaged Navier-Stokes equations require such control explicitly.

For generating the fully boundary conforming grids, the following set-up has been decided upon [2]. First, the physical space surrounding the aircraft (or some of its components) is made finite by placing boundary surfaces, assuming that outside these surfaces the flow is known. Subsequently, this finite physical domain is subdivided into blocks in such a way that each block is topologically equivalent to a cube in computational space. Each block (cube) has six faces, twelve edges, and eight vertices. Though block-packing is restricted to "face-to-face", they can still be assembled to computational domains of arbitrary topological complexity (Fig. 7). Once the block-subdivision is established, and the relationships between all faces, edges, and vertices are determined, the grid is set up by the subsequent generation of gridpoint distributions in each edge, face and block, using (transfinite) linear interpolation. This leads to a grid with hexahedral cells and grid lines that are continuous across the faces, edges and vertices of adjacent blocks (Fig. 8). The final step is to smooth the grid in each block using an elliptic method that acts on user-provided information affecting the cell-size distribution. The set-up chosen is sufficiently flexible to allow generalizations such as slope-continuous gridlines or discontinuous changes in gridpoint distributions across block faces.

The advantages of the above described approach to gridgeneration become clear if flow solver development is considered in conjunction with the vector/parallel processing capabilities of modern supercomputers. By construction, the data corresponding to each block are well-ordered, and this is favourable from the point of view of efficiently approximating and solving the flow equations. This also contributes to the vectorizability of the flow solver algorithm. But also the fact that there can be drawn on an extensive literature on finite-difference/volume technology should not be forgotten. Finally, the accuracy of the approximation of the flow equations benefits from the smoothness of the grid in each block. Further advantages of the block-structuring are its amenability to parallel processing, and the inherent possibility of using a different subset of the Reynolds-averaged Navier-Stokes equations in each block.

The above argument shows that a block-structured grid of hexahedral cells is a flexible approach towards the development of Euler and Reynolds-averaged Navier-Stokes based methodology for complex three-dimensional aerodynamic shapes. As such it is considered to be an alternative to the often advocated unstructured grids using tetrahedral cells. Note, however, that a hexahedral cell can be subdivided into either five or six tetrahedral cells, whence the gridgeneration approach described above can be used also to generate an unstructured grid with tetrahedral cells.

With respect to the development of flow solvers for the Euler and Reynolds-averaged Navier-Stokes equations, the afore mentioned general strategy, viz. that a flow solver for the full Reynolds-averaged Navier-Stokes equations must function properly for all subsets, leads to the following considerations.

At present there are strong indications that the solution of the steady Reynolds-averaged Navier-Stokes equations may be hampered by non-existence as well as by non-uniqueness problems. Non-existence of steady solutions for two-dimensional laminar flow involving separation has been demonstrated in an asymptotic framework [4], and in interacting boundary-layer theory [5]. In [4], also non-uniqueness was found. For three-dimensional flow, non-existence of a steady solution of the Navier-Stokes equations was observed in [6]. The above reasons support the viewpoint that the notion of a steady solution should be replaced by the notion of a limit solution of the unsteady Reynolds-averaged Navier-Stokes equations as time goes on. Hence, regarding the building of flow solvers, there is a strong preference to base these on (pseudo) time integration of the unsteady equations. A bycoming advantage is then, that such solvers can be generalized to time-accurate unsteady flow (e.g. buffet) with comparative ease.

Another consideration of a general nature is a plea for adaptive local grid refinement. As observed before, the most difficult part of doing Euler and Reynolds-averaged Navier-Stokes calculations is the achievement of a physically relevant solution. Locally this will require an extremely fine grid to obtain the necessary accuracy. A good example in viscous flow is the resolution of shear layers of which the position is unknown beforehand. An example in inviscid flow (Euler equations) is the avoidance of spurious entropy production. However, it is mandatory that the total number of grid points be kept as low as possible from the point of view of acceptable computational time and cost. Two grid refinement strategies are possible. The first one is repositioning of a fixed number of grid-points; this strategy has the obvious disadvantage that a local increase in resolution is inevitably accompanied by a local decrease in resolution elsewhere, and can therefore easily lead to areas of too low resolution. The second one is the local insertion/deletion of gridpoints in an otherwise fixed grid. In light of the above discussion, the second strategy is definitely favoured.

More specific considerations with respect to flow solver development concern discretization and solution strategy.

Consider the subset Euler equations. These inviscid equations allow the occurrence of true discontinuities, viz. shockwaves and contact discontinuities. It is well known that the proper capture of such discontinuities by any numerical method requires discretization schemes which are fully conservative approximations of the Euler equations in full conservation form. With respect to the Reynolds-averaged Navier-Stokes equations, this requirement carries over to the convective parts of the equations. But even for flows without shockwaves or shear layers (contact discontinuities in inviscid flow), there is strong evidence that maintaining conservation in discretized form enhances accuracy considerably (such evidence comes from potential flow solutions of internal as well as external flow, [7]). In maintaining conservation in discretized form, grid discontinuities across block-faces, and similar discontinuities caused by adaptive local grid refinement, require special attention (Fig. 9).

A few considerations with respect to the solution strategy are the following. Time-explicit integration schemes are computationally simple and well amenable to vectorization. Time-implicit integration schemes are computationally definitely more complex and less amenable to vectorization. Also, with time-explicit integration schemes the allowable time-step is seriously limited by stability considerations (accuracy considerations are of interest only if a time-accurate solution is required). In general, the performance of a time integration scheme depends on the combined effect of its stability and vectorizability properties. In (almost) inviscid flow the allowable time-step of an explicit integration scheme is proportional to the local spatial mesh size; as time-accuracy is not required, the local allowable time-step can be used to accelerate convergence. Mainly based on their computational simplicity, and on their amenability to vectorization, time-explicit integration schemes are preferred over time-implicit integration schemes in this case. However, in viscous-dominated flows the situation changes, because in the viscous regions the local spatial mesh sizes have to be much smaller then in the inviscid region. This leads to a drastic reduction of the allowable time-step for explicit schemes. It is not believed that the good vectorization properties of time-explicit schemes are sufficient compensation for the indeed very small allowable time-steps. Hence, in this case there is a definite preference for time-implicit integration schemes per block. This way the scheme is still amenable to parallel processing. However, it should be realized, that the solution algorithm of a time-implicit integration scheme per block is seriously affected in case gridpoints are added locally for adaptive grid refinement.

Finally, the subject of convergence acceleration will be discussed in terms of cost effectiveness. Here the viewpoint will be taken, that a convergence acceleration technique for a given method is cost effective, provided that a solution of given accuracy is reached in significantly less computing time without having negative effects on the robustness and flexibility of the original method, and provided that the extra development time and cost are justified in the light of its usage. For the very complicated Euler and Reynolds-averaged Navier-Stokes flow solvers, that are required to treat the complex aerodynamic shapes associated with the design of future transport aircraft, this viewpoint is believed to limit the choice of convergence acceleration techniques to those computationally simpler than the original method, and equally vectorizable. Multigrid technique, which in practice has turned out to be complex, and to require high development cost, is certainly not among them. This does not preclude, however, the usage of multigrid technique for special well-defined applications.

Apart from the need for the computing power of a modern vectorcomputer (as illustrated in the next chapter) there is one common pacing item in the development of Euler and Reynolds-averaged Navier-Stokes methodology. This is building up the technology for adaptive local grid refinement, involving research in establishing the proper criteria for refinement, in devising a flexible discretization strategy, and in coping with the consequences for the solution process. Though not discussed in this paper, turbulence modelling is yet another important pacing item in the development of Reynolds-averaged Navier-Stokes methodology.

COMPUTING POWER REQUIRED

Integration of CFD methods as discussed above in the aerodynamic design process presupposes extensive testing and evaluation. Experience has taught that this process can only be carried out efficiently if full-scale calculations can be performed within, say, one half hour turn-around time. This requirement is based on the fact that the timely development (in a number of years) of such complicated methods demands that, on each working day, a number of full-scale computations can be executed and analyzed.

The above requirement is quantified in table 1 for two current developments and one mid-term development. The current developments are (1) the Euler finite-volume method to calculate propeller-slipstream/nacelle/wing interaction (propulsion system installation effects, see Fig. 3) and (2) the inverse potential finite-volume method to design the wing of a transport aircraft under transonic cruise conditions. The mid-term development is (3) the two-dimensional Reynolds-averaged Navier-Stokes method for airfoil/slats/flaps configurations (Fig. 5). Considering that future three-dimensional Reynolds-averaged Navier-Stokes calculations require a yet significantly larger computing power, this table shows that computing speed must be in the order of at least 300 Mflop/s, and the central memory must be larger than 40 M numbers.

Computing power such as indicated above is today only available in modern vectorcomputers such as e.g. CRAY-2, CRAY X-MP, NEC SX-2, and the upcoming ETA-10. Software development therefore presupposes short-term access to a supercomputer of this class.

INFORMATICS ASPECTS

The computational aerodynamics process involves repeated application of the following functions:

- Geometry definition (and manipulation), in order to obtain a mathematical representation of the geometry of the aerodynamic shapes. This function can be attributed to commercially available CAD/CAM packages.

- Grid generation (and manipulation), in order to cover the flow domain with a computational grid, accessible to a particular flow solver. As was already explained, flow solver developments are based on block-structured grids. The block subdivision of the flow domain, and the grid inside each block, are typical for the mathematical flow model used. This is dictated by the flow phenomena that the method must describe, and by the resolution required. Block subdivision is a process that can be carried out manually only in very simple cases. However, complex aerodynamic shapes can easily require in the order of one hundred blocks, and an automated process is mandatory. Commercially available CAD/CAM solid modelling packages might be of help. Since general purpose software for the subsequent generation of grids in the block structure, and for establishing their mutual relationships, are not (yet) commercially available, this is an in-house development.

- Flow calculation, in order to obtain a flow solution for a given aerodynamic shape, grid, mathematical model, and flow condition. Flow solvers are predominantly the subject of in-house developments. Commercial general purpose software is -generally speaking- not available for the many highly nonlinear problems of computational aerodynamics.

- Postprocessing for presentation and analysis of the computed flow. This involves both inspection (quick-look) postprocessing and analysis (detailed, complex-look) postprocessing. This function can only partly be accomplished by commercially available CAD/CAM and graphics packages, and therefore in-house developments are being carried out in parallel.

The above functions require different types of hardware equipment. Geometry definition (and manipulation), as well as postprocessing, are heavily interactive graphics applications, requiring at least workstation power. Grid generation requires both graphics and mainframe computing power, while flow calculation is a typical number crushing activity requiring a supercomputer.

In computational aerodynamics, a steadily graving amount of software is becoming available to perform the functions described above. In computational aerodynamic design processes, many different methods are being used, ranging from the simplest linear potential methods to highly complicated Reynolds-averaged Navier-Stokes methods; the choice of a flow solver depends on the balance between information required, its capabilities, and its computational cost. Growth can be observed both in latitude (more different methods and applications; more large scale applications, i.e. more gridpoints) as well as in depth (more complex physical modelling; increasing range of physical phenomena). As a result, a hierarchy of methods has developed, whereby each method has its own range of applicability (inviscid/viscous flow; rotational/irrotational flow; wing-alone/complex aircraft configuration) and its own computer resource requirements (little/much computing time; central processor/central memory usage). None of these methods can be used at a reasonable cost to cover all applications. Domain splitting concepts will lead to even more complicated methods, integrating e.g. full potential, Euler, boundary layer, and Reynolds-averaged Navier-Stokes methods. The amount of data absorbed and produced by present day computational aerodynamics methods, which is already tremendous, will grow even further with the new generation of vectorcomputers such as CRAY-2, CRAY X-MP, NEC SX-2, and the upcoming ETA-10.

Ways have to be found to stay in control of the above indicated developments. This involves the management of the methods, as well as of the modules of which they are composed. It also involves the management of the associated data. Finally, the interaction with the user and the embedding of computers/workstations in an efficient communication network are important aspects.

The general technical concept which is being developed at NLR to stay in control of e.g. the computational aerodynamics process, i.e. the hierarchy of the methods and modules of which the software is composed, the associated data, and the interaction with the user, is shown in figure 10. The concept requires that both methods, and modules, can be coupled on the functional level, while data-management is required to control the communication between the various methods. Method/module- and data-management can be automated when strict agreements are made on how the methods/modules in the hierarchy are to be interfaced, as well as strict rules are defined as to their individual usage.

For the management of methods, and of the modules of which they are composed, the system MEBAS (MEthod BAse System is being developed (in this framework a module is also called a method), see also [8]. MEBAS can operate (store, retrieve, couple) on a method-base of well-described methods, and is composed of two subsystems, viz. the method manager, and the executive. The method manager can be used for activities such as assemblage, repair, replacement, and versions management of methods, and of the modules of which these are composed. The executive takes care of the job execution task and operates on a library of executable methods (programs) built for specific applications.

Data-management is realized through the use of the system EDIPAS (Engineering Data Interactive Presentation and Analysis System), see [9]. Usage of EDIPAS requires that a common database be defined. The structure of this common database depends on the application to which an end-user applies the computational aerodynamics process. Each application can use its own database structure. As such, the common EDIPAS database forms the transfer point of data and the associated information between the various functions (geometry definition, grid generation, flow calculation, postprocessing) of the computational aerodynamics process (Fig. 11). All methods must therefore have a formal, application-dependent, interface with EDIPAS. It follows, that all methods can in principle communicate with each other via the common EDIPAS database, requiring only one interface for each method. An important advantage of the concept is also, that it requires of the software developer the careful a priori definition of the common database structure, of the EDIPAS interfaces, and of the control over each individual method.

EDIPAS can also partly perform the postprocessing function for presentation and analysis.

User interaction is realized using COLAS (COmmand Language System), see again [8].

The use of the above general technical concept (MEBAS, EDIPAS, COLAS) requires the proper definition of interfaces between methods, and as such avoids patchwork when integrating methods to applications. It also stimulates the reusability of software, and thus reduces software development costs.

CONCLUDING REMARKS

Current and mid-term developments in computational 3-D steady aerodynamics software at NLR have been shown to be directed towards the needs of the Dutch aircraft industry, and to cover a widening part of the transport aircraft operating range, crossing the separation onset boundary and occasionally protruding into the Reynolds-averaged Navier-Stokes range (Fig. 6). All levels of physical flow modelling, ranging from the linear Prantl-Glauert equation to the highly nonlinear Reynolds-averaged Navier-Stokes equations, are involved and have their specific area of application. There is in general a clear tendency towards complex geometries, which places a heavy accent on the integrated uniform approach to gridgeneration, flow solver development, and visualization. In this respect, a block-decomposition strategy has been accepted to generate boundary conforming structured grids with regular connectivity in each block. In the area of the well-established methods, based on (linearized) potential theory (with or without boundary layers), the emphasis is on complete aircraft in take-off, cruise, and landing configuration. In the area of the most advanced Euler and Reynolds-averaged Navier-Stokes based methods, the emphasis is first of all on installation effects of the propulsion system and high-lift devices to be used in take-off and landing. The development of Euler methods is well underway, while the development of Reynolds-averaged Navier-Stokes methods is about to start.

The approach towards Euler and Reynolds-averaged Navier-Stokes methods has been discussed. Time-dependent equations, adaptive local grid refinement, and fully-conservative finite volume schemes are favoured. Opinions are expressed with respect to the (pseudo) time integration of the scheme (explicit, implicit, multigrid). Pacing items are identified to be the various aspects of adaptive local grid refinement, and turbulence modelling.

The need for a modern vectorcomputer of the class CRAY-2, CRAY X-MP, NEC SX-2, ETA-10 is stressed.

A general technical informatics concept, which is being developed at NLR to remain in control of the rapidly expanding software and associated data in computational aerodynamics, is presented. This involves method-management, data-management, and user interaction. Computers/workstations are embedded in an efficient communication network.

REFERENCES

[1] Van der Vooren, J., Van der Wees, A.J., Meelker, J.H.: MATRICS, transonic potential flow calculations about transport aircraft, NLR MP 86019 U (1986).

[2] Boerstoel, J.W.: Problem and solution formulations for the generation of 3-D block-structured grids, NLR MP 86020 U (1986).

[3] Oskam, B.: Transonic panel method for the full potential equation applied to multi-component airfoils, AIAA paper 83-1855.

[4] Stewartson, K., Smith, F.T., Kaups, K.: Marginal separation, Stud. Appl. Math. 67 (1982), pp. 45-61.

[5] Henkes, R.A.W.M., Veldman, A.E.P.: Breakdown of steady and unsteady interacting boundary layers, NLR MP 86047 U (1986).

[6] Kordulla, W., Vollmers, H., Dallmann, U.: Simulation of three-dimensional transonic flow with separation past a hemisphere-cylinder configuration. Paper presented at the AGARD Symposium on Applications of Computational Fluid Dynamics in Aeronautics, 7-10 April 1986, Aix-en-Provence, France.

[7] Van der Vooren, J.: Private communication (1986).

[8] Loeve, W., Heerema, F.J., Van Hulzen, J.J.P.: The CAD environment of the National Aerospace Laboratory NLR, The Netherlands, NLR MP 86005 U (1986).

[9] Heerema, F.J., Van Hedel, H.: An engineering data management system for computer aided design, Adv. Eng. Software, Vol. 5, No. 2, pp. 67-75 (1983).

Table 1 Computing power required for current and mid-term method developmemt.

method	total flop	flop/s	CM	total DATA
1	135 - 360 G	80 - 200 M	10 - 40 M	3 - 25 M
2	70 - 280 G	40 - 160 M	5 - 30 M	1,5 - 6 M
3	200 - 500 G	110 - 280 M	1 M	1 M

1: propeller-slipstream/nacelle/wing interaction; Euler equations

2: transonic flow about a complex aircraft configuration under cruise conditions: inverse wing design; potential theory

3 airfoil/slats/flaps; two-dimensional Reynolds-averaged Navier-Stokes equations

G = giga = 10^9, M = mega = 10^6

total flop: total number of floating point operations required

flop/s : number of flop per second required for one half hour turn-around time

CM : central memory size required, expressed in 32 or 64 bit words

total DATA: size of dataset required for input preparation and output inspection, expressed in 32 or 64 bit words

CONFIGURATION POSSIBILITIES

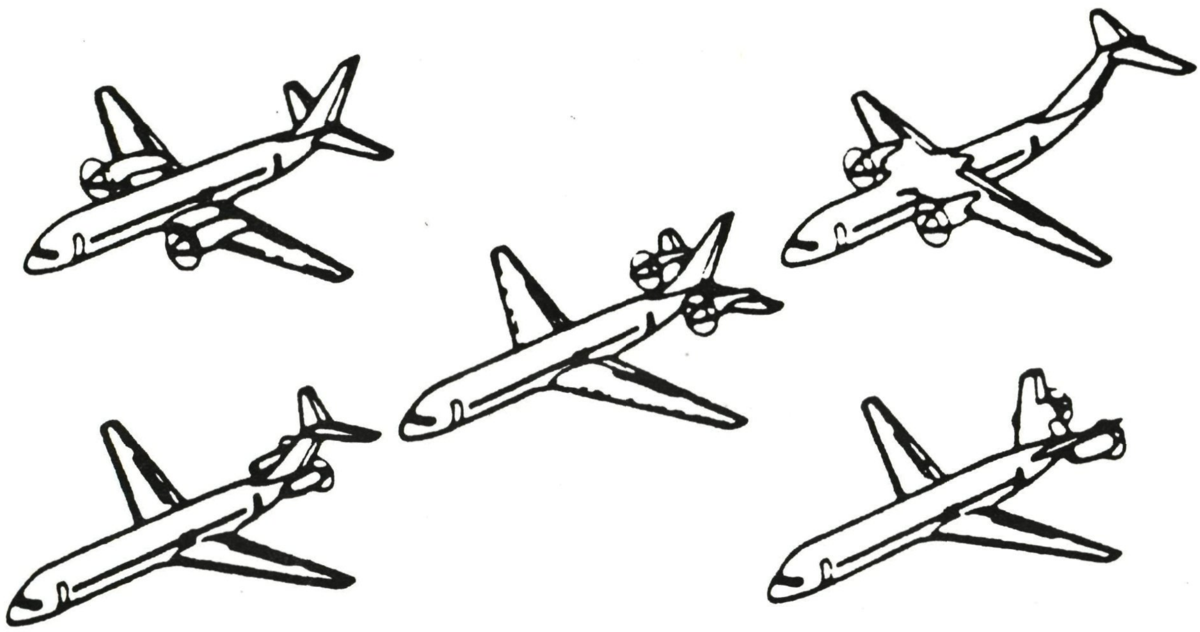


Fig. 1 Examples of complex transport aircraft configurations

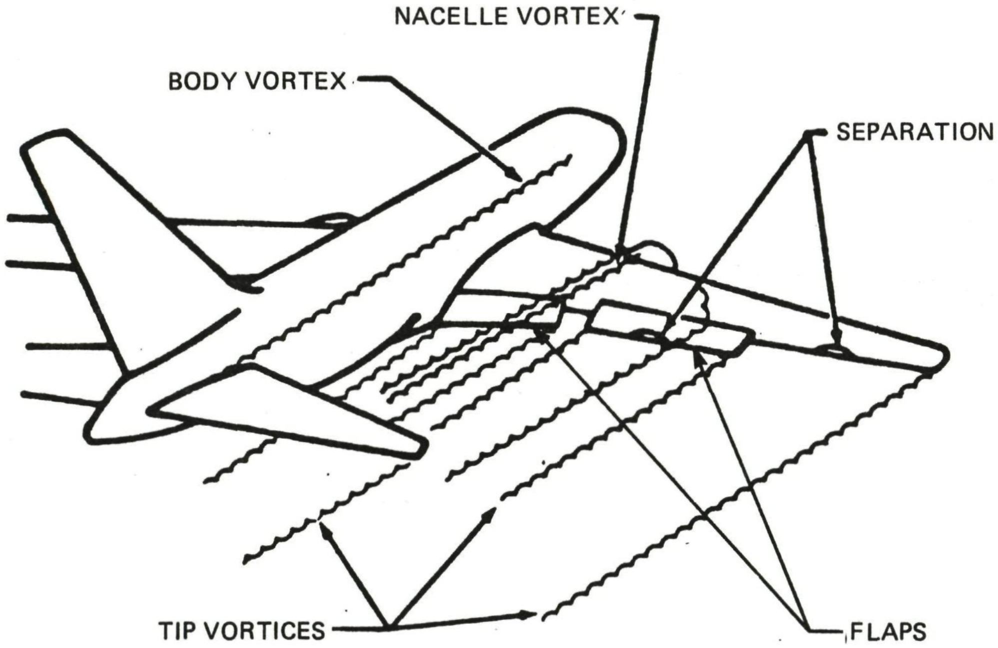


Fig. 2 Transport aircraft in take-off, or landing

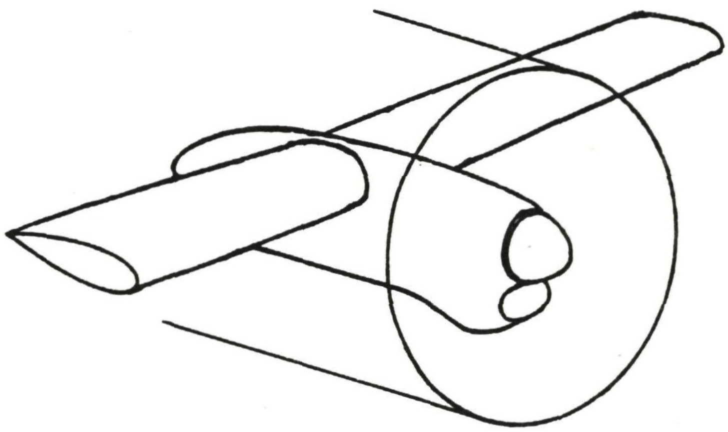


Fig. 3 Wing/nacelle/propeller-slipstream interaction

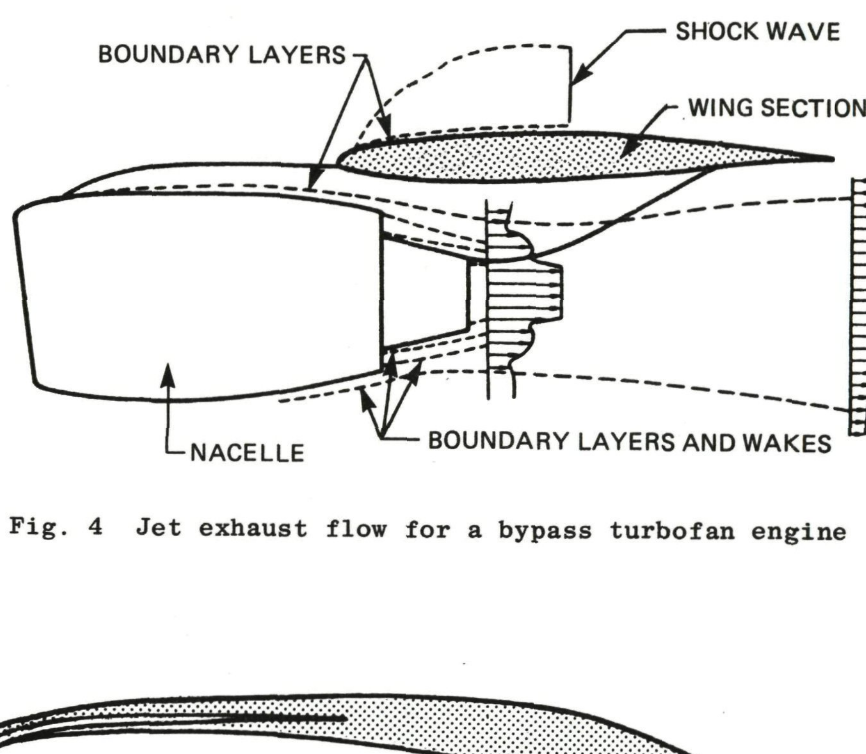


Fig. 4 Jet exhaust flow for a bypass turbofan engine

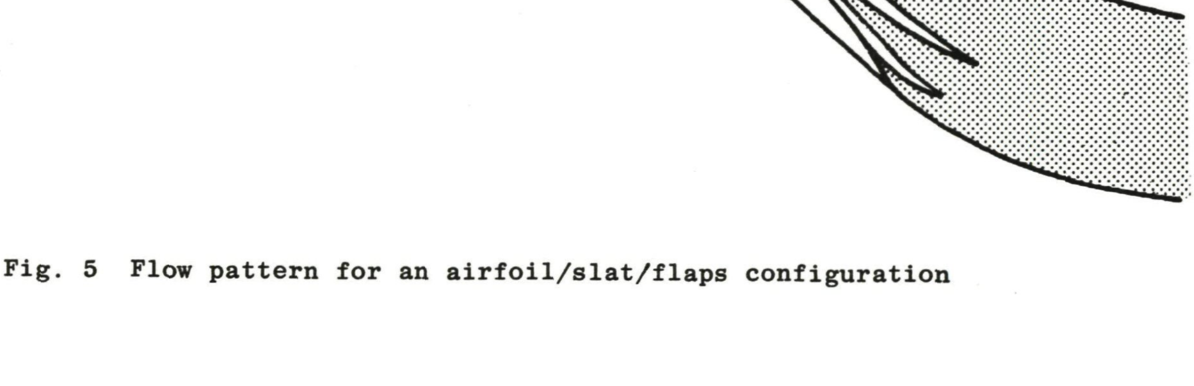
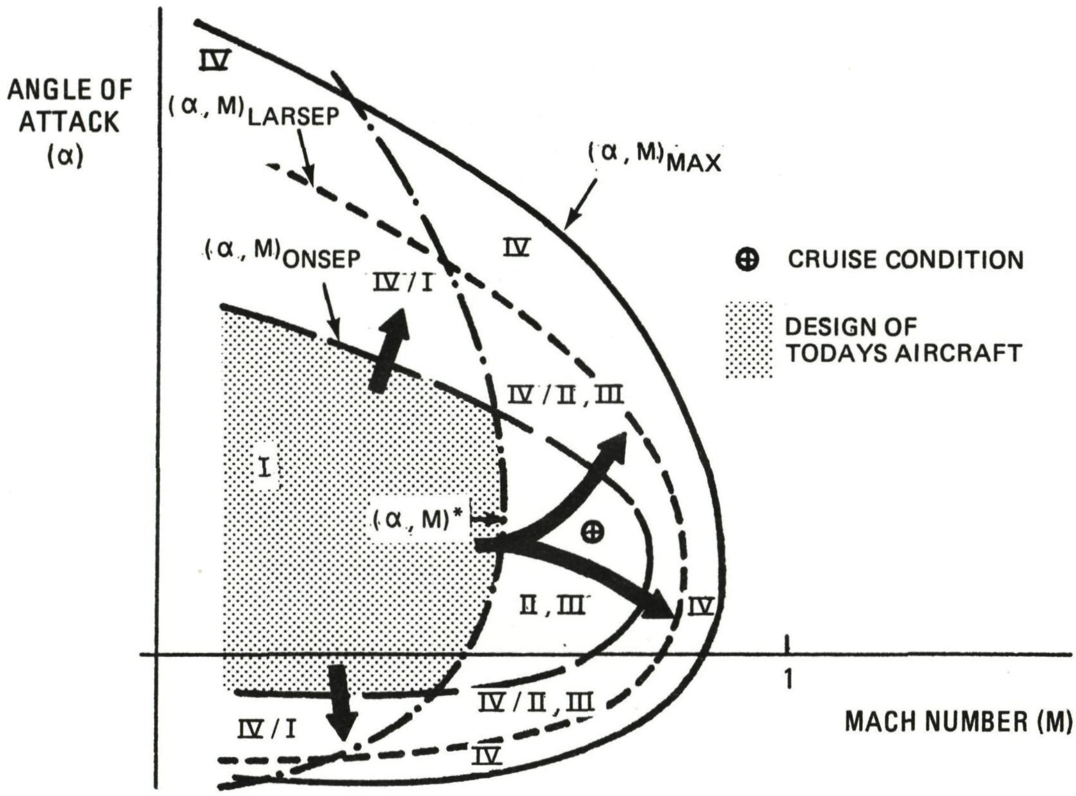


Fig. 5 Flow pattern for an airfoil/slat/flaps configuration



LEGEND

- $(\alpha, M)^*$ — ONSET OF TRANSONIC FLOW
- $(\alpha, M)_{ONSEP}$ — ONSET OF SEPARATION
- $(\alpha, M)_{LARSEP}$ — ONSET OF LARGE SEPARATION
- $(\alpha, M)_{MAX}$ — BOUNDARY OF AIRCRAFT OPERATING RANGE
- I — RANGE COVERED BY PRANDTL/GLAUERT METHODS WITH/WITHOUT WEAK BOUNDARY LAYER INTERACTION
- II, III — EXTRA RANGE COVERED BY POTENTIAL METHODS AND EULER METHODS WITH/WITHOUT WEAK BOUNDARY LAYER INTERACTION
- IV/I — LIKE I, BUT WITH STRONG BOUNDARY LAYER INTERACTION
- IV/II, III — LIKE II, III, BUT WITH STRONG BOUNDARY LAYER INTERACTION
- IV — EXTRA RANGE COVERED BY (SUBSETS OF) RE–AVERAGED N.S. EQUATIONS

Fig. 6 Parts of the operating range of a subsonic transport aircraft covered by methods based on the various levels of physical flow modelling

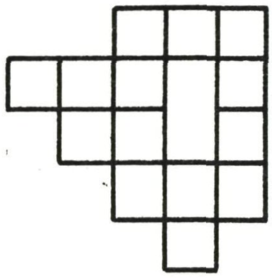


Fig. 7 Example computational domain of arbitrary topological conplexity (two-dimensional)

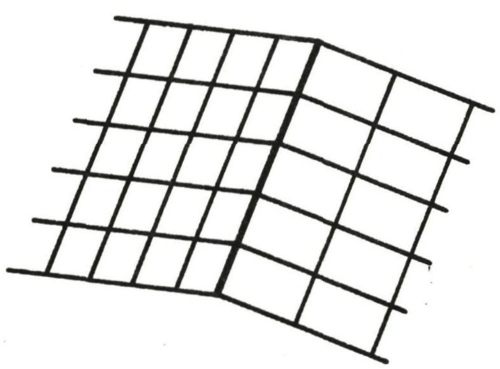


Fig. 8 Example of a continuous grid across block-faces (two-dimensional)

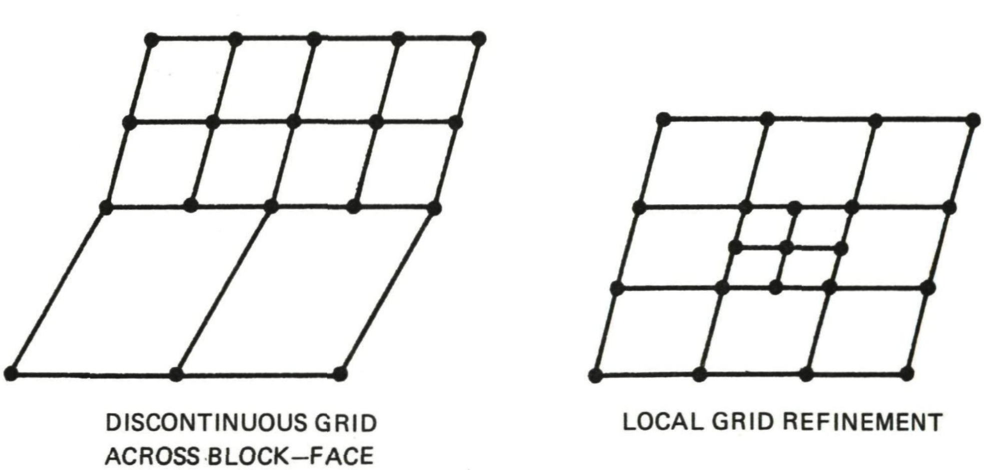


Fig. 9 Grid discontinuities (two-dimensional)

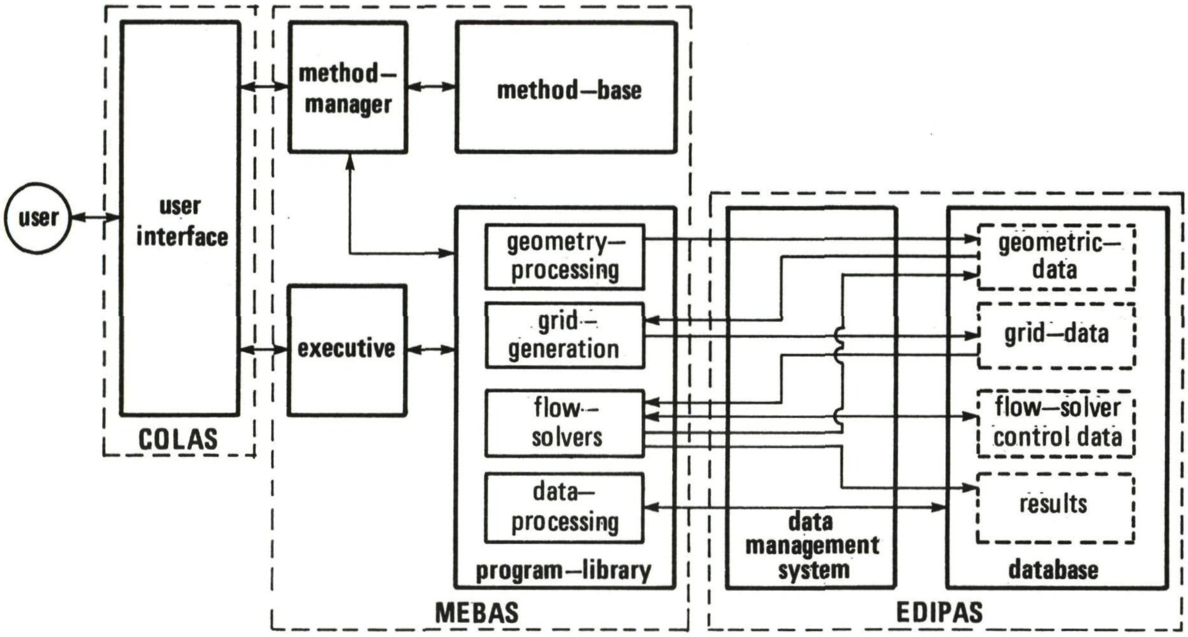


Fig. 10 General technical concept at NLR for data management, method management, and user interaction

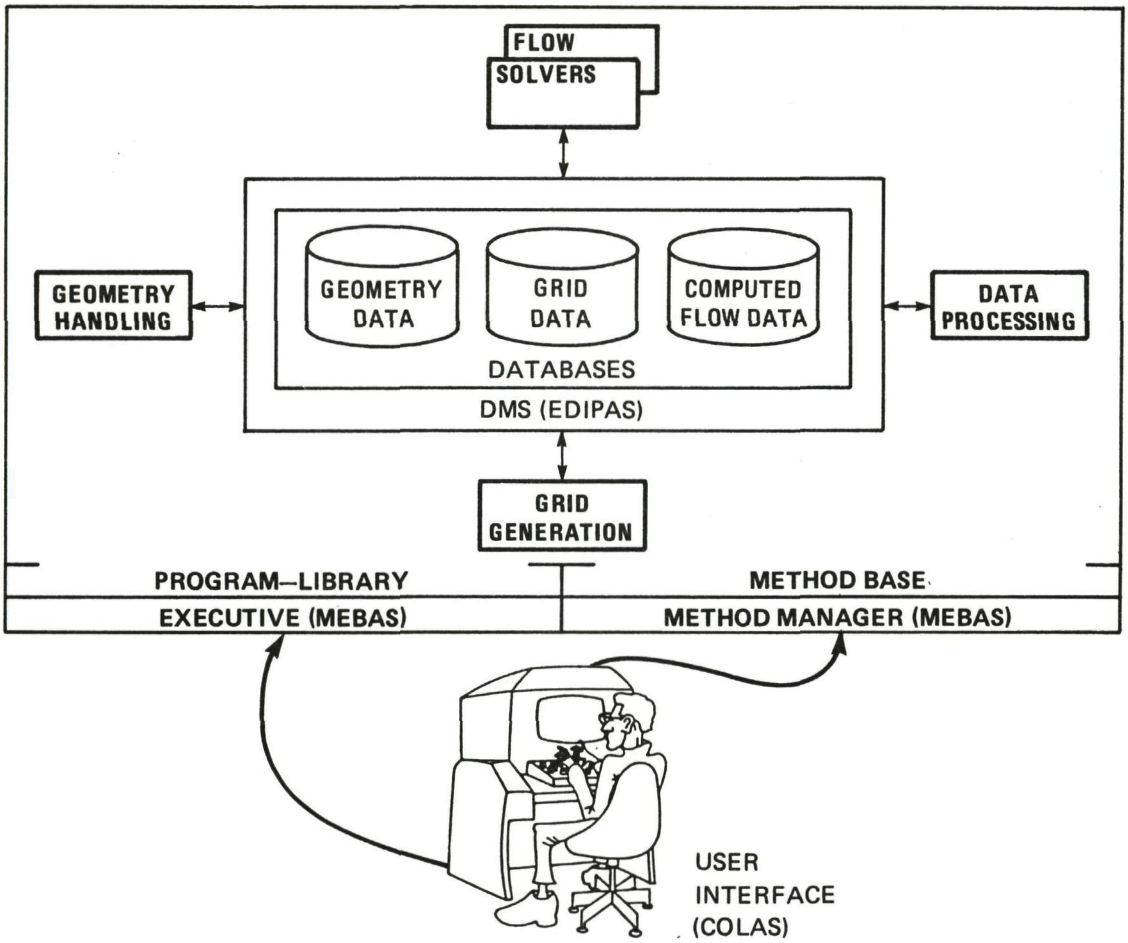


Fig. 11 Technical concept for computational aerodynamics infrastructure at NLR

NATIONAAL LUCHT- EN RUIMTEVAARTLABORATORIUM

Anthony Fokkerweg 2, 1059 CM AMSTERDAM
Postbus 90502 , 1006 BM AMSTERDAM
Tel. (020) 511313 - telex 11118 (nlraa nl)
Telegram-adres: Windtunnel Amsterdam