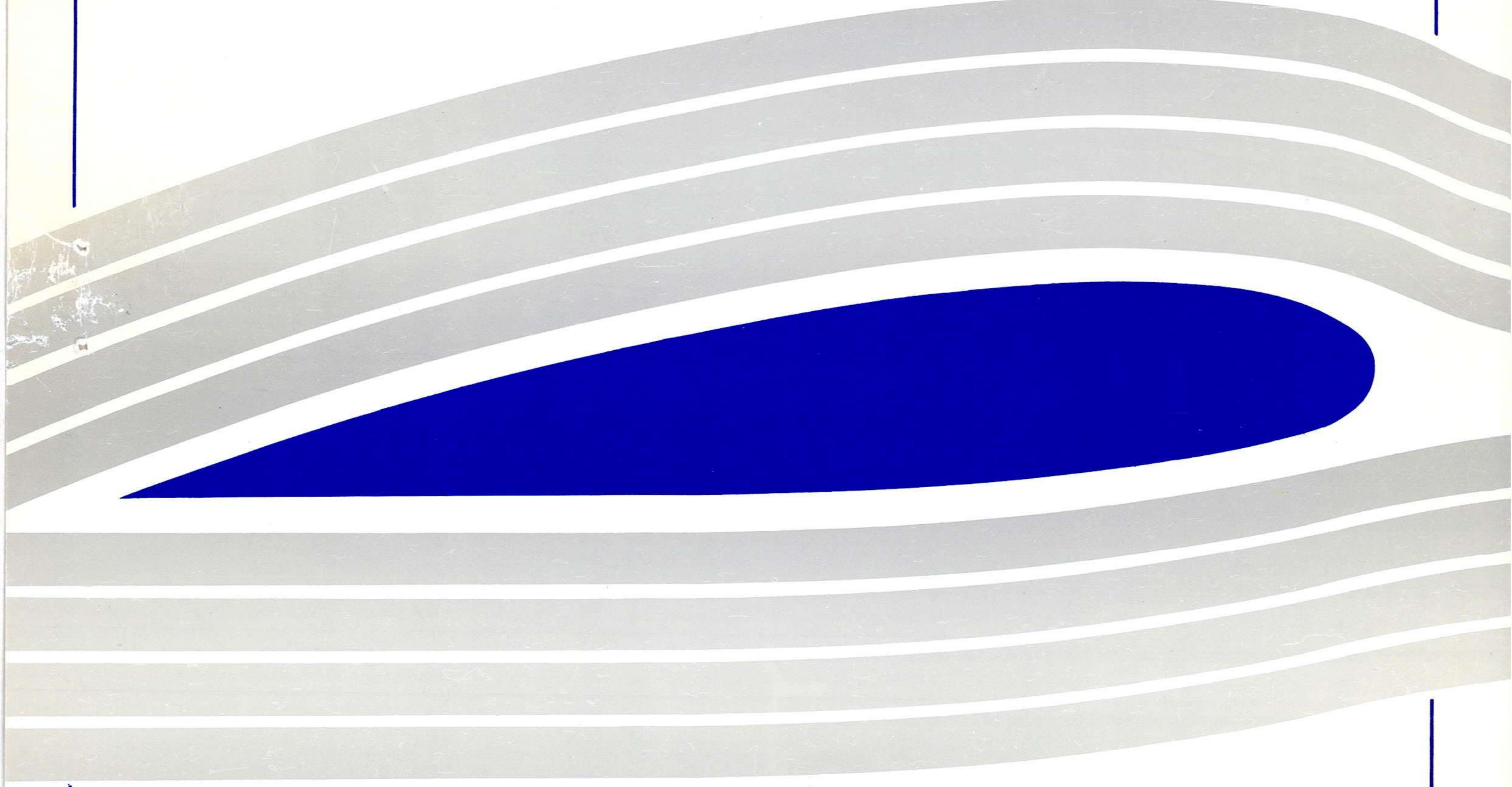
University of Glasgow
DEPARTMENT OF
AEROSPACE
ENGINEERING

# Development of Viscous Techniques for Application to Weapons Configuration Throughout the Flight Regime

## A Review

B.E. RICHARDS and N. QIN



# Development of Viscous Techniques for Application to Weapons Configuration Throughout the Flight Regime

## A Review

B.E. RICHARDS and N. QIN

G.U. Aero Report 9119

Dept. of Aerospace Engineering
University of Glasgow
GLASGOW
G12 8QQ

## Abstract

This report provides a review of current technology on the prediction of viscous flows using numerical techniques for applications to weapons configurations. The work is aimed at aiding the preparation of a strategy for advising a capability for predicting the aerodynamics of weapon configuration and to eventually bring such a strategy to fruition. Such a strategy would have to involve a long term research and development programme.

## 1. Introduction

The flow around weapons configurations at incidence inevitable involves significant viscous effects in the form of body/lifting surface separation and resulting vortex fields. The tools available to designers to predict the flows over complete configurations involve panel methods, small disturbance equations, potential flow methods and solution of the Euler equations which solve the inviscid flow field. However, flight at incidence involves extensive regions of viscous flow which changes significantly the well behaved flow fields which occur at lower angles of incidence. The way forward is inevitably through solutions of the appropriate Navier-Stokes equations.

The Navier-Stokes approach has been slow to be adopted by designers because of the large amount of computing power and storage required. However, rapid progress in computer hardware and computational techniques is encouraging the development of suitable codes. It is appropriate now to develop the relevant Navier-Stokes technology to keep pace in this industry.

Over the past eight years the group at Glasgow have developed Navier-Stokes codes for the prediction of transonic, supersonic and hypersonic external flows over vehicle components. The work has involved mainly the extension of Euler solvers designed to capture shocks with high resolution, to produce solutions to the Navier-Stokes equations where the interest is particularly focused on heating, aerothermodynamics and separation on transonic aerofoils. Schemes explained, and in most cases implemented, include central difference schemes (e.g. McCormack explicit and predictor corrector method, Jameson Runge-Kutta approach) and upwind scheme (e.g. flux vector splitting schemes of Steger/Warming and van Leer, TVD scheme such as Harten, Yee and flux difference splitting schemes such as Roe and Osher). Experience has been accumulated in the application of various acceleration schemes such as multigrid and local time stepping. A new method of rapid convergence has been developed for CFD codes called the sparse quasi-Newton technique, which has been found to provide an extremely valuable tool when the very sensitive parameters of heat transfer and skin friction need to be predicted. This experience is now in the process of being applied to parabolised Navier-Stokes technology and some preliminary validation has already been achieved in the Tracy cone case.

Computers presently used in this work include the University's IBM3090 150E VF and the national SERC facilities of the ULCC CRAY XMP 12 and the RAL CRAY XMP 48 and IBM 3090 600. Our experience to date has been that programs developed here have ported easily between these computers and that, except in the early stages, there has been no difficulties in usage of these remote facilities. Some codes have been vectorised with speed-up rates of 5 achieved. At present preliminary usage of an IBM RS/6000 325 workstation, Intel Hypercube (with 16 Intel i860 processors), a Parsytec 64 node T800 transputer system, SUN Sparc station and Convex machines are being tested and/or used to test Navier-Stokes codes.

In this report, a preliminary survey of CFD in weapon aerodynamics has been made. One of the authors has been intermittently involved with weapon aerodynamics, firstly at VKI, Belgium up to 1980, involved with experimental work and the organising of lecture series, culminating in the organising and editing of the AGARD LS 98. Apart from the supervision of a SERC CASE student in collaboration with British Aerospace Dynamics on vortex breakdown, CFD work at Glasgow has been associated with aircraft or spacecraft aerodynamics and aerothermodynamics. Part of the present review was associated with familiarisation of the present scope of the topic. Since it was perceived that CFD in weapons aerodynamics lags behind that in aircraft, then an element of the review concerns the translation of the more advanced material into the weapons context.

## 2. Features of weapon configuration which bear on their aerodynamics.

There exists a number of reviews which outline the differences in missile or weapon aerodynamics from conventional aerodynamics. These include Brebner (Ref. 1), various chapters from Hemsch and Nielsen (Ref, 2) and Lacun and Robert (Ref. 3).

Brebner explains these features as:

" 1. The objectives of the design of flying vehicles are usually quite different.
2. The range of attitudes and flight conditions are quite different.
3. The shapes which are appropriate to achieve the design objectives in the required flight conditions can be very different, and since applied aerodynamics usually require a number of assumptions and approximations to be made in order to obtain practically useful results, the assumptions and approximations appropriate for typical weapons may not be appropriate for aircraft shapes."

He defined the field of air flight weapons as "the aerodynamics of narrow, low-aspect ratio, slender wing-body-control combinations." More fully he listed these characteristics as contributions to the problem:

- flow around inclined bodies
- effect of the complete range of roll angle on monoplane and cruciform configuration
- boundary layers, especially on bodies
- base pressures and base drag with a very hot jet efflux present
- effects of vortices on downstream components
- effect of nose shape on drag and drag due to excrescences and protuberances
- kinetic heating effect
- incidence range from 0 - $180^0$
- Mach number range from moderate subsonic to high supersonic
- static and dynamic stability derivatives

and these characteristics need be explored on unconventional shapes.

More recently Lacun and Robert explored the new problems facing the aerodynamicist which arise because of the increasing complexity of configurations as follows:

<u>Unconventional Shapes</u>:-

- bodies with side intakes. Here problems involve external aerodynamics of the intake; the influence of the flow field around the fuselage on the location of the intake, and the internal aerodynamics.

## 2. Features of weapon configuration which bear on their aerodynamics.

There exists a number of reviews which outline the differences in missile or weapon aerodynamics from conventional aerodynamics. These include Brebner (Ref. 1), various chapters from Hemsch and Nielsen (Ref, 2) and Lacun and Robert (Ref. 3).

Brebner explains these features as:

> "1. The objectives of the design of flying vehicles are usually quite different.
> 2. The range of attitudes and flight conditions are quite different.
> 3. The shapes which are appropriate to achieve the design objectives in the required flight conditions can be very different, and since applied aerodynamics usually require a number of assumptions and approximations to be made in order to obtain practically useful results, the assumptions and approximations appropriate for typical weapons may not be appropriate for aircraft shapes."

He defined the field of air flight weapons as "the aerodynamics of narrow, low-aspect ratio, slender wing-body-control combinations." More fully he listed these characteristics as contributions to the problem:

- flow around inclined bodies
- effect of the complete range of roll angle on monoplane and cruciform configuration
- boundary layers, especially on bodies
- base pressures and base drag with a very hot jet efflux present
- effects of vortices on downstream components
- effect of nose shape on drag and drag due to excrescences and protuberances
- kinetic heating effect
- incidence range from 0 - $180^0$
- Mach number range from moderate subsonic to high supersonic
- static and dynamic stability derivatives

and these characteristics need be explored on unconventional shapes.

More recently Lacun and Robert explored the new problems facing the aerodynamicist which arise because of the increasing complexity of configurations as follows:

<u>Unconventional Shapes</u>:-

- bodies with side intakes. Here problems involve external aerodynamics of the intake; the influence of the flow field around the fuselage on the location of the intake, and the internal aerodynamics.

- non-circular bodies: subsonic modular stand-off missiles with square or rectangular cross sections; supersonic/hypersonic air breathing missiles with elliptical or triangular etc. cross sections.

### Stealth Considerations

Missiles with relatively long flight times need low radar and infra-red signatures. For the aerodynamicist these involve avoidance of sharp angles, surface irregularities, straight leading edges; suppression of reflective surfaces and corner reflectors. This means: the design of smooth profiles for lifting surfaces and fuselage; body-wing junction fairing; elliptic fuselages swept and curved leading and trailing edges; smooth and shaped leading edged duct intakes; subsonic intakes flush with fuselage; non-orthogonal lifting surfaces.

### Lateral Jet Control Thrusters

- rocket powered force control, acting at centre of gravity
- aerodynamic interactions due to lateral jet.

### Very High Angles of Attack

- achieved for vertically launched low altitude missiles after launch
- achieved during rapid manoeuvre

### Non Rigid Airframes

Shape changes due to flexibility of long missiles constructed from composite materials, and also aeroelastic problems with $^{L}/_{D} \geq 20$ using conventional materials, under air loads.

Many of the key elements of those features involve major viscous effects whose features can only be expected to be predicted by calculating the global flow field combining both inviscid and viscous modelling. The resulting flow field is very complex and any simulation , experimental or numerical , with realism is bound to be a costly process.

## 3. The place of wind tunnel techniques.

Wind tunnel techniques play a role in weapon aerodynamics in a number of ways, two important areas of which are:-

- the measurement of overall forces and moments on configurations
- provision of an understanding of flow phenomena and physics in order to assist the

modelling or empiricism used in the prediction process.

An area of strategic importance at present is the development of high spatial resolution experimental data bases involving surface and field measurements of simple and complex shapes to assist the validation of CFD codes. It is a feature of such codes that high resolution data is produced, and because of its well ordered nature, such data can be very convincing but could be in error if, for example, the initial equation and boundary condition modelling, the modelling of the physics or the discretisation approximation used are in error.

The wind tunnel approach for weapon configuration modelling has advantages and disadvantages. Advantages include the ability to test the most complex of configurations although cost could impose on the running of a large matrix of changes in configurations. Apart from the high costs that arise from full simulation of flight conditions, other important disadvantages include:

- interference effects from supports that are needed to keep the model in the flow - crucial aerodynamics involves base flows and the leeside of high incidence flows which could be affected by the support, unless a type of sting is used to minimise the problem.
- many flow field measurement techniques intrude into the flow, and can affect flows, particularly those with large viscous regions
- simulation of propulsion systems and control jet flows
- the testing in hypersonic and transonic flows. In the former case high energy levels are required to provide the full M ~ Re ~ enthalpy simulation desired and the tunnel flows produced can be unrepresentative (e.g.not in thermal or chemical equilibrium) or difficult to diagnose. In transonic simulation it is difficult to obtain high Reynolds number data without tunnel interference effects. These happen to be two flow areas where the mathematical modelling is highly non-linear, such that high order models and hence numerical techniques are required.

Experimentalists are however making rapid progress in meeting present demands with, for example, the use of, where appropriate, magnetic suspensions, optical non-intrusive techniques (LDV, particle tracking techniques) use of miniature propulsion units, short duration facilities with fast response instrumentation, cryogenic wind tunnels etc. However, even with cheap powerful computing facilities, modern manufacturing techniques, data acquisition and processing methods, such experimental techniques come at great expense, and the intensifying competition in the technology sector gives little room for manoeuvre in deploying such techniques.

The most cost effective strategy for progress appears to be in pooling CFD and experimental techniques, in an iterative way, in which experiments provide validation data to provide the confidence in the computer codes, and the codes provide indicators for where the crucial data is required.

## 4. Review of present predictive schemes.

The main tools used at present for missile design from the aerodynamics viewpoint, as viewed from two recent reviews (Ref. 2, and 4), are semi-empirical models, panel methods and Euler solvers. The latter have been extended by using models that identify separation lines and include separation models.

The ESDU and DATCOM data sheets provide information mainly on isolated wing, body and component parts; overall characteristics can be obtained by adding components and correcting for interference effects. They contribute to semi-empirical models.

S-HABP (Ref. 5, 6) involves the approximation of the surface of a configuration by many flat surfaces, called elements, at incidence. Analytical and empirical formulae are used to calculate the effect on these elements. It includes both inviscid and viscous contributions to the flows.

ABACUS (Ref. 7), also breaks down a configuration into component parts but uses a variety of results from data sheets, panel methods and Euler codes on idealised shapes as inputs again along with corrections for interference effects. NUFA (Ref. 8) is an extension of this in dividing the body into many segments before applying the techniques used in ABACUS. It enables calculations to be carried out in non-linear flows such as induced in stores carriage and separation or propulsion integration.

Panel methods or more descriptively linear potential schemes are now well integrated in design in aerospace vehicles. These are well suited to complex configurations and to computational aerodynamics and the present power of computer hardware is such that calculations are economic and can be turned round very quickly. A present review of such schemes is given in Ref. 9, 10. Nowadays between 5,000 and 10,000 panels each side of the symmetry plane are used in the aircraft industry to represent say a transport aircraft in a landing configuration. The calculations are inviscid only, linearisation means that the calculations are only reasonably valid for fully subsonic or supersonic cases. Typically each design organisation has developed their own system relevant to their application. Named methods used in missile applications include USSAERO and NLR derivatives of this, PANAIR, HISSS and LRCDM2.

Incorporating non-linear effects in inviscid flow fields as typified by transonic flow with short waves was first achieved by Murman & Cole (Ref. 11). This has led to the widespread development of methods for calculating transonic potential flows in two and three dimensions using either the transonic-small-disturbance equation (as in the original paper) or the full-potential-flow equation. This also provided the breakthrough for efforts to devise efficient algorithms for solving the non-linear Euler and Navier-Stokes equations, and some of the shock capturing ideas and convergence acceleration schemes (such as approximate factorisation of the difference operator and acceleration by the use of multiple grids) that were first developed for potential-flow calculations have proved transferable to models such as the Euler equations. Jameson's codes FL027, FL028 and FL030 are the most used in the industry to tackle modelling at this level of approximation. The three dimensional codes have involved finite element approaches. Unstructured grids are undoubtably the most suited to complex configurations. A useful review of these as well as Euler and Navier-Stokes solvers (to be reviewed) is given in Ref. 12.

Considerable attention has been given to Euler solvers. It has generally been expedient to use the time dependent equations as a vehicle for obtaining steady state solutions mainly since they provide a rational framework for the design of non-oscillating shock capturing schemes which reflect the physics of wave propagation. In the space discretisation part of the process then the shock capturing process may require added dissipation in order to prevent oscillations. A variety of dissipative and upwind schemes based on the work of Godunov have evolved into schemes with good shock capturing capabilities. Time stepping is achieved typically using explicit one step multistage methods resulting in solutions of systems of ordinary differential equations using Runge-Kutta schemes. This approach has been particularly effective for transonic flows. Implicit approaches of various forms have been developed and used in hypersonic flows.

In the application to missile flows, an example of an Euler solver is that given by Priolo and Wardlaw (Ref. 13). The code ZEUS is a space marching solver incorporating a multiple zone gridding technique and a second order extension of Godunov's method. It includes a separation model which assumes a vortex sheet leaving the surface at an experimentally observed separation point. Fairly good agreement with body alone experiments at $M = 2.5 - 4.5$, $\alpha = 16^0$ were achieved with improvement over inviscid flow alone calculations by including the separation model. Encouraging results were achieved on finned bodies. The content of the paper emphasises the complexity of the flows even for these simple cases. Trials of several Euler and a parabolised Navier-Stokes code developed in U.S. and Canadian laboratories is given by Jones et al (Ref. 14).

The majority of CFD developments towards the production of industrial codes to solve applied aerodynamic problems have focussed on aircraft or spacecraft applications. These applications usually involve very high unit costs with long project lifetimes (circa 20 years) in which aerodynamic optimisation is important. A useful compilation of the state of the art in the aircraft sector, with emphasis on the transonic speed range, is publicised in a recent book (Ref. 15). Optimisation of the aerodynamics for weapons aerodynamics is now important because of the accuracy and effectiveness of weapons systems in use, due to improved systems design and this accumulation of technology and experience from the airframe side is useful to explore.

The rapid progress of CFD in providing industry tools is well illustrated in Ref. 15. There exist promising validations of Navier-Stokes codes as well as codes with lower order models and demonstrations of the calculation of flows of considerable complexity. At the same time it is recognised that many of the demonstrations of the highest order models are for set piece (or show piece) cases and are not examined over a broad range of situations. It is recognised that before CFD can seriously be taken up by management, there is a need for critical demonstrations of repeated successes. But still the time-frame for achieving these successes is too long. The next chapters outline an opinion given from experience of the Glasgow Group as to the improvements that may be achieved in numerical developments to match the rapid improvement in computing hardware to achieve these demonstrations and some of our attempts to achieve them.

## 5. Required Developments to make Navier-Stokes Solvers respectable to managers.

The main general requirements to bring Navier-Stokes solvers into the design domain are as follows.

- Calculations on complex configurations can be achieved in short time spans.
- The resulting calculations are accurate and are capable of simulating the main features of the flow.
- The codes are robust and can handle a reasonably wide range of applications.
- The code can be used by individuals or teams who are not necessarily CFD or computer science specialists.

There is sufficient evidence that these requirements could be achieved eventually (practitioners in industry indicate that a decade maybe realistic), however they will come about only through continued developments and the accumulation of considerable experience.

### 5.1 Use of Computer Hardware.

Access to adequate fast computing power with large memory is an important immediate requirement. Progress in Navier-Stokes codes development can only be made if fast turn round times are achieved. Tests of these high order models inevitably are highly computer intensive. Such machines have been available to developers recently, but usually these are corporate or national main-frames. This results in competition with other users, with usually the intensive computing practitioner (for whom the concurrent machines were provided in the first instance) losing out to users with modest requirements in the priority list. Low cost workstations (IBM RS/6000, SUN Sparcstation, etc.) which can be dedicated to the work are now available to provide for useful Navier-Stokes solver testing. Management however needs to provide the finance for these facilities (and explore distributed workstation systems instead of expensive multi-user mainframes with huge overheads) for both the developer and user. Limitations on memory size (64MB physical memory) can still limit speed when operating in double precision even though the virtual memory size is infinite.

For the designer, with complex configurations in mind and, with speed and cost of importance, the most likely contender for achievement of his requirements is the distributed memory multi-processor. A review of novel architecture applicable to CFD is given in Ref. 16. At present there exist fine, medium or course grain parallel machines. Fine grain computers are well applicable to research into such areas as full simulation of viscous (turbulent) flow and for application of panel methods to complex situations. However course to medium grain are considered more applicable to Navier-Stokes solvers. The most popular usage of parallel computing at present is for domain-decomposition techniques. Some experience in these techniques in low speed Navier-Stokes solvers in the Department (Ref. 17) has demonstrated diminishing returns for not much more than 16 processors using this approach. The favoured approach then is to use the powerful Intel i860 processor involved in such machines as the Meiko computing surface (with MK086 boards) and the Intel Hypercube. This is the thinking behind the use of the Meiko machine in the SERC/MoD proposal, the case for which is given in Appendix III. Incidentally such a machine can be used as a multi-user machine when used in a network, and since the i860 processors have potentially a speed up to 80 MFlops (with vectorisation), such a machine can be additionally useful for development work.

Looking further to the future Taylor (Ref, 18) envisages that optimum matching of CFD algorithms with computer hardware will be achieved. Silicon wafers would be manufactured to provide algorithm equivalents instead of use of software producing orders of magnitude faster processing time.

## 5.2 Physics Modelling

Acceptance of codes for design purposes is only given if realistic modelling of physics is included. In weapon aerodynamics, the size, speeds and attitudes involved will certainly include turbulence, and since many of the flows of interest involve separated flows then high order turbulence modelling on a Reynolds Averaged Navier-Stokes (RANS) framework will be necessary. In some cases transitional flow will be predominant, and since in the foreseeable future there is likely to be little progress on full simulation of Navier-Stokes equations in transitional flow for configurational applications, we will have to rely on correlations from experimental studies. The means of incorporating these correlations into the codes will be important. Most of the work published is involved with the inclusion of turbulence models in boundary layer, triple deck or simple shear flows. Relevant ongoing research needed is the inclusion of these models into Navier-Stokes solvers, in which there exist no preconditions as to where the viscous regions exist. The Glasgow group is supported by SERC/MoD to explore the fundamentals of this topic towards implementation for complex configurations.

For high speed applications high enthalpy flows may be generated. Again in this topic there is a basic knowledge of the physics involved, but the applications of this have been mainly in inviscid flow or boundary layer applications. The relevant interest is again in the incorporation of this knowledge into a Navier-Stokes solver scheme. Again support from SERC/MoD to Glasgow allows progress on the fundamentals of this topic.

In both these cases there exist widely varying scales in the phenomena concerned. In turbulence modelling spatial scaling is important and high gradients of various parameters are encountered. In the modelling of high temperature flows large variations in timescale can occur which have an effect on the convergence of the iterative procedures used. The direction of code development then is not only towards the inclusion of the models but also in devising the strategy towards obtaining fast and robust convergence schemes for complex configurations in which there is no prior knowledge of the nature of whereabouts of these high gradients. This again illustrates the need for sufficient computing power for testing purposes.

## 5.3 Grid Generation for complex configuration.

The usefulness of CFD techniques to designers becomes apparent only if they are able to treat extremely complex configurations leading to complete aircraft or weapons. A major pacing item to attain this goal is the problem of mesh generation, associated with the space discretisation of the problem. Realisation of this has sparked considerable interest and the application to external aerodynamic applications has been recently reviewed in Ref. 19.

A major choice is between structured or unstructured grids. For complex configurations unstructured grids are easier to generate, but connectivity relationships between the grid elements are required resulting in a high storage. Also the numerical flow calculations are more complex and time consuming. Unstructured grids were a rational development arising from the use of the finite element method in fluids, but more recently finite volumes methods, based on conservative formulations of the flow equations, and hence applicable to use on arbitrary element shapes, have proved possible. Numerical computations using unstructured grids do not use well the possible beneficial effects of parallel computation. Structured meshes are not difficult to generate for relatively simple shapes, and because of the simple geometrical relationships of elements, then code development is simpler. The early favoured technique has been the finite difference approach but finite volume methods using conservation forms of the flow equations have proved more useful when shock waves are present and in which conservation of properties could be problematical.

Another choice to be made is whether the mesh should be single block or multiple blocks. Multi-block techniques for complex configurations appear to be attractive. The chief problems concern the connectivity between individual blocks and the way that the calculations are treated at the boundaries. Experience to date is that a considerable amount of experience needs to be acquired by an operator to deal with the topology, or initial choice of blocks, for a particular configuration. Code development is also complex, but the approach has potential in dealing with multi-domain and hence domain decomposition techniques in parallel computers.

The probable optimum technique is likely to be a mixture of the two. There are good reasons to have a structured grid close to the surfaces of the complex geometry where viscous effects, and high gradients occur. Away from these surfaces then unstructured grids may be more attractive in allowing a single block mesh to be obtained.

Grid adaptation or grid enrichment provides a means of controlling the mesh size, so that dense meshes are provided where high gradients are encountered and vice-versa. Schemes have been developed where this procedure is done automatically. It is important, however, that control is kept on this process, since the further refined nature of sharp gradients such as caused by shock waves may not necessarily improve the accuracy of the final data sought by the designer and could unnecessarily increase the computer cost.

At present, structured grids on a single block framework is used at Glasgow. Grids of reasonable complexity for PNS codes are achievable from a body which can be defined as y = o planes along a given axis, which could be produced from a solid modeller. Having defined an outer limit of the field to be calculated then an algebraic technique is used to complete the mesh in successive planes. Alternatively for 3-D Navier-Stokes codes a transfinite interpolation algebraic grid generator is used to form shapes of reasonably realistic complexity.

## 5.4 Linear Algebra Solvers.

Generally the implicit approaches used result in the solution of large linear sparse non-symmetric systems. In the work at Glasgow this happens from schemes with or without the use of the sparse quasi-Newton method (see 5.6). The majority of the time of calculation in a code is taken up with these linear algebra solvers, and improvement of this process is important. This area is being considered at Glasgow using the GMRES (generalised minimum residue) algorithm devised by Perriaux's group at AMD-BA, France. The main benefits come from preconditioning. Even more benefit would arise from the introduction of a parallelised version of this.

### 5.5 Acceleration Schemes.

Multi-grid time stepping techniques have been shown to considerably improve the convergence rate of explicit schemes (Ref. 20). The motivation of multi-grid techniques was to propagate waves more efficiently on course grids whilst maintaining fine grid accuracy. The idea of Ni's (Ref. 20) multi-grid scheme is to use course grids to propagate fine grid corrections properly and efficiently throughout the field thus improving convergence rate to steady state while maintaining low truncation errors. A more direct application of the method is the application of the full approximation scheme (FAS) to accelerate time marching schemes (Ref. 21). These methods are the focus of attention for acceleration techniques and their success with regard to Navier-Stokes codes is problem dependent. There are many factors in multi-grid procedures e.g. cycling strategy, the level of grids and the number of iterations on each grid which can influence the convergence such that extensive numerical experiments are necessary if an optimum is to be found. The technique of multi-grid acceleration is not confined to explicit schemes, and Qin (Ref. 22) outlined two multi-grid schemes which improve convergence to provide efficient implicit schemes.

### 5.6 Sparse Quasi-Newton Technique.

The numerical solution of Navier-Stokes equations is strongly dependent on the cell Reynolds numbers. This necessitates the use of a very fine grid to resolve high gradients near the wall. To achieve the high resolution necessary it is needed to employ discretisation schemes which can capture not only strong shocks but also the shear layers as well as stretching the grid near the wall. The combination of high order upwind schemes (van Leer's FVS, Roe's or Ocher's FDS and Harten's or Yee's TVD schemes) have created problems in the Navier-Stokes equation solutions. The efficiency of multi-grid techniques have been found to be significantly reduced on highly stretched grids. Furthermore, implicit procedures involve linearisation which if exact implies analytical calculations of the Jacobian on the non-linear system which affects Navier-Stokes solutions using high order schemes. Approximate linearisation commonly used in practice generally results in poor convergence for Navier-Stokes solutions because of the unbalanced left and right hand sides. It is for this reason that the team at Glasgow embarked on the sparse quasi-Newton approach to avoid the difficulties in the linearisation needed in implicit schemes and the achievement of fast convergence to steady state.

The basic idea of the quasi-Newton method (Ref. 23) is to approximate the Jacobian of the non-linear system using only function values already calculated. When solving sparse systems however, using this idea, the inversion of the sparse matrix is not sparse and hence the need for special treatment of the sparse system. The final procedure depends only on the sparcity of the Jacobian and the solution from the previous iteration. An initial time marching scheme is thus employed to achieve preliminary convergence. The calculation can then be switched to the sparse quasi-Newton method which can provide quadratic convergence properties. The rapid local convergence of the scheme is thus combined with the robustness of a time marching scheme to obtain a globally convergent procedure. The simplicity and generality of the procedure suggests its use as an efficient tool for fast steady state solutions in CFD.

The method has been proved for Euler and Navier-Stokes solutions for simple geometries (Ref. 24) but needs developing and proving for cases with more complex physics and configurations.

## References.

1. BREBNER, G.G. “A brief review of air flight weapons” and “General missile aerodynamics” AGARD-LS-98. B.E. Richards (ed.) February, 1979.

2. HEMSCH, M.J. and Nielsen, J.N. (ed.) “Tactical Missile Aerodynamics” Progress in Astronautics and Aeronautics, AIAA, 1986.

3. LACUN, R.G. and Robert, M. “Some trends in missile aerodynamics” Paper 1 in “Missile Aerodynamics” AGARD-CP-493, April 1990.

4. BIGNELL, P.R. (ed.) “Missile Aerodynamics” AGARD-CP-493, April, 1990.

5. GENTRY, A.E. et al. “The Mark IV supersonic/hypersonic arbitrary body program” Vols. I-III, AFFDL-TR-73-159, November, 1973.

6. FISHER, C.M.E. “Experiences using the Mark IV supersonic/hypersonic arbitrary body program.” Paper 31, AGARD Conference on Aerodynamics of Hypersonic Lifting Vehicles, Bristol, April, 1987.

7. HERRING, P.G.C. “A computer program which evaluates the longitudinal characteristics of typical weapon configurations” AGARD-CP-336, paper 26, Missile Aerodynamics, September, 1982.

8. McDOUGALL, S., Press, A.J. and Barratt, P.S. "NUFA: a semi-empirical method for the prediction of isolated weapon aerodynamics" AGARD-CP-493, April, 1990.

9. DILLENIUS, M.F.E. and Allen, J.M. "Paneling methods with vorticity effects and corrections for non-linear compressibility" from "Tactical Missile Aerodynamics" Progress in Astronautics and Aeronautics, Hemsch, M.J. and Nielsen, J.N. (ed.) Vol. 104, AIAA, 1986.

10. HESS, J.L. "Linear Potential Schemes" Ch. 2 from "Applied Computational Aerodynamics" Progress in Astronautics and Aeronautics, Vol. 125, AIAA, 1990.

11. MURMAN, E.M. and Cole, J.D. "Calculation of plane steady transonic flows" AIAA Journal, Vol. 9, No.1, 1971, pp. 114-121.

12. JAMESON, A. "Full potential, Euler and Navier-Stokes schemes" Ch. 3 in "Applied Computational Aerodynamics" Progress in Astronautics and Aeronautics, Vol. 125, AIAA, 1990.

13. PRIOLO, F.J. and Wardlaw, A.B. "Supersonic tactical missile computations using Euler's equations with crossflow separation modelling" Paper 6 in "Missile Aerodynamics", AGARD-CP-493, April, 1990.

14. JONES, D.J., Evans, J., Priolo. F.and Sterek, W. "Application of Euler and Navier-Stokes codes to missile type bodies with high L/D ratios" Paper 8 in "Missile Aerodynamics", AGARD-CP-493, April, 1990.

15. HENNE, P.A. "Applied computational aerodynamics " Progress in Astronautics and Aeronautics, Vol. 125, AIAA, 1990.

16. BORIS, J.P. "New directions in computational fluid dynamics" in Annual Review Fluid Mechanics, Vol. 21, 1989.

17. SHU, C. "Generalised differential integral quadrature and application to the simulation of incompressible viscous flows including parallel computation" Ph.D. dissertation, University of Glasgow, April, 1991.

18. TAYLOR, T.D. "Computer trend for CFD" Twelfth International Conference on Numerical Methods in Fluid Dynamics, Oxford, 1990. Lecture Notes in Physics, Vol. 371 (K.W. Morten, Ed.)

19. STEINBRENNER, J.P. and Andersen, D.A. "Grid generation methodology in applied aerodynamics" Ch. 4 from "Applied Computational Aerodynamics" Progress in Astronautics and Aeronautics, Vol. 125, AIAA, 1990.

20. NI, R.H. "A multiple grid scheme for solving Euler equations" AIAA Journal, Vol. 20, 1982, pp. 1565-1571.

21. JAMESON, A. "Solution of the Euler equations by a multi-grid method" Applied Math. and Computation, Vol. 13, 1983, pp. 327-356.

22. QIN, N. "Towards numerical simulation of hypersonic flow around spaceplane shapes" Ph.D. Thesis, University of Glasgow 1987.

23. QIN, N. and Richards, B.E. "Sparse quasi-Newton method for Navier-Stokes solutions" presented at 8th GAMM Conference on Numerical Methods, 1989 Notes in Numerical fluid Mechanics, Vieweg, Braunschweig,1990.

24. QIN, N. and Richards, B.E. "A parabolised Navier-Stokes code combining high resolution with fast convergence" AGCFM Workshop, September, 1990.

BER/efg.