

AD-A184 580

COMPUTATIONAL FLUID DYNAMICS IN THE UNITED KINGDOM(U)  
ROYAL AIRCRAFT ESTABLISHMENT FARNBOROUGH (ENGLAND)  
M G HALL ET AL APR 87 RAE-TM-AERO-2098 DRIC-BR-102723

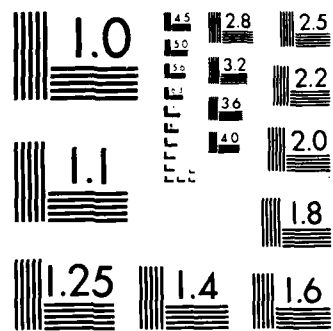
1/1

UNCLASSIFIED

F/G 20/4

NL





MICROCOPY RESOLUTION TEST CHART  
NATIONAL BUREAU OF STANDARDS-1963-A

DTIC FILE COPY

BR102723

TECH. MEMO  
AERO 2098

TECH. MEMO  
AERO 2098

2

AD-A184 580



ROYAL AIRCRAFT ESTABLISHMENT

COMPUTATIONAL FLUID DYNAMICS IN THE UNITED KINGDOM

by

M. G. Hall  
S. P. Fiddes

April 1987

DTIC  
SELECTED  
SEP 11 1987  
S E D

Procurement Executive, Ministry of Defence  
Farnborough, Hants

87 8 17 07

R O Y A L   A I R C R A F T   E S T A B L I S H M E N T

Technical Memorandum Aero 2098

Received for printing 3 April 1987

COMPUTATIONAL FLUID DYNAMICS IN THE UNITED KINGDOM\*

by

M. G. Hall

S. P. Fiddes

SUMMARY

A review is presented of computational methods in aerodynamic research and design, with application of the methods and associated computing facilities included. The review begins with a brief survey of the field, to give an overall view and to identify what seem to be notable features. These are then described in turn. The first is a development by Hall and his colleagues of accurate and fast schemes for solving the Euler equations, based on the finite-volume cell-vertex methods introduced by Denton and Ni. Next is a development of a block-structured ('multiblock') grid generation technique by the Aircraft Research Association. There follows an application of this grid generation scheme, in conjunction with an Euler solver, to the calculation of the transonic flow past a wing-body-canard configuration. Finally, an application of viscous-inviscid interaction techniques to the design of a fan rotor is described.

\* Invited AIAA Paper No. 87-1132-CP, presented at AIAA 8th Computational Fluid Dynamics Conference, Honolulu, USA, 9-11 June 1987.

*Copyright*

©

*Controller HMSO London*

*1987*

*This Memorandum expresses the opinions of the authors and does not necessarily represent the official view of the Royal Aircraft Establishment.*

Accession For	
KEYS GRAM	<input checked="" type="checkbox"/>
BTIC TAB	<input type="checkbox"/>
Unannounced	<input type="checkbox"/>
Justification	
By	
Distribution/	
Availability Codes	
Dist	Avail and/or Special
A-1	

QUALITY INSPECTED  
2

(TSP) equation are still in use. However the unreliability of TSP methods when perturbations are not small, and of all potential methods when shock waves are present, no longer outweighs their relative simplicity. It seems appropriate, as our TSP methods are being retired, to note the key role they have played in the history of CFD. Their introduction by Murman and Cole and their revolutionary effect on the development of aerodynamics were discussed in an earlier review given at the 5th AIAA CFD Conference in 1981<sup>4</sup>. Since that time a number of aircraft designed with the aid of such methods have gone into service. For the UK the first of the 'CFD airliners' was the A310 Airbus. The impact on the A310 of the availability of CFD methods that could adequately calculate flows with shock waves about wing-body combinations has been summarised by Jupp<sup>5</sup>. He notes that in comparison with the inferior ('pre-CFD') A300 wing design, the A310 design required one-third the number of windtunnel models and 13 months less time to arrive at the final design. The importance of CFD in the design of the A320 wing is similarly described by Back and Wedderspoon<sup>6</sup>. The A320 is proving an outstanding commercial success, with over 400 orders placed before the first flight.

Introduction

The review begins with a survey that is restricted to the range of computational methods that are in use or under active development for application in aerodynamic research and design. The survey deals first with methods that are well established but for which no major developments are envisaged, and then moves on to methods that are in use and still undergoing active development. The latter fall mainly into four classes - methods for the Euler equations, for viscous-inviscid interactions, for the Navier-Stokes equations and for grid generation - and these are covered in turn. For each class of method some relevant applications are noted. In the course of the survey a number of features are singled out for more detailed description in the remainder of the review. The survey concludes with a brief account of computing facilities.

Another, perhaps less well known, field where potential methods have had a revolutionary role is that of propeller design. We note here the radical improvements that have been obtained following work at the Aircraft Research Association (ARA) on the design of high-performance propellers for commuter aircraft<sup>7,8</sup>.

In spite of the restriction to aerodynamics the range of methods covered here is so wide that no method can be discussed in depth. Reference may be made to a number of specialised reviews by British authors that give a more comprehensive account of their subject matter. In particular the reader's attention is drawn to the recent review by Dunham<sup>1</sup> on turbomachinery applications, the detailed discussion of viscous-inviscid interactions by Lock and Williams<sup>2</sup>, and the survey by Roe<sup>3</sup> of characteristic-based schemes for solution of the Euler equations.

There are, however, certain special roles in which potential methods seem likely to remain useful for some time. Where trends rather than absolute accuracy are important (such as in preliminary design studies, or in unsteady calculations for flutter estimation) the simple TSP methods will remain useful. On the other hand, full-potential methods offer high accuracy for subcritical flows and thus can provide datum solutions against which the newer Euler methods can be judged. Panel methods will remain useful in the calculation of low-speed flows, while field-panel methods continue to offer relatively simple solutions for mildly supercritical flows past complex configurations. Finally, potential methods will continue to have a role in composite schemes, such as zonal methods and viscous-inviscid interaction schemes, where the potential equation is valid for an outer part of the flow.

Survey of Field

Established Methods

Methods for solving the full-potential equation and the transonic small-perturbation

Another type of method that provides a high standard of solution is the space-marching scheme for the Euler equations. The limitation, with such schemes, to supersonic flow everywhere is leading to the predominance of time-marching Euler methods for practical applications in the UK. At present, however, it is difficult to assess the accuracy of the latter whereas, for example, accurate solutions have been obtained for the wave drag of a delta wing calculated via a space-marching method<sup>9</sup>.

BM 2007 AV NI

### Methods for the Euler Equations

Jameson's original cell-centred finite-volume scheme for the Euler equations has been developed by British Aerospace (BAe) for extensive practical applications. The work involved, and some applications, have been described by Doe, Brown and Pagano<sup>10</sup>, while other UK applications, for military aircraft, have been described by Marchbank<sup>11</sup>. An important development of the work has been its integration with the ARA multiblock grid generation technique.

Another feature of UK work is the development of multigrid finite-volume schemes of the cell-vertex type. These techniques, as originally proposed by Denton and Ni suffered from shortcomings in accuracy and speed (see, for example, references 12 and 13), but Hall<sup>14</sup> and his colleagues at RAE have achieved substantial improvements in these two key areas and have proceeded on to extensions of the basic multigrid method<sup>15</sup>, the inclusion of adaptive grids<sup>16</sup>, and the development of a three-dimensional method<sup>17</sup>. Morton and Paisley<sup>18</sup> and Roe<sup>19</sup> have placed such schemes on a more secure mathematical foundation. Reference 18 also includes results for two-dimensional cell-vertex Euler solutions with fitted shocks. An account of these developments of the cell-vertex method will be presented below.

### Viscous-Inviscid Interactions

UK work on viscous-inviscid interactions is described in the detailed and wide-ranging review by Lock and Williams<sup>2</sup>. The UK continues to favour integral methods for boundary layer calculations mainly because of their speed and simplicity. They have also been developed to yield sufficiently accurate solutions for a useful range of practical problems. Developments in techniques for coupling boundary layer methods to inviscid methods (both Euler and potential) have produced methods that can predict flows involving boundary layer separation, with most success being achieved for two-dimensional flows thus far. In three dimensions simple, direct coupling has been implemented in a number of methods, while for two-dimensional methods direct, inverse, semi-inverse and quasi-simultaneous coupling procedures have all been used. These types of coupling and their relative merits are discussed in the above review. Cross<sup>20</sup> gives a description of a quasi-simultaneous technique and its use in conjunction with a panel method. An example of the application of coupled viscous-inviscid methods to the design of a fan rotor is described by Ginder and Calvert<sup>21</sup>, and this is discussed below as one of our chosen highlights.

### Methods for Navier-Stokes Equations

Although there is considerable research effort in the UK relating to the Navier-Stokes equations, there are relatively few methods that are in use or being actively developed for aerodynamic research and design. The methods that have found application are based on methods developed outside the UK. Thus Rolls-Royce have under continuing development three-dimensional methods based on the work of J. and J.G. Moore<sup>22</sup> and Thompkins and his colleagues<sup>23</sup> in USA. These methods have been used as analysis tools, in the

design of the high-pressure turbine nozzle guide vanes for the RB211-535 and RB211-524, and in the design of the compressor outlet guide vanes for the IAE-V2500. Progress in the development of practical Navier-Stokes methods is presently hampered not only by inadequacies in computing capacity, but also by fundamental difficulties: the generation of suitable grids, the modelling of turbulence, and the attainment of satisfactory accuracy and convergence with grids that are very highly stretched or distorted.

### Grid Generation

Grid generation methods for complex configurations make up the last class of methods that are both in use and under active development. Because of their practical importance, several different approaches have been explored. These include non-aligned grids<sup>24,25</sup>, where the grid is not aligned with the body surface (leading to simple grid generation but complicated treatment of boundary conditions), unstructured grids, overlapping grids, embedded grids<sup>26</sup>, solution-adaptive grids<sup>16</sup> and block structured, or multiblock, grids<sup>27,28</sup>. The power of the multiblock approach has encouraged UK researchers to put considerable effort into the development, evaluation and practical use of multiblock methods. A multiblock framework can incorporate the other types of grid and be applied to most (if not all) practical configurations. The computational advantages of having a regular structured grid can be retained for individual blocks, although the array of blocks themselves will in general be unstructured. The development of the method has involved study of several questions: how to define individual blocks and to set up the array of blocks to cover the complete field; what degree of continuity of grid lines at block interfaces is optimal; how to control the point or cell density; how to avoid singularities; and what degree of automation of grid generation is feasible, or even desirable? An account will be given here of the multiblock method developed at the Aircraft Research Association (ARA), and its application to computing the flow past a wing-body-canard configuration.

### Computing Facilities

The computing facilities used for CFD in the UK range from micro and mini-computers and graphics workstations through conventional mainframes to supercomputers. The main users of supercomputers are groups in the airframe and aero-engine industry, who access such facilities through bureaus. Other major supercomputer users are RAE and ARA (using the Cray 1-S sited at RAE Farnborough) and Universities (who have network access to a Cyber 205, a Cray X-MP and 2 Cray 1's). The larger institutions have their own mainframe computers, but the trend is towards greater use of supercomputers for CFD. Associated with this trend is a rapid growth in the facilities available for pre- and post-processing, particularly graphics workstations. There is a growing awareness of the need for more fast memory in supercomputers. This need may not be met by out-of-core memory unless it is both efficient and transparent in its use.

The latest supercomputers are characterised by having a small number (typically four) of very

high speed vector processors sharing a large amount (megawords) of common memory. These offer limited prospects for multiprocessing, both in the simpler single-instruction multiple-data (SIMD) mode where the processors execute the same instruction simultaneously on different data, and in the multiple-instruction (MIMD) mode where, even with only a few processors, the programming is far from trivial. However, in the United Kingdom, the concept of multiprocessing hardware has been taken much further. Two noteworthy developments will be discussed - the Distributed Array Processor (DAP) originally developed by International Computers Limited and the Transputer developed by Inmos.

The DAP is a SIMD device intended for use with a host computer. By analogy with the concept of a vector processor, the DAP may be viewed as a matrix processor. The processor array currently comprises 1024 single-bit processors arranged as a 32 by 32 array. Each processor is supported by a 32K bit memory. Communication between processors is local, to the four nearest neighbours. Grosch<sup>29</sup> has recently described the use of a DAP in conjunction with a two-dimensional Navier-Stokes solution method based on relaxation. The architecture of the DAP, as a matrix processor, is well suited to such an approach. However, the present DAP architecture is rather limited in size and processing power, and substantial increases in memory, speed and communications capability of a DAP-like device are required before practical three-dimensional problems can be contemplated.

By contrast, the Transputer (which is to be regarded as an element of a larger system) comprises a powerful 32-bit processor giving 10 MIPS processing speed, 2 kilobyte of memory and four links each providing a 10 megabit per second data highway, ie it does not use a bus structure for data transfer between processors. It comes on a single chip and is a useful computer in its own right. However, it is designed to be a building block for a true multiprocessor system in that a Transputer system can be configured to the user's requirements and operated in either SIMD or MIMD mode, or in both modes simultaneously. Recent developments include the ability to reconfigure the system by software.

These impressive developments in hardware and computing architecture now pose a challenge for the CFD method developer - how to use them. The burden of using a multiprocessor system effectively rests with the applications programmer - there are, as yet, virtually no software tools that will take a code developed on a 'general purpose' computer and partition the code for execution over a number of processors - the programmer must perform this 'task splitting' himself. At present this is not a desirable or practical computing environment for code developers concerned with developing new methods or trying new applications. However, it may be feasible to implement on a multiprocessor system a proven code that is used on a production basis. The multiblock concept suggests a possible way forward in this, where each processor could calculate the flow in one block. Note, however, that the wing-body-canard configuration referred to above uses 430 blocks, and a 'general-purpose' multiblock multiprocessor system may therefore require several hundred processors, not all of

which would be used all of the time. A more realistic approach may be to consider 'super-blocks', where a group of blocks are associated with each processor. In any case multiblock seems to offer a natural 'task-splitting' scheme for use of a multiprocessor system in the calculation of flowfields past complex configurations.

#### Cell-Vertex Multigrid Solution of the Euler Equations

##### Cell-Vertex Formulation

The cell-vertex formulation<sup>14</sup> differs from the better known cell-centred formulation in having the dependent variables specified at the vertices of a computational cell rather than at the centre. In both formulations the fluxes through the sides of the cell are approximated in order to evaluate the flux balances and hence the changes in dependent variables for the entire cell. The approximation used in the cell-vertex scheme for the flux through a cell side is the trapezoidal rule for integration and hence is second-order accurate in cell size whatever the grid, whereas the corresponding cell-centred approximation, being an average of quantities at the centres of adjacent cells, will only be second-order accurate on nearly uniform grids. An additional advantage of the cell-vertex formulation is that no extrapolation is required to obtain values for the dependent variables on body surfaces.

Recently, studies of accuracy by Morton and Paisley<sup>18</sup> and Roe<sup>19</sup> have yielded the following results. For a quadrilateral cell with sides  $O(h)$  and flux balance  $O(h^2)$  the truncation error in the flux balance is  $O(h^3)$  or first order in the cell-vertex formulation, irrespective of the size of the adjacent cells. For a quadrilateral with opposite sides differing by  $O(h^2)$ , approximating a parallelogram, the truncation error is  $O(h^4)$  or second order. Such quadrilaterals are frequently found in practice, for example when the grid is generated analytically, or by repeated bisection of an arbitrary coarse grid. In contrast, the truncation error of the cell flux balance on the cell-centred formulation depends on the sizes and shapes of the adjoining cells. Analysis in this case is difficult. However, for a non-uniform rectangular grid (for which the cell-vertex flux balance is second-order accurate) the ratio of successive cell sides must be  $[(1 + O(h)) : 1]$  for first-order accuracy and  $[(1 + O(h^2)) : 1]$  for second-order accuracy. Furthermore, if the vertex of a cell of an initially rectangular grid is displaced by an amount  $\epsilon$ , then  $\epsilon$  must be  $O(h^2)$  for first-order accuracy, while  $\epsilon$  of order  $h^3$  is required for second-order accuracy. (By contrast, the cell-vertex scheme yields second-order accuracy for  $\epsilon$  of order  $h^2$ .) The truncation error arising from the simple averaging of quantities at the centre of adjacent cells can be reduced (in principle) by weighting the average. However this complicates the formulation and, more seriously, makes it more difficult to ensure the convergence of the iterative process.

Spurious modes are another possible source of error. Analysis<sup>18</sup> shows that for a model two-dimensional problem the cell-vertex and cell-

centred formulations yield the following spurious modes:

Cell-vertex:	+ - +			
	- + -			
	+ - +			
Cell-centred:	+ - +	+ - +	+ + +	
	- + -	+ - +	- - -	
	+ - +	+ - +	+ + +	

Whilst the cell-vertex scheme has one spurious mode (the well-known chequer-board mode), the cell-centred scheme has three possible modes. Thus it should be expected that the prevention and damping of spurious modes will be easier in the cell-vertex than in the cell-centred formulation.

#### Iteration to Steady State

In the original version of Hall's cell-vertex multigrid method the solution was iterated to the steady state by use of a Lax-Wendroff time-stepping algorithm in combination with a simple multigrid algorithm. The emphasis during the development of this early method had been feasibility and accuracy, rather than speed. Two devices for increasing speed have since been incorporated, and are now described.

Consider first the changes calculated on the coarser grids of a multigrid cycle. In the original method the changes were simply a redistribution and re-scaling of changes calculated on the previous finer grid. It would be expected that a direct integration of the Euler equations (in a form appropriate for defining changes in the solution) would accelerate the rate of convergence to a steady state - indeed, Jameson adopts such a technique. Thus a time-stepping algorithm of the Runge-Kutta type was introduced<sup>14</sup>, for calculations on the coarser grids only. It was tried at all grid levels but found then to be slower. The algorithm differs from Jameson's, however, in that it does not include fourth-order spatial dissipation, but includes a temporal damping provided by part of the Lax-Wendroff increment. This adds no additional truncation error to the steady-state solution and is also more compact.

The second accelerative device is a simple form of residual averaging, applied on the coarser grids only. In the original method changes on the coarser grids were interpolated linearly to give the changes at each of the points on the finest grid. The interpolation is now smoothed, thus averaging the residuals.

The convergence and stability of the above scheme have been studied by Morton and Paisley<sup>18</sup>. With the cell-vertex formulation and Lax-Wendroff time-stepping it is shown that Ni's simple, non-reflective boundary condition (in which any residuals lying outside the computational domain are set to zero) is a necessary condition for convergence. In cell-centred formulations there is no simple boundary treatment. Morton and Paisley deduce that provided the boundaries are treated correctly the residuals in the individual cells will be decoupled so that at convergence all the individual residuals will vanish, rather than any averages. Non-uniformity of a grid is shown to be destabilising. Hall had found it effective, in averaging the residuals from four neighbouring

cells to obtain an update for the common vertex, to weight the residuals by individual cell areas. This weighted average is shown to emerge naturally from an alternative finite-element formulation of the Taylor-Galerkin type. Multi-stage Runge-Kutta algorithms are shown to be destabilised by non-uniformity. With the cell-vertex formulation this can be alleviated by weighting residuals, but further damping may be required. With the cell-centred formulation the addition of extra dissipation can stabilise the iteration, and indeed the fourth-order dissipation of Jameson is a popular remedy.

#### Shock Fitting

Although the cell-vertex formulation has important advantages in accuracy and in the number of spurious modes, it will still be subject to significant errors if shock waves are captured, in common with other formulations. Morton and Paisley<sup>18</sup> have developed a shock-fitting scheme and thus allow the cell-vertex scheme to yield accurate solutions for supercritical flows with embedded shock waves. The scheme involves an adaptive shock-aligned grid. The conventional shock-capturing version of the cell-vertex scheme is used to give an initial estimate of the shock position. The grid is then aligned with the estimated shock location, and the grid line running along the shock is treated as a boundary of the computational domain. A cell on the upstream side of the shock is treated as a supersonic outflow boundary cell, while a cell on the downstream side is treated as a subsonic inflow cell, with the Rankine-Hugoniot conditions replacing the usual inflow matching conditions. At each time step the calculation yields a shock speed. The shock is moved accordingly and the grid re-aligned repeatedly until a steady state is reached.

#### Results for NACA 0012 Aerofoil

A selection of results for the NACA 0012 aerofoil is presented to illustrate the accuracy and speed of the cell-vertex multigrid method described above. Figure 1 shows Mach number distributions on the aerofoil surface for captured and fitted shocks. Table 1 shows a comparison of calculated lift coefficients for cell-vertex and cell-centred methods, for three test cases on both coarse and fine grids<sup>30</sup>. The same C-mesh was used

Table 1 Calculated lift coefficients. NACA 0012

Case	No. cells	Captured shocks		Fitted shocks cell vertex	Other results
		Cell centred	Cell vertex		
$M_\infty=0.30$	128x16	1.23	1.271		
$\alpha=10^\circ$	256x32	1.26	1.273		1.273 <sup>31</sup>
$M_\infty=0.80$	128x16	0.337	0.366	0.360	
$\alpha=1.25^\circ$	256x32	0.365	0.353		0.362 <sup>32</sup>
$M_\infty=0.85$	128x16	0.401	0.380	0.382	
$\alpha=1.0^\circ$	256x32	0.457	0.406		0.394 <sup>32</sup>



for both sets of shock-capturing calculations. Included in the table are results obtained by cell-vertex shock-fitting and results from other accurate calculations. For the first test case an accurate potential flow solution has been obtained by Salas et al<sup>31</sup>; it can be seen that the corresponding cell-vertex results are in very good agreement with the potential flow result. It can also be seen that the results from use of shock fitting, in the second and third test cases, agree well with the results obtained by Pulliam and Barton<sup>32</sup> with shock capturing on an exceedingly fine grid. The results from the cell-vertex method, with captured shocks, are closer to the accurate results than the results from the cell-centred method. Moreover, the cell-vertex results obtained on the coarse grid are close to those from the fine grid.

Table 2 shows a comparison of CPU times (Cray 1-S) for the lift to reach within 0.25% of the fully converged value. The benefits of including Runge-Kutta time-stepping and residual averaging in the cell-vertex method are clear. The results from Jameson's 1985 cell-centred method<sup>33</sup> (which included both the above accelerative devices) were obtained on the same grid as used for the cell-vertex calculations.

Table 2 CPUsec for lift to converge to 0.25%

No. cells	Hall multigrid 1985	Hall multigrid+ 1986	Jameson multigrid+ 1985
128x16	7.3	6.3	7.7
256x32	29.5	13.3	24.7

Extension to Three-Dimensional Flows

A three-dimensional version of the cell-vertex multigrid method for swept wings, based on the RAE work for two-dimensional solutions, has been developed by Salmond<sup>17</sup>. Only Lax-Wendroff time-stepping was used in this scheme. More recently the method has been accelerated by incorporating Runge-Kutta time-stepping and smoothed interpolation (residual averaging), and a typical CPU time (Cray 1-S) for the lift to converge to 0.5% of the final value is 140 seconds for a 128 x 16 x 24 grid. Another improvement has been the incorporation of an asymptotic far-field boundary condition that more correctly represents the behaviour of a lifting solution in the far field. Klunker's<sup>34</sup> transonic small-disturbance approximation for the far-field solution is used. This involves an integral over the surface of the wing to determine the appropriate condition, for every point on the far-field boundary. Table 3 shows, for the ONERA M6 wing, the variation of total lift with distance of the far-field boundary from the wing. It can be seen that matching the boundary conditions to a small-disturbance far-field solution, rather than free-stream flow, allows the boundary to be brought significantly closer to the wing without loss of accuracy.

Table 3 Variation of lift with boundary distance. ONERA M6 wing,  $M_\infty=0.84$ ,  $\alpha=3.06^\circ$

Distance chords (spans)	Free-stream condition	Small-disturbance condition
2.5 (0.84)	0.271	0.287
5 (1.69)	0.285	0.290
10 (3.37)	0.286	0.288

Multiblock Grid Generation

Introduction

The guiding principle of the multiblock grid generation method developed at ARA<sup>27,28</sup> is that with each component of the configuration about which a grid is to be generated there is associated a particular type of grid that is best suited to describing the flow past that component. This contrasts with the more common approach where a block-based structure is seen primarily as a means of reducing the in-core memory requirements of a method. The ARA multiblock method can be used to generate H-grids everywhere in the field, but it can do far more; in particular, it offers a capability for automated generation of component adaptive grids. The price paid for such flexibility and power is complicated program logic. The blocks themselves are simple, non-overlapping volumes, each with six faces and eight corners. A key feature is that only one type of boundary condition (eg solid boundary, specified transpiration velocity, continuity, etc) is permitted on each face of each block. This simplifies the logic but increases the number of blocks required to cover the flow field.

The grid generation procedure can be broken down into three stages: topology definition, surface grid generation and field grid generation. Successful topology definition is crucial. A good topology can ease the grid control problems both on the surface and in the field. An inappropriate topology can make it impossible to generate a satisfactory grid. Each of the three stages are now described in turn. The account of multiblock concludes with a discussion of the prospects for automated grid generation.

Topology Definition

The block topology is the arrangement of the blocks with respect to each other. Given an arbitrary configuration the definition of an appropriate topology is a challenging task. Inspection and trial, aided by experience and interactive computer graphics, is an obvious strategy. Clearly, a systematic approach, with access to a library of successful topologies and at least some automation, is desirable. The present method requires the setting up of a 'topology file' which lists, for each block face, the type of boundary condition, the adjacent block, the adjacent face of that block and the orientation of the adjacent block with respect to the current face. This file controls the complete multiblock procedure.

A guide to the construction of a topology may be obtained from study of Figure 2. Three basic units, "O", "C" and "H" grids, are adopted

RD07 2V 11

for the construction of a multiblock grid. O-grids are computationally efficient, but present topological difficulties in the multiblock framework. H-grids are the least efficient computationally but topologically the simplest. C-grids lie in between and are particularly well suited for flows past wings. The figure shows two possible topologies for a simple wing-body combination:

- (i) C-H-H, ie C-grid around the wing section, H-grid spanwise over the wing tip, and H-grid vertically around the body section.
- (ii) C-H-O

Topology (i) is simple enough to be constructed by inspection. The section AA' shows the H-grid around the body section, and the (spanwise) H-grid over the tip. It also shows how the wing thickness is accommodated by opening up or splitting a surface on the H-grid. Sections BB' and CC' show how the wing, and its extension beyond the tip, is embedded in a C-grid, and how the C-grid is in turn embedded in an H-grid. A count of the number of blocks in the BB' and CC' sections gives the total, 24, for the complete topology.

Topology (ii) can now be constructed. The sections CC' and DD' are identical to BB' and CC' respectively in topology (i). However, the introduction of an O-grid around the body complicates the topology and, as shown in sections AA' and BB', care is needed in interfacing the O-grid with the C-grid over the wing and with the H-grid above and below the body. Here the total number of blocks is 42. It is apparent from considering such a simple configuration that the policy of admitting only one type of boundary condition on each block face does indeed lead to a large number of blocks.

#### Surface Grid Generation

Suitable grids are generated on the surface of the configuration, and on the outer boundary of the computational domain, before constructing grids in the interior of the domain. The first step is to convert the input geometry of each component of the configuration to a continuous function of two parametric coordinates using the Coons bi-cubic patch formulation. This transformation from a discrete to a continuous description of the surface allows the component intersections to be calculated accurately. The surface grids, with a topology derived from the topology defined above, are then generated in terms of the parametric coordinates by solving a set of non-linear elliptic equations as proposed by Thompson, Thames and Mastin<sup>35</sup>, using over-relaxation or approximate factorisation methods. The solution is then mapped back to physical space by use of the bi-cubic patches, which ensures consistency with the original geometry, and provides boundary conditions for the generation of the field grid.

#### Field Grid Generation

The approach adopted at ARA to generate the field grid has been to solve the elliptic Thompson equations in three-dimensional form on a block-by-block basis for the entire field. The physical locations of the interior block boundaries become fixed in the course of an iterative solution. The aim is to produce a grid with smooth variation

across block boundaries as well as within a block. The presence of singular points and lines is accepted (and may be inevitable). To control the distribution of grid points within a block the control functions used as forcing terms in the Thompson equations are expressed in terms of the point distribution on a given boundary, as suggested by Thomas and Middlecoff<sup>36</sup>. When used with multiblock, considerable care is needed with this approach. Grid cross-over can occur and point distributions frequently fall short of the quality attainable in two-dimensional grids. Grid cross-over, which is apparently only possible with discretised forms of the Thompson equations, seems most likely to occur near sharp edges, such as wing tips and trailing edges.

#### Automatic Grid Generation

The generation of a satisfactory grid for a practical configuration is an exceptionally demanding task, requiring both skill and experience and absorbing time and effort. It is desirable, therefore, to seek means of automating the task as far as possible, while maintaining the quality of the grid. This is not a well-defined objective. The possible level of automation, what is meant by a satisfactory quality of grid, and the nature of the task itself are indefinite and varying. However, some guidelines can be identified.

The automation of topology definition presents the greatest challenge. This is the area of grid generation where skill and experience are of tantamount importance, and the generation of a suitable topology is a necessary condition for the generation of a good grid. However, topology definition is an essentially creative task - for a given configuration there is an unlimited number of possible topologies - and so automation of this process is particularly difficult. By contrast, once the topology has been defined, the generation of the surface and field grids is relatively straightforward.

Complete automation of grid generation for arbitrary configurations does not, at present, seem feasible. However, for particular types of configuration, such as a wing-body-canard, the range of appropriate topologies can be limited and it becomes feasible to develop an automatic routine for topology definition within this subset of topologies. Nevertheless, it is expected that some degree of user interaction will still be desirable.

The ARA work<sup>28</sup> on grid generation for wing-body-canard configurations illustrates how such an approach might be feasible. Within the class of wing-body-canard configurations, there is a large range of 'metrically' different configurations, associated with different wing sweeps, different relative location of canard and wing, etc. Although the range of appropriate topologies is limited there are nevertheless many possible topologies for this range of configurations. In order to proceed, ARA choose to have C-grids around streamwise sections of lifting surfaces, an O-grid around the body section and an H-grid around spanwise sections of the lifting surfaces. These grids are embedded in a 'global' H-grid. This provides a basis for automatic topology definition. The user must supply the key geometrical features

that determine the topology, such as whether the canard is to lie on the same coordinate surface as the wing. A simple format is provided for this as a 'user interface'. In addition, the user specifies the number of blocks in each of the coordinate directions of the global H-grid. The number actually used may thus be greater than the minimum required for a given topology, for example to give increased grid density in appropriate regions. The surface grids are generated with a surface topology derived from the field topology, and require two inputs. The first is used to define the grid density in the region of leading and trailing edges of the lifting surfaces, while the second, from the topology definition, provides for consistent grid densities on the surface and on the outer boundary of the computational domain. Generation of the field grid is then a straightforward process.

### Examples of Applications of CFD Methods

#### Analysis of Wing-Body-Canard Design

The multiblock grid generation scheme described above has been used, in conjunction with the BAe method<sup>10</sup> for solving the Euler equations, to calculate the inviscid compressible flow about a wing-body-canard configuration, known as M165. This is a research model that has been the subject of a number of wind-tunnel tests.

The first step in the use of the multiblock scheme is the definition of a suitable topology. Here a C-H-O topology is adopted with an O-grid around the body section. Many other reported grid generation techniques for wing-body combinations use H-grid structures or C-grids alone, with the result that the grid around the body is inadequate, particularly in the important nose region.

Canard configurations can present appreciable problems for grid generation if there is substantial spanwise variation in the distance between the canard and the wing. For the configuration considered here, the trailing edge of the canard and the wing leading edge lie in the same vertical plane at the body side, but the tip of the canard lies well upstream of the wing leading edge, due to the large sweep of the wing. This spanwise variation of relative position of the canard and wing imposes conflicting requirements on the body-side and canard-tip grids, and this dilemma has received special attention in the present example in that two distinct topologies for the region of the canard and wing have been tried.

Figure 3 shows one of the surface grids used. The grid has 44 points around the canard section, with 8 spanwise stations, while the wing was represented by 78 points around the section and 15 spanwise stations. The O-grid structure around the body is clearly visible. The complete grid comprised some 88000 cells, constituting 430 blocks.

Three sets of results are now presented. The first set are for a freestream Mach number of 0.9 and incidence of 6 degrees. Figure 4 shows a comparison of predicted and measured pressures on the wing (the canard was not pressure-plotted on the model). Good overall agreement has been obtained, although the suction peaks near the

leading edge are poorly represented, indicating that the grid near the leading edge needs refining. The predicted shocks are stronger and further aft than the observed ones. However, the incorporation of viscous effects would produce a weakening of the predicted shocks and forward movement in position - we can infer therefore that the present inviscid results for shock strength and position are consistent with the measurements.

The second comparison, for a supersonic onset flow (Mach number 1.2), again at 6 degrees incidence, is presented in Figure 5. Here the pressure distribution on the wing shows a strong trailing-edge shock that is well-predicted by the method. Again, overall agreement between theory and experiment is good.

A further set of results is shown in Figure 6. Here, the method has been used to predict the effect of the canard on the wing pressures at  $M = 0.9$  and 6 degrees incidence. The expected effect of the canard downwash can be seen in the results, namely a reduction in wing loading, particularly inboard of the canard tip position, again in good overall agreement with experimental results.

#### Design of an Advanced Civil Fan Rotor

The next example of the application of CFD methods is the design of a fan rotor for a civil turbofan engine<sup>21</sup>. The method used comprises three main components:

- (i) S1 - an inviscid Euler method (based on the method of Denton)
- (ii) BYL2 - an integral boundary layer method
- (iii) S2 - a streamline curvature throughflow analysis.

These three elements are combined in an iterative manner to produce a quasi-three-dimensional technique for analysing fan-blade performance.

The Euler method and integral boundary layer method are combined (using the inverse coupling techniques described in Reference 2) to give a method (referred to as S1BYL2) that is capable of predicting blade-to-blade flows including separation induced by shock waves or marked adverse pressure gradients (diffusion).

The blade-to-blade calculation takes place on a specified axisymmetric stream surface, and incorporates the effects of rotation, varying radius and variation of stream tube thickness in the axial direction. This information is obtained from the throughflow calculation method (S2), derived assuming axisymmetry. A number of streamlines along a radius are 'tracked' by the S2 calculation, which accounts for the blockage, momentum loss and change of axial momentum. This data is obtained from the blade-to-blade calculation, and so a complete solution is obtained by coupling the S2 and S1BYL2 methods in an iterative manner.

The technique has been used as a 'design by analysis' method in conjunction with a family of parametrically defined blade profiles. The suction surface of the blade is made up of four circular arcs, whilst the pressure side is con-

structured from two cubic splines. The suction surface shape is defined from aerodynamic considerations, and the lower surface shape is then chosen primarily to give a structurally feasible blade section.

The transonic fan rotor specified for the design is typical of a modern, high bypass-ratio turbofan. A bypass ratio of 1.8 was specified with a tip speed of 450 metres per second. The inlet Mach number varied from 0.8 to 1.5 hub to tip, with a corresponding variation of flow angle of 35 to 65 degrees.

The main considerations during the design process were to avoid choking due to thickness and close pitch at the blade root, to maximise diffusion without flow separation at the mid-span, and to avoid shock-induced separation at the tip. This latter requirement meant reducing the Mach number ahead of the shock by supersonic compression.

Care was taken to ensure that the three types of blade section that met these three design considerations could be 'stacked' to produce a realistic blade shape, and the S2 calculation was repeated a number of times to ensure the sections were matched. The S2 calculation used 19 streamlines, while 8 radial stations were chosen for the blade-to-blade calculations.

Typical sections from the resulting design are shown in Figure 7, together with predicted flow properties at the design condition. Corresponding results for an earlier conventional design are included. The results show reduced pre-shock Mach numbers for the new design, and boundary layers that are attached, unlike those for the earlier design. This leads to a marked reduction in loss of total pressure. Overall, the fan was predicted as being some 2% more efficient than previous conventional designs, and this gain in performance was in fact exceeded during tests of the rotor. The improvement is a prime example of what can be achieved by combining the Euler methods of the cell-vertex type with the boundary layer calculation and coupling techniques described in Reference 2.

#### Concluding Remarks

It is clear from the present review that while the UK has a useful capability in the numerical simulation of flows past aircraft and propulsion configurations, this falls far short of the ultimate practical requirement of simulation for complete aircraft and propulsion systems. How our goals might be reached is an important question, but the focus here on the current status of CFD has left little scope for discussion of the question. Currently our discipline is rapidly growing and changing, with many promising approaches for exploration and evaluation. New, as yet unimagined, approaches will no doubt emerge as well, so adding to the difficulty of choosing the approach to follow. However, we expect some resolution of the problem of choice in the next few years. Thus we expect to see progress in the treatment of turbulence and shock waves. Turbulence may be modelled crudely but simply, or with much elaboration (for example, using large-eddy simulation). Shocks may be captured or fitted, probably on adaptive grids. Grids, in turn, may

be structured and thus computationally efficient but not well suited for some configurations, or unstructured and thus relatively inefficient but versatile, or, in a multiblock framework, partly structured and partly unstructured. For strongly interacting flows the use of viscous-inviscid coupling techniques may give way entirely to solution of the Navier-Stokes equations, but new couplings may be devised that successfully exploit the distinct viscous-inviscid structure of high Reynolds-number flows for computational efficiency. Finally, large-scale computing may evolve slowly, with reliance on the present super-computer architecture and a trend towards a modest degree of multiprocessing, or be transformed by new developments in the simultaneous use of large arrays of processors.

#### References

1. Dunham, J., "The Role of Flow Field Computation in Improving Turbomachinery", ICAS-86-3.7.3, ICAS Proceedings 1986, pp 967-980.
2. Lock, R.C. and Williams, B.R., "Viscous-Inviscid Interactions in External Aerodynamics", to be published in Prog. Aerospace Sci., Vol.24, Pergamon Press, 1987.
3. Roe, P.L., "Characteristic-Based Schemes for the Euler Equations", Ann. Rev. Fluid Mech. 18, 1986, pp 337-365.
4. Hall, M.G., "Computational Fluid Dynamics - a Revolutionary Force in Aerodynamics, AIAA Paper 81-1015, 1981.
5. Jupp, J.A., "Interference Aspects of the A310 High Speed Wing Configuration", AGARD CP285, Paper 11, 1980.
6. Back, R.F. and Wedderspoon, J.R., "The A320 Wing - Designing for Commercial Success", Aerospace, January 1986, pp 12-19.
7. Bocci, A.J., "A New Series of Aerofoil Sections Suitable for Aircraft Propellers", Aeronautical Quarterly, Vol.28, 1977, pp 59-73.
8. Bocci, A.J. and Morrison, J.I., "A Review of ARA Research in Propeller Aerodynamic Prediction Methods", AGARD-CP-366, Paper 5, 1984.
9. Smith, J.H.B., "Numerical Solutions for Three-Dimensional Cases - Delta Wings", AGARD-AR-21, Chapter 8, 1985.
10. Doe, R.H., Brown, T.W. and Pagano, A., "The Development of Practical Euler Methods for Aerodynamic Design", ICAS-86-1.4.2, 1986.
11. Marchbank, W.R., "The Integration of Computational Fluid Mechanics into the Military Aircraft Design Process", Proceedings, 58th AGARD/FDP Meeting - "Applications of Computational Fluid Dynamics in Aeronautics", Aix-en-Provence, 1986.
12. Viviani, H., "Numerical Solutions of Two-Dimensional Reference Test Cases", AGARD-AR-21, Chapter 6, 1985.
13. Chima, R.V., Turkel, E. and Schaffer, S., "Comparison of Three Explicit Multigrid Methods for the Euler and Navier-Stokes Equations", AIAA-87-0602, 1987.
14. Hall, M.G., "Cell-Vertex Multigrid Schemes for Solution of the Euler Equations", Numerical Methods for Fluid Dynamics II, edit K.W. Morton and M.J. Baines, Oxford University Press, 1986, pp 303-345.

15. Arthur, M.T., "A Generalisation of Hall's Scheme for Solving the Euler Equations for Two-Dimensional Flows", GMD-Studien Nr.110, Multigrid Methods: Special Topics and Applications, edit. W. Hackbusch and U. Trottenberg, 1986, pp 9-23.

16. Catherall, D., "A Solution-Adaptive Grid Procedure for Transonic Flows around Aerofoils", RAE report in preparation.

17. Salmund, D.J., "A Cell-Vertex Multigrid Scheme for Solution of the Euler Equations for Transonic Flow Past a Wing", Proc. Int. Conf. Num. Meth. Fluid Dynamics 10, Beijing, 1986.

18. Morton, K.W. and Paisley, M.F., "On the Cell-Centre and Cell-Vertex Approaches to the Steady Euler Equations and the Use of Shock Fitting", Proc. Int. Conf. Num. Meth. Fluid Dynamics 10, Beijing, 1986.

19. Roe, P.L., "Error Estimates for Cell-Vertex Solutions of the Compressible Euler Equations", ICASE Report 87-6, 1987.

20. Cross, A.G.T., "Boundary Layer Calculations and Viscous-Inviscid Coupling", ICAS-86-2.4.1, ICAS Proceedings 1986, pp 502-512.

21. Ginder, R.B. and Calvert, W.J., "The Design of an Advanced Civil Fan Rotor", Paper presented at ASME Gas Turbine Conference, 31 May - 4 June 1987.

22. Moore, J. and Moore, J.G., "Calculations of Three-Dimensional Viscous Flow and Wake Development in a Centrifugal Impeller", Trans. ASME Journal of Engineering for Power, Vol. 103, 1981, pp 367-372.

23. Norton, R.J.G., Thompkins, W.T. and Halmes, R., "Implicit Finite Difference Schemes with Non-Simply Connected Grids - A Novel Approach", AIAA-84-0003, 1984.

24. Catherall, D. and Johnson, M., "A Fast Method for Computing Transonic Two-Dimensional Potential Flows Using a Non-Aligned Mesh, RAE TR 87001, 1987.

25. Joyce, Gaynor, "A Method for Calculating Transonic Potential Flow past a Wing-Fuselage Combination Using a Non-Aligned Mesh", ARA Memo 271, 1986.

26. Albone, C.M., "Mesh Generation Employing Curvilinear and Multiply-Embedded Cartesian Meshes", RAE report in preparation.

27. Shaw, J., Forsey, C.R., Weatherill, N.P. and Rose, K.E., "A Block Structured Mesh Generation Technique for Aerodynamic Geometries", Numerical Grid Generation in Computational Fluid Dynamics, edit, J. Häuser and C. Taylor, Proc. Int. Conf., Landshut, July 1986, Pineridge Press, Swansea, pp 329-340.

28. Weatherill, N.P. and Shaw, J.A., "Component Adaptive Grid Generation for Aircraft Configurations", contribution to AGARDograph, ed. J. Thompson and J. Steger, to be published in 1987.

29. Grosch, C.E., "Adapting a Navier-Stokes Code to the ICL-DAP", ICASE Report 85-61, 1985.

30. Hall, M.G. and Salas, M.D., "Comparison of Two Multigrid Methods for the Two-Dimensional Euler Equations", revised version of AIAA Paper 85-1515-CP, submitted for publication.

31. Salas, M.D., Jameson, A. and Melnik, R.E., "A Comparative Study of the Nonuniqueness Problem of the Potential Equation", NASA TP-2385, 1985.

32. Pulliam, T.H. and Barton, J.T., "Euler Computations of AGARD Working Group 07 Airfoil Test Cases", AIAA Paper 85-0018, 1985.

33. Jameson, A., Private Communication, 1985.

34. Klunker, E.B., "Contribution to Methods for Calculating the Flow about Thin Lifting Wings at Transonic Speeds - Analytic Expressions for the Far Field", NASA TN D-6530, 1971.

35. Thompson, J.F., Thomas, F.C. and Mastin, C.W. "Automatic Numerical Generation of Body-Fitted Curvilinear Coordinate System for Field Containing any Number of Arbitrary Two-Dimensional Bodies", J. Comp. Phys., Vol.15, 1974, pp 299-314.

36. Thomas, P.D. and Middlecoff, J.F., "Direct Control of the Grid Point Distribution in Meshes Generated by Elliptic Equations", AIAA J, Vol.18, 1980, pp 652-656.

REPORTS QUOTED ARE NOT NECESSARILY AVAILABLE TO MEMBERS OF THE PUBLIC OR TO COMMERCIAL ORGANISATIONS

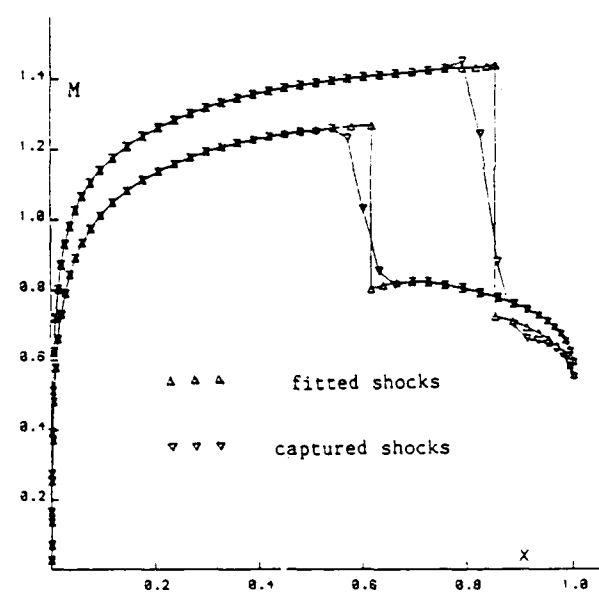
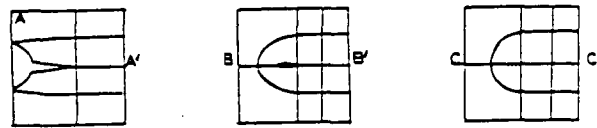
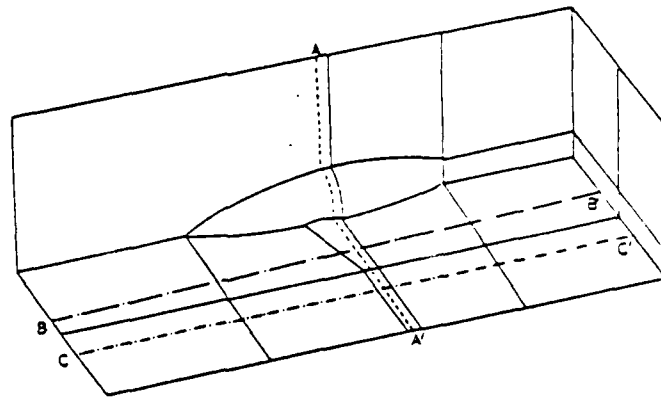


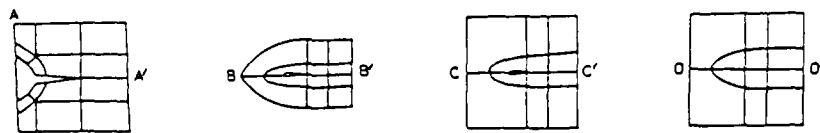
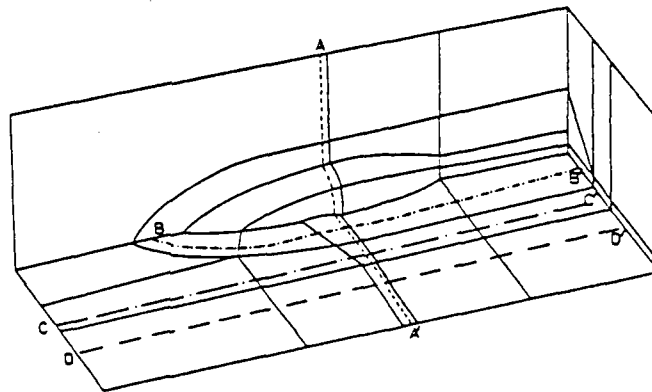
Fig 1 Mach number distributions on NACA 0012 for fitted and captured shocks.  $M_{\infty} = 0.85$ ,  $\alpha = 1.0^\circ$

RAE 2008

Fig 2



Schematic C-H-H topology



Schematic C-H-O topology

Fig 2 Topologies for wing-body combination

Figs 3&4

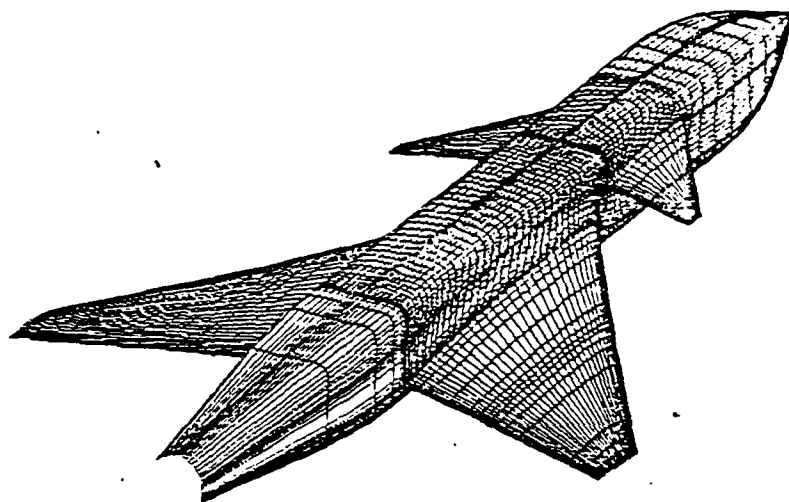


Fig 3 Wing-body-canard. Multiblock surface grid

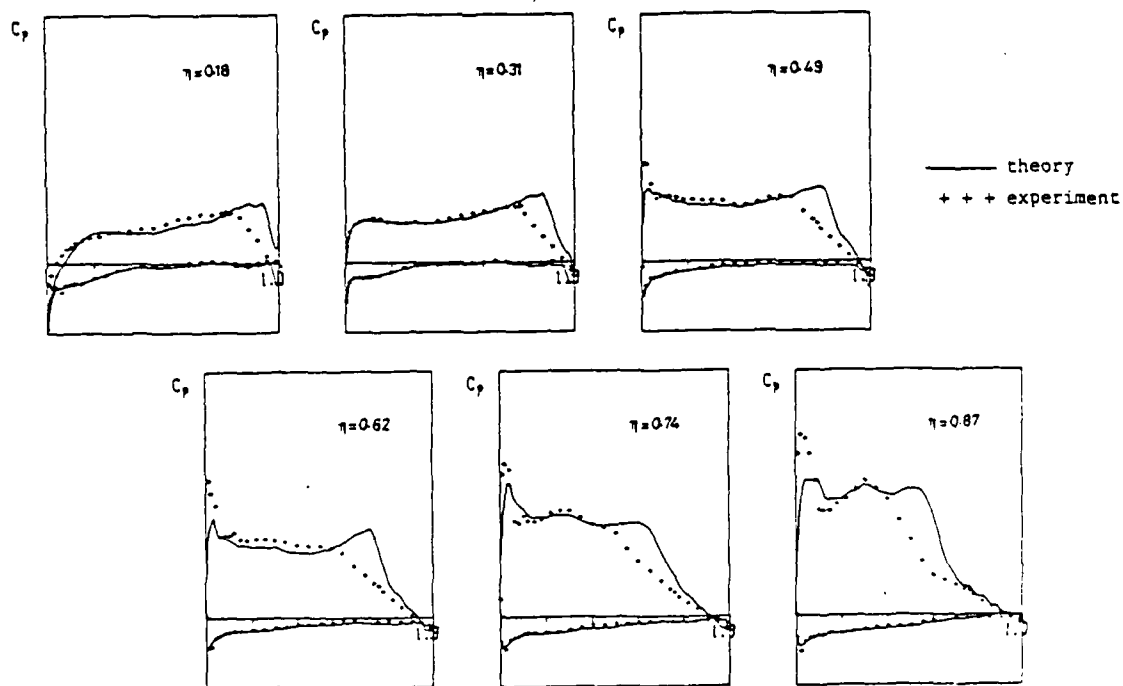


Fig 4 Wing-body-canard. Wing pressures.  $M_\infty = 0.9$ ,  $\alpha = 6^\circ$

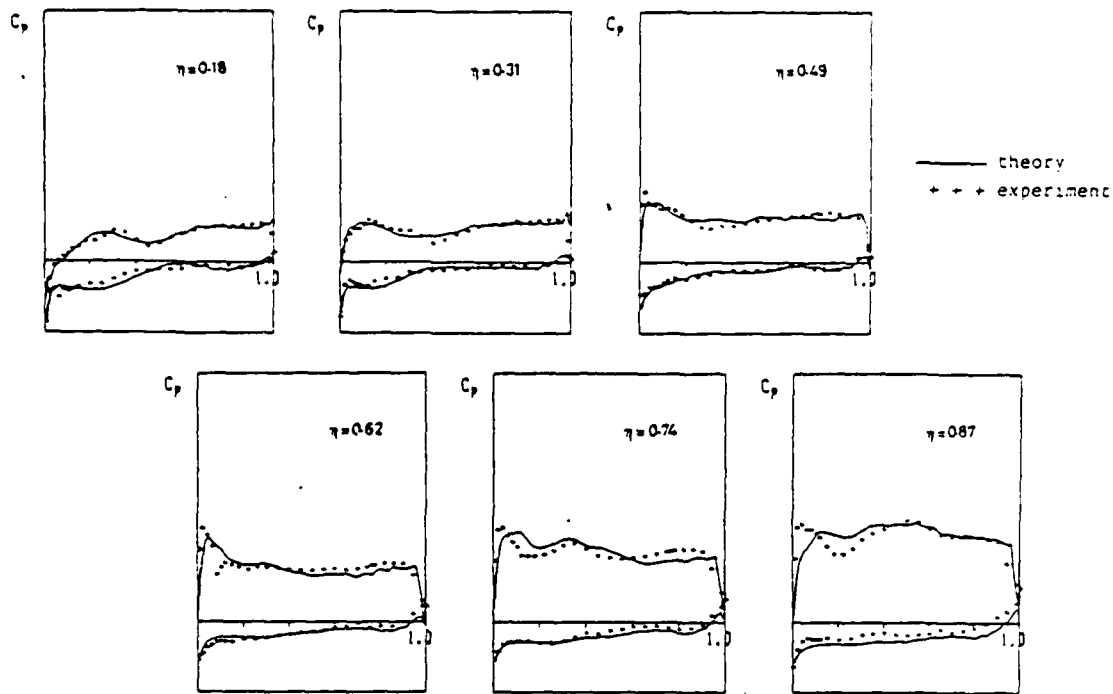


Fig 5 Wing-body-canard. Wing pressures.  $M_\infty = 1.2$ ,  $\alpha = 6^\circ$

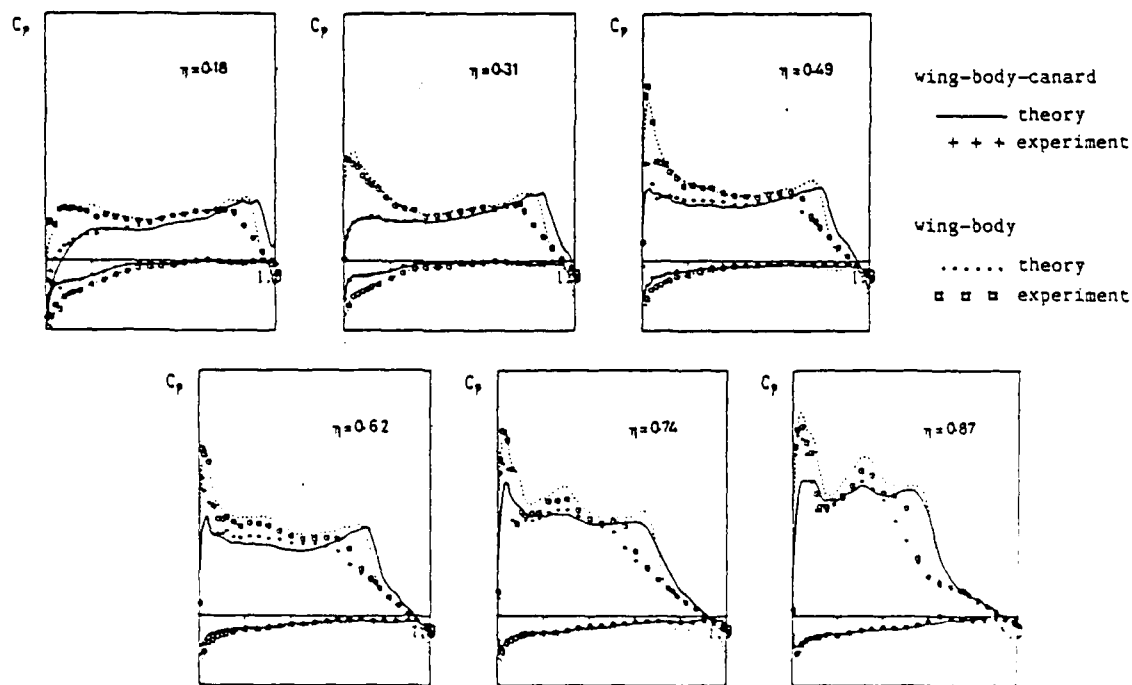


Fig 6 Effect of canard. Wing pressures.  $M_\infty = 0.9$ ,  $\alpha = 6^\circ$



Fig 7

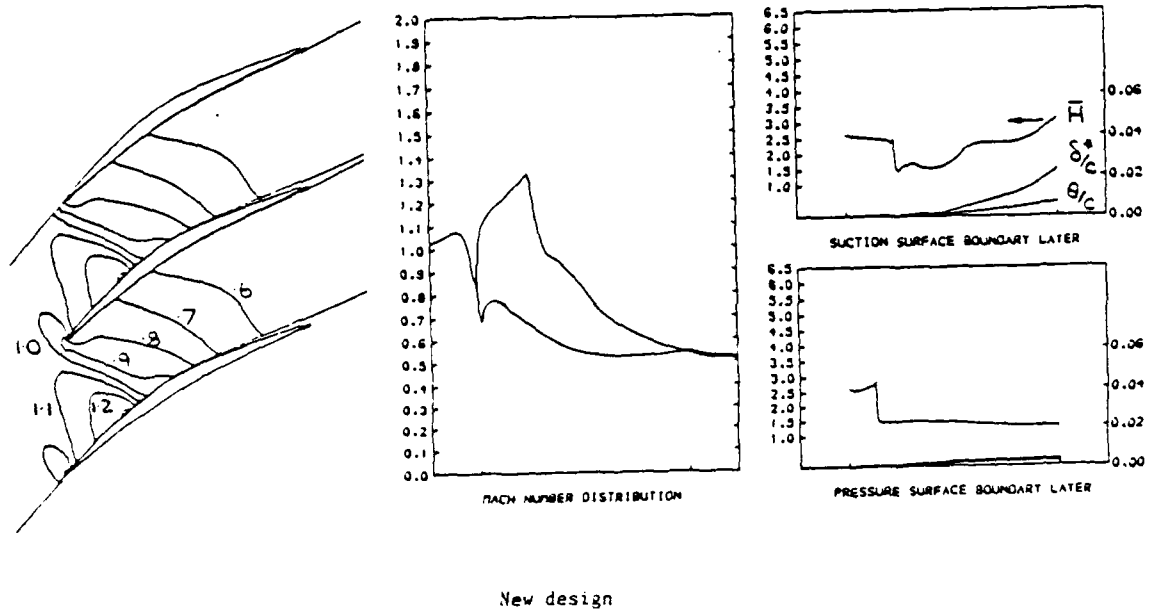
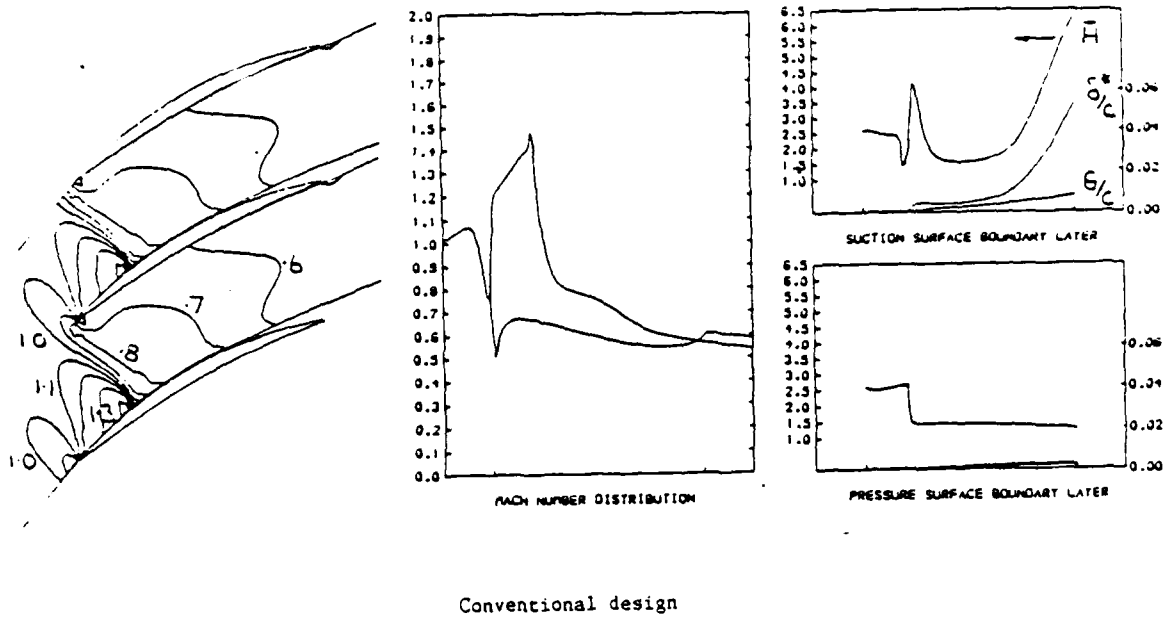


Fig 7 Rotor-fan section. Comparison of conventional and new design

REPORT DOCUMENTATION PAGE

Overall security classification of this page

UNCLASSIFIED

As far as possible this page should contain only unclassified information. If it is necessary to enter classified information, the box above must be marked to indicate the classification, e.g. Restricted, Confidential or Secret.

1. DRIC Reference (to be added by DRIC)	2. Originator's Reference RAE TM Aero 2098	3. Agency Reference	4. Report Security Classification/Marking UNCLASSIFIED
5. DRIC Code for Originator 7673000W		6. Originator (Corporate Author) Name and Location Royal Aircraft Establishment, Farnborough, Hants, UK	
5a. Sponsoring Agency's Code N/A		6a. Sponsoring Agency (Contract Authority) Name and Location N/A	
7. Title Computational fluid dynamics in the United Kingdom			
7a. (For Translations) Title in Foreign Language			
7b. (For Conference Papers) Title, Place and Date of Conference AIAA 8th Computational Fluid Dynamics Conference, Honolulu, 9-11 June 1987			
8. Author 1. Surname, Initials Hall, M.G.	9a. Author 2 Fiddes, S.P.	9b. Authors 3, 4 ....	10. Date Pages Refs. April 14 36 1987
11. Contract Number	12. Period	13. Project	14. Other Reference Nos.
15. Distribution statement (a) Controlled by -  (b) Special limitations (if any) - No distribution before 9 June 1987. If it is intended that a copy of this document shall be released overseas refer to RAE Leaflet No.3 to Supplement 6 of MOD Manual 4.			
16. Descriptors (Keywords) (Descriptors marked * are selected from TEST)			
17. Abstract A review is presented of computational methods in aerodynamic research and design, with application of the methods and associated computing facilities included. The review begins with a brief survey of the field, to give an overall view and to identify what seem to be notable features. These are then described in turn. The first is a development by Hall and his colleagues of accurate and fast schemes for solving the Euler equations, based on the finite-volume cell-vertex methods introduced by Denton and Ni. Next is a development of a block-structured ('multiblock') grid generation technique by the Aircraft Research Association. There follows an application of this grid generation scheme, in conjunction with an Euler solver, to the calculation of the transonic flow past a wing-body-canard configuration. Finally, an application of viscous-inviscid interaction techniques to the design of a fan rotor is described.			

F5910/1

END

10-87

DTIC