NUMERICAL EXPERIMENTS USING NAVIER STOKES CODES FOR GENERALISED HYPERSONIC SHAPES

Ning QIN, Zhijian WANG, Bryan RICHARDS.

Department of Aerospace Engineering,
UNIVERSITY OF GLASGOW,
GLASGOW, G12 8QQ, U.K.

Abstract

In this paper numerical experiments are carried out to model the hypersonic flow over generalised shapes representing a spaceplane during re-entry. The modelling uses solutions of the locally conical reduced set of Navier-Stokes (N-S) equations as well as the full 3-d N-S equations. A variety of high resolution schemes such as flux vector splitting, flux difference splitting and total variational diminishing have been explored as well as some novel acceleration techniques. Results are presented of the shock interaction in a corner and the high Reynolds number flows over a blunt delta wing and a body with canopy at 30 degrees angle of attack representing shapes being explored under the HERMES programme. The excellent comparisons with experiment demonstrate the value of developments of this nature bearing in mind the difficulty in generating quality experimental results in the hypersonic flow regime.

Introduction

The subject Computational Fluid Dynamics has assumed an important role in the studies of fluid dynamics and aerodynamics. It has become prominent because advances in computing hardware and numerical methods have enabled realistic modelling of applied engineering systems to be accomplished. Such examples in the Aerospace field include aircraft wings, as on the Airbus family of aircraft transport, by British Aerospace and the Royal Aerospace Establishment. With further progress in the subject this technology will be indispensable to designers. One area where CFD is particularly required as a tool is the area of trans-atmospheric flight, in which at some point in the mission information on the aero-thermodynamics of complex shapes is required because of the high speeds sustained during the atmospheric stages of flight. At one stage such high temperatures are achieved that chemical reactions occur in the flow. At another stage high Reynolds numbers are achieved such that the flow is turbulent. Depending on the trajectory, both of these physical phenomena could coincide. Experimental testing in wind tunnels is a difficult and expensive way to generate sufficient well resolved data, because of the high energy levels required to provide the test flow, the problems of integrity of materials of the tunnel and models at these high temperatures, the difficulty in flow diagnostics and also the availability of satisfactory instrumentation with the necessary resolution and accuracy. Numerical techniques, duly validated so that confidence in their application can be ensured, provides an alternative way forward.

In numerical techniques, following hypersonic small disturbance approaches used in the last generation of flight vehicles, preliminary advances have been made in the solution of the Euler equations on complex vehicle shapes. This approach allows prediction of the inviscid flow around these shapes, which combined with three dimensional boundary layer calculations, has proven adequate to deal with the well behaved flow over atmospheric vehicles. However trans atmospheric craft need to slow down during re-entry into the atmosphere at high angles of attack and thus the flow is dominated by large separated regions and shock and viscous interactions. Such flow cases can only be effectively tackled by solutions of the Navier Stokes (NS) equations which, except for very high altitudes when molecular theories must be applied, have been shown to model the flow adequately. The barrier at the moment, however, lies in solving the N-S equations including the relevant physics on the computer hardware expected to be available in the near future. As in the past history of aviation design, compromises have to be sought to come up with satisfactory solutions.

At high Reynolds numbers, and high heat transfer rates, apart from the abrupt gradients associated with shock waves, there will exist very high gradients in shear layers, which have to be well resolved in order to define accurately skin friction and heat transfer. Success in capturing such gradients depends critically on the choice of mesh size. The number of points used has an important bearing on the length of computation. Savings can thus be made if the fineness of the mesh can be chosen to suit the local region under calculation. This can be done by preselection, or automatically during the computer run using adaptive mesh techniques.

Modelling turbulence correctly using full or large eddy simulation is unrealistic at present on all but the simplest configurations because of the need for fine meshes to incorporate all eddy scales. Thus this problem is being tackled at present in a statistical way using the Reynolds averaged Navier Stokes Equations, but with models of increasing complexity with greater aspirations as to success in prediction. The most used techniques include the Baldwin-Lomax eddy viscosity approach which, although essentially an equilibrium model, has met with moderate success when "calibrated" in dealing with moderate separated flows. Two equation, Reynold's stress models such as K-e models and the Johnson-King hybrid technique have been applied to tackle the non-equilibrium effects. These problems exist as well as the fact that, because of the energetic nature of this viscous flow, the gradients at the wall are considerably higher than for the laminar counterparts with the resultant impact on mesh size.

In dealing with high temperature gas effects, the flight time of particles in the calculation zone of the vehicle are so short that the chemistry effects associated with the heating through shock waves and layers and the cooling in flow expansion areas are not in equilibrium. Because of the wide variety of the chemical reaction rates, the resulting flow equations become "stiff" to solution. These effects can be overcome by using assumptions such as: limiting the number of reactions to those which are important; by assuming flow fully in equilibrium or fully frozen in appropriate positions; or using a fixed but lower value of the ratio of specific heats as in perfect gas calculations.

A popular approach in N-S solvers is to use the time dependent solution as an iterative scheme to attain steady state. Although the introduction of implicit techniques, which improve mesh size limitations of explicit methods through the CFL condition, has improved the efficiency of this approach, nevertheless run times can be excessively large when dealing with more complex configurations in which a large number of mesh points are required. For cases in which separated flows have a streamwise orientation, then a parabolised version of the N-S equations in which those viscous terms that involve derivatives in the streamwise direction are dropped. Then the stationary form of the N-S equations can be solved by an explicit procedure in the x-direction and an appropriate calculation procedure in the cross planes. This significantly reduces the mesh requirements for calculation and thus on the overall calculation time. For special cases, such as flow over cones, delta wings or some corner flows, when the flow is essentially conical, then considerable savings can be achieved by using the locally conical N-S (LCNS) equations. This can provide a valuable proving ground for testing discretisation technique for complex viscous flows in an economic fashion.

Further ways of reducing computation time lie in the use of various techniques that accelerate the convergence of the computational method. These very often involve a relaxation of time accuracy during the iterative procedure, which can be acceptable when only steady state solutions exist. An example is that of the use of local time steps, which differ from one point to another as a function of local propagation speeds corresponding to the local CFL number. Multigrid methods propagate information more efficiently by using a sequence of grids from coarse to fine. Another method under development for CFD applications at Glasgow is the sparse quasi-Newton method. Such an approach provides the quadratic convergence of a Newton method by approximating the Jacobian of the highly nonlinear system, resulting from tackling complex configurations, by making use of function values already calculated. The technique makes use of the sparsity of the matrix generated. The technique needs the use of discretisation techniques which are continuously differentiable of which Osher's flux difference splitting method is appropriate. The work is explained in [1] and [2].

All of these problems in which the focus has been on N-S solvers are being tackled at Glasgow University in a continuing programme of research started in 1984. Work on laminar perfect gas calculations on flows with strong interactions in corner flows, delta wings and a forebody with canopy at high angles of attack are outlined in the main body of this paper. Other work includes: sensitivity studies on discretisation techniques and grid sizes [3]; exploration of TVD schemes for use in viscous flows [4]; turbulence modelling in transonic and hypersonic flows using Baldwin-Lomax and Johnson-King models [5, 6]; inclusion of equilibrium real gas effects in a multiblock Euler code [7]. Studies are also ongoing on the exploration of a discretisation technique to reduce the number of grid points [8]. Experience is being gained on parallel computer architectures, in recognition of the further reduction of costs using this approach [9]. Experience in parabolised Navier Stokes technology has been obtained in the prediction of flow over sharp and blunted cones at high angle of attack [10, 11]. Some effort has been placed in the development of flow visualisation approaches, since interpretation of the fluid physics can only be achieved through compacting the vast amount of data obtained from these multi-dimensional CFD codes [12].

The LCNS Code and Corner Flow

In this section, we describe the locally conical Navier-Stokes code developed for simulating high speed flows around conical geometries of aeronautical interest and also for testing the capabilities of different numerical schemes in capturing both shock waves and shear layers.

The Governing Equations

The locally conical Navier-Stokes equations are obtained from the 3D Navier-Stokes equations subject to a locally conical approximation. The approximation assumes that the variation of the flow is much smaller in the conical direction than that in the crossflow surface so that derivatives of the flow properties in the conical direction can be neglected. This reduces the computation of a full 3D flow field to 2D crossflow surfaces. The LCNS equations are a reasonable model for high speed flows around shapes such as cones, delta wings and combinations of these which generate flows of locally conical nature.

Discretization Schemes

In order to capture both the strong shock waves and the shear layers for accurate prediction of the flowfield and, therefore, accurate prediction of the heat transfer rate distributions, the choice of numerical schemes becomes an important factor in the computer simulation. It is well-known that the current sophisticated upwind schemes (e.g. flux vector splitting, flux difference splitting and TVD schemes) behave quite well in Euler solution in capturing shock waves without adding explicitly an artificial viscosity term. But their behaviour for Navier-Stoke solution in modelling viscous shear layers is still uncertain as most of the schemes were developed for inviscid shock wave modelling. This necessitates a systematic test of these schemes for accurate Navier-Stokes solutions.

The following three discretization schemes have been studied:-

(1) MacCormack's central differencing scheme;
(2) Van Leer's flux vector differencing scheme;
(3) Osher's flux difference splitting scheme.

An adaptive artificial viscosity term is used in the central differencing scheme. Details of these schemes can be found in [3] or the original papers.

The Corner Flow Problem

A severe test case including strong hypersonic shock wave/boundary layer interaction is provided by a Mach 12.76 hypersonic flow in a 90 degree corner formed by 30 degree swept back intersecting 8 degree wedges. The geometry is shown in Fig.1. The model length is 100 mm and the measurements were taken at a distance of 90 mm from the tip of the model. The numerical simulation was based on the following flow conditions to match the experiment [14].

$M_\infty = 12.76, \quad Re_\infty / m = 5.0 \times 10^6$
$T_\infty = 38.73, \quad T_w = 300\,K$

Results and Discussion

Fig.1. also shows the 65x65 grid in the cross section of the corner, which is clustered towards the wall to resolve the high gradiant in the viscous layer.

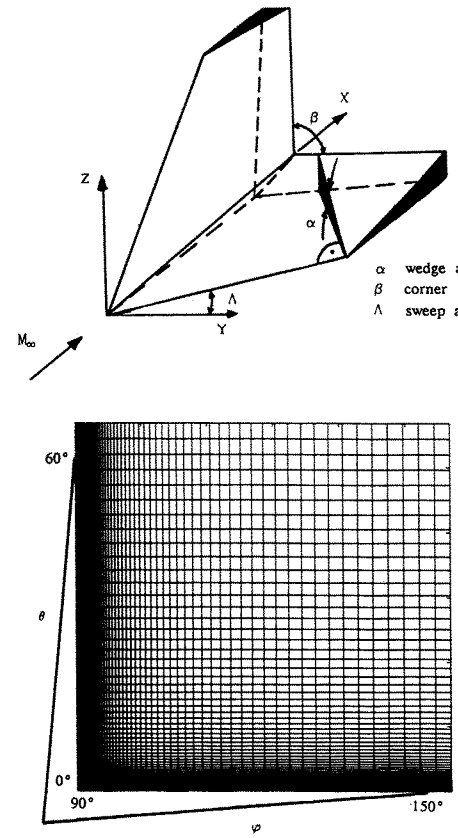

Fig.1. The swept corner geometry and the grid

Fig.2. compares the computed heat transfer distributions with the experimental data. It is clear that the flux vector splitting scheme seriously underpredicted the heat transfer peak while the central differencing scheme and the flux difference splitting scheme gives reasonablly good prediction. The flux difference scheme gives the best comparison with the experimental data. Fig.3. plots flowfield velocity vectors. The vortices in the corner region are clearly seen. Fig.4. gives a comparison of pitot pressure contours and an interpretation of the flow field from the experiment and the computation.

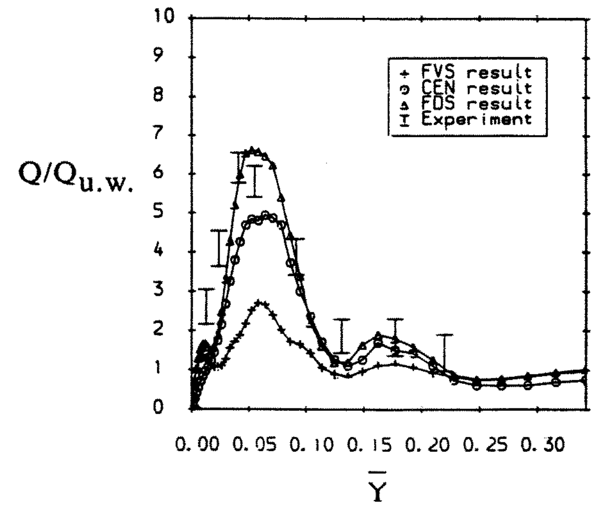

Fig.2. Comparison of heat transfer

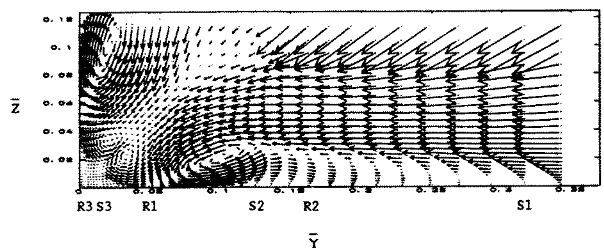

Fig.3. Velocity vectors

The 3D Finite Volume High Resolution NS Code and Delta Wing Flow

Based on the study of the previous section with the LCNS code, a 3D finite volume high resolution Navier-Stokes code has been developed for solution of high speed flows.

The Governing Equations

The starting point is the 3D Navier-Stokes equations written in integral form. A general coordinate transformation is adopted which transform the Cartesian coordinates into general coordinates. As discussed in the last section, Osher's flux difference splitting scheme exhibits high resolution behaviour for both strong shock waves and shear layers and so is appropriate for hypersonic Navier-Stokes solution. This scheme is used in the spatial discretization in the 3D NS code.

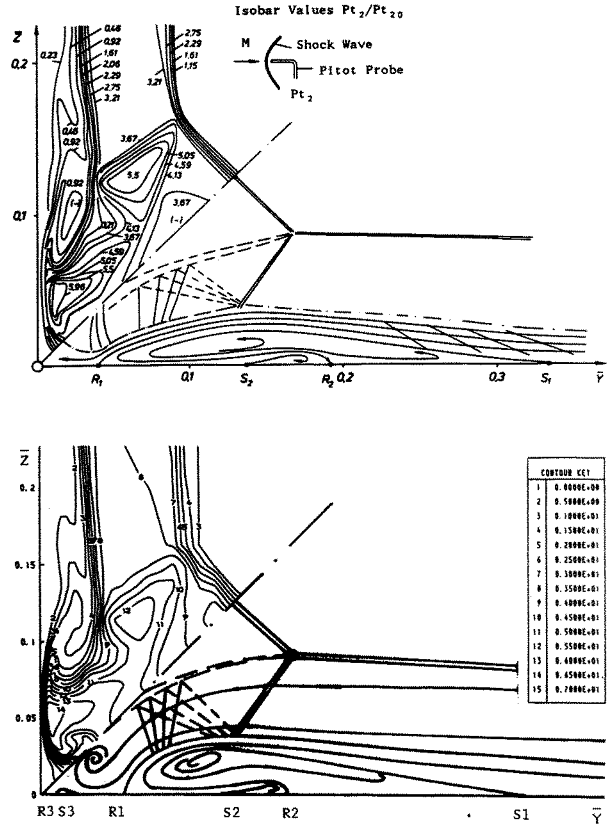

Fig.4. Measured and computed pitot pressure and interpretations

The Delta Wing Problem

Simulations of hypersonic flows around blunt cones, ogive shapes and delta wings have been carried out using the present code. Here we describe its applicaton in the simulation of a hypersonic flow around a delta wing with blunt leading edges. This is one of the bench mark cases for the Workshop on Hypersonic Flows for Re-entry Problems held in Antibes, France. The conditions for the case is:-

$$M_\infty = 7.15, \qquad Re_\infty / m = 3.9 \times 10^7,$$
$$T_\infty = 74\,K, \qquad T_\infty = 288\,K.$$

The angle of attack is 30 degrees.

Grid Generation

A C-O grid is generated using a transfinite interpolation technique. The grid is clustered towards the wall and is required to be orthogonal there to resolve the viscous layer and to simplify the treatment of the boundary conditions respectively. The grid is generated in the following order:

(1) Generation of the 1D line grids at all the junctions of the domain surfaces. The grid is stretched according to the curve length.

(2) Generation of the 2D surface grids at all the domain boundary surfaces using the 1D line grids as the interpolation boundaries. A defective Hermite interpolation is employed when orthogonality at the wall is required.

(3) Generation of the 3D field grid by interpolation of the surface grids. The orthogonality of the grid at the wall is achieved by specifing the derivative information at the wall in the defective Hermite interpolation. The cluster of the grid at wall is controlled by either the magnitude of the derivative or the stretch factor in the interpolation.

Results and Discussion

The computation was carried out on an IBM 3090-150E using a 33x35x75 grid as shown in Fig.5.

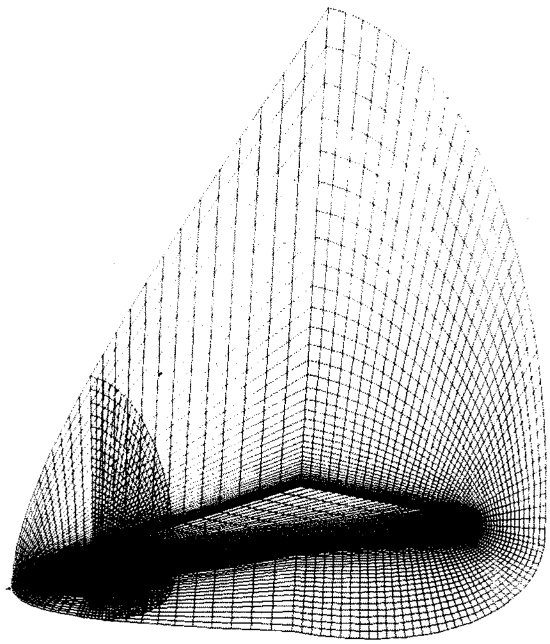

Fig.5. The C-O grid for the delta wing

Fig.6. shows the Mach number contours at the 80% cross flow plane, where the increment is 0.5. The strong shock wave is clearly seen on the windward side. Also seen in the contour plot are the

embedded shock and shear layer on the leeward side indicating strong viscous interaction flow over the wing.

Fig.6. Mach contours at 80% crossflow plane

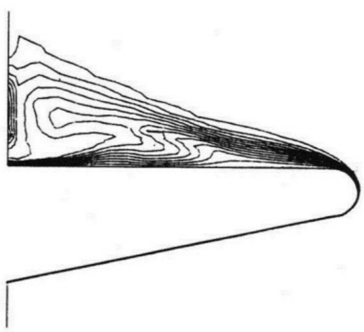

Fig.7. Contours of total temperature loss at 80% crossflow plane

In Fig.7, the total temperature loss contours is plotted at the 80% cross flow plane. This picture gives a clear indication of the viscous layer around the wing. The fact that there is no contour line outside the shear layer demonstrates the low level of nonphysical dissipation and, therefore, the accuracy of the simulation. The boundary layer is very thin on the windward side but relatively thick on the leeward side where it is separated from the wall and rotates into a vortical flow. This vortical flow can be seen more clearly in the crossflow velocity vector plot in Fig.8.

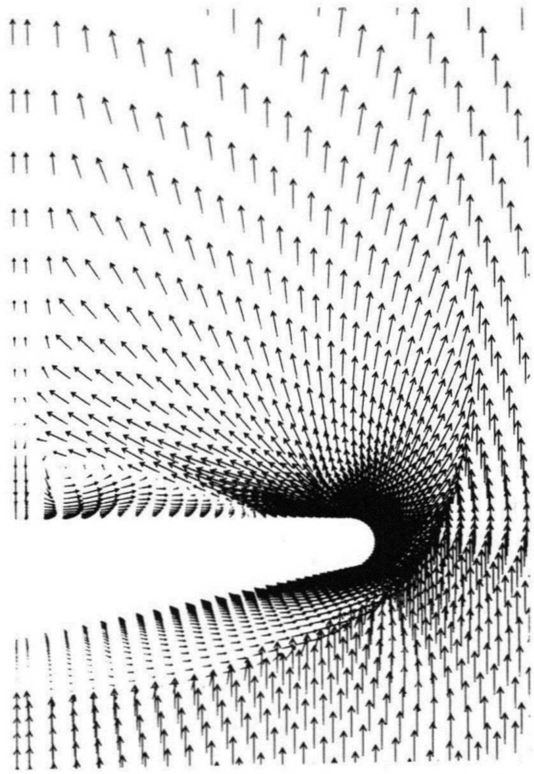

Fig.8. Velocity vectors at 80% crossflow plane

The skin friction lines are shown in Fig.9, which crisply marks the primary separation line, i.e. the first white converged line away from the leading edge. Also noticable is the second white converged band, which indicates a nearly separated secondary vortical flow.

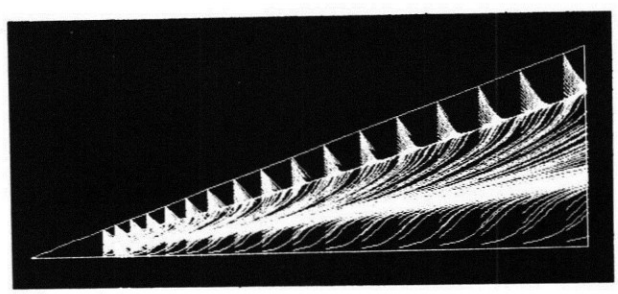

Fig.9. Skin friction lines on the upper surface of the delta wing

The heat transfer rate is shown in Fig.10. Highest heating is observed on the windward side near the leading edge. The variation of the heat transfer on the leeward side is relatively small. Fig.11. shows the pressure distribution on the wall at the 80% crossflow plane.

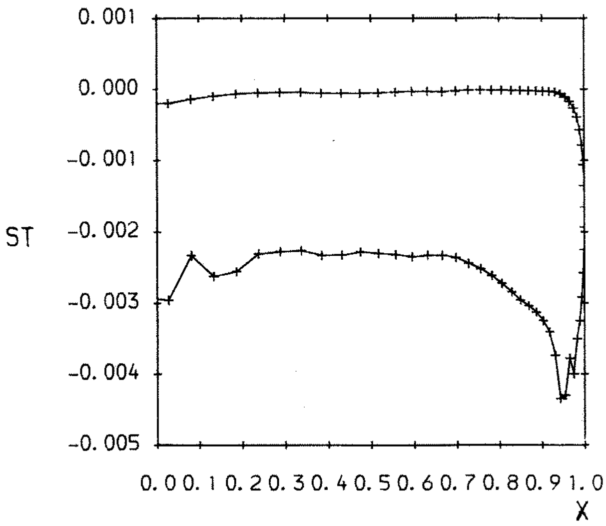

Fig.10. Heat transfer on the wall at 80%

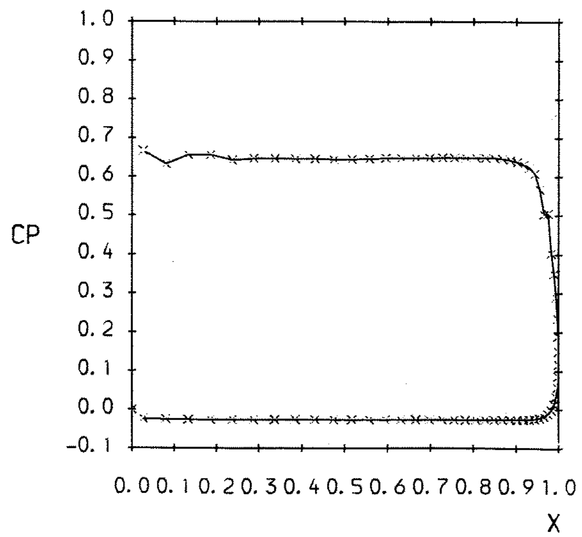

Fig.11. Pressure distribution at 80%

A 3D Code Using High Resolution TVD Schemes

Introduction

For the past few years, the modern shock-capturing Total Variation Diminishing schemes have gained favour in the CFD community. These schemes have the property of avoiding spurious oscillations near sharp gradients and thus have proved to be very robust for hypersonic problems with strong shock waves. Another advantage of these schemes is that no numerical "artificial viscosity coefficient" needs to be introduced. Their applications in the solution of inviscid flow have achieved some excellent results. The applications in the simulation of viscous flow have been studied in [8]. Two prominent TVD schemes, the Osher-Chakravarthy and Yee symmetric TVD scheme, have been investigated. It was found that blind application of TVD schemes to viscous flow can sometimes cause unrepresentative results. Therefore, modifications have been made to make these schemes more suitable for viscous flow calculations. Numerical experiments did show that the modified schemes gave better results than the original schemes. Then these schemes have been coded to a general-purpose three-dimensional flow solver VNS3D. This code has since been vectorized and validated . In this paper, the code is used to simulate hypersonic viscous flow around a double ellipsoid representing the forebody of a space vehicle including a canopy.

Governing Equations and Numerical Approach

For continuum compressible flow, the appropriate models to use for inviscid and viscous flow are the Euler or Navier-Stokes equations respectively. In realistic flow simulation, a body-fitted curvilinear coordinate system is sought because of its ease in dealing with boundary conditions. Therefore the governing equations are written in conservation form using general coordinates. This form is particularly convenient for finite difference discretization. As the current research represents the first step towards the simulation of realistic hypersonic flow, the assumption has been made that the flow is laminar and thermally and calorically perfect. Sutherland's formula is employed to calculate the coefficient of viscosity.

The computational discretization technique must be reliable and robust if is to capture the complex physics of viscous hypersonic flow fields. The TVD discretization technique for convection terms combined with Roe's approximate Riemann solver for the flux formula offers an excellent procedure for that purpose. The viscous terms are discretized by means of the well-behaved central differences. Then conventional approximate factorization methods are used to solve the three-dimensional implicit formulations. Several techniques are employed to speed up the convergence to the steady-state. These include the local time stepping, grid sequencing procedure, vectorization of the code, etc. It has been found that these techniques greatly accelerate the convergence rate.

Results and Discussions

The flow solver code described previously was validated through various computations testing its shock and viscous layer capturing capabilities [4]. Then it was applied to calculate a hypersonic viscous flow around a double-ellipsoid at a high angle of attack. Experimentation has been carried out to serve as a validation procedure [13].

The geometry together with the computational mesh is displayed in Fig.12. The grid is generated

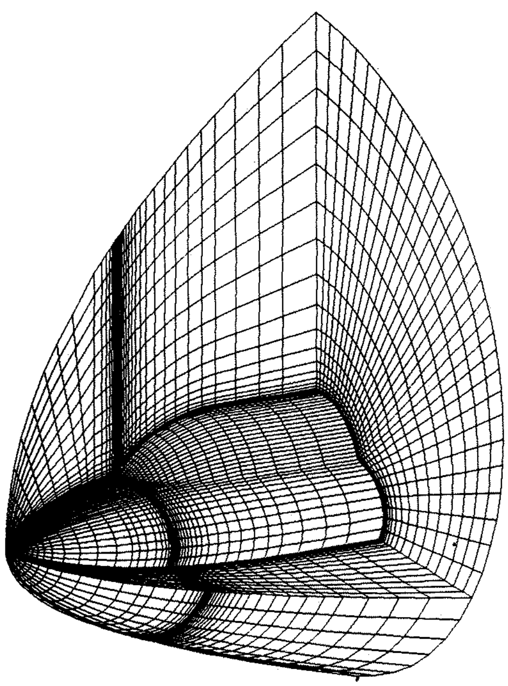

Fig. 12 The geometry and mesh of the double ellipsoid

using an algebraic method which includes 49, 41 and 25 points in the mainstream, circumferential and wall-normal directions respectively. The mesh is clustered near the wall for high resolution of boundary layers.

The flow conditions for this case are as follows. The free-stream Mach number is 8.15, angle of attack is 30 degrees, the free stream temperature is 56K, the wall temperature is 288K and the Reynolds number is 1670000.

The Mach number contour is displayed in Fig. 13 on the plane of symmetry, the body surface and the cross section plane. It is obvious that there is a steep boundary layer near the surface. The shock wave is well captured with at most one intermediate grid point in the windward direction. The oil flow pattern on the body surface is presented in Fig. 14b. The experimental oil flow pattern is shown in Fig. 14a. It is found that the position of the computed separation line agrees quite well with the experimental separation line.

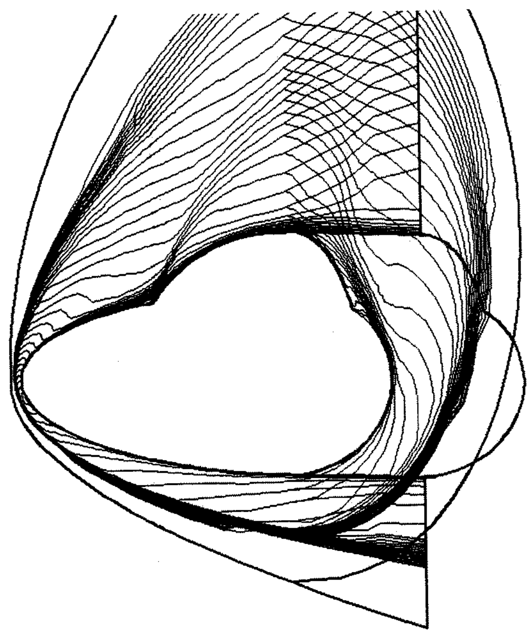

Fig. 13 Mach contour

Experimental data are available for the pressure coefficient and Stanton number on the wall in selected planes. Hence more detailed comparisons are now made between the calculation and the experimentation. Fig. 15 shows the Cp on the plane of symmetry. It is seen that the numerical results are

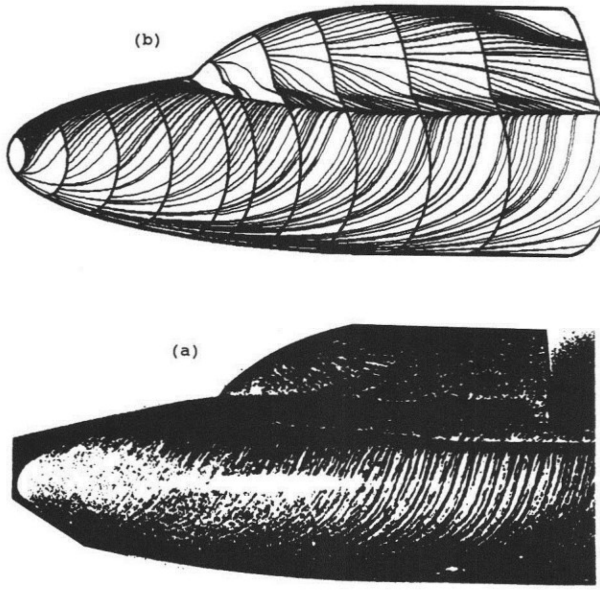

Fig. 14 Oil flow pattern a) experiment, b) calculation

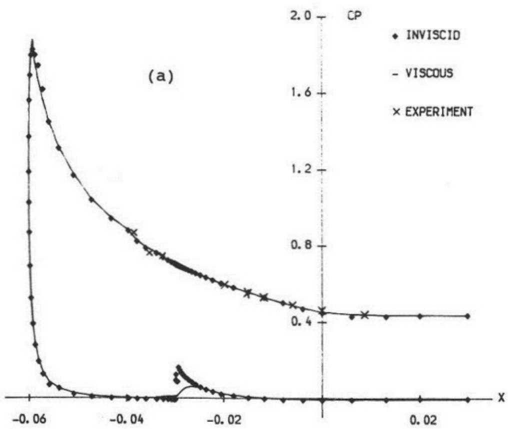

Fig. 15 Cp on the plane of symmetry

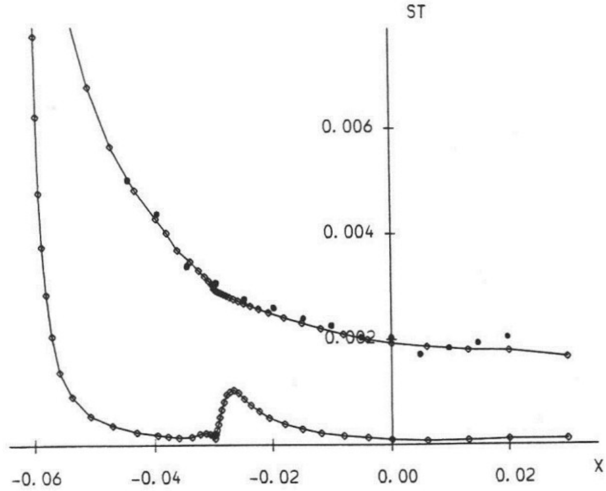

Fig. 16 St number on the plane of symmetry (• experiment, -∘- calculation)

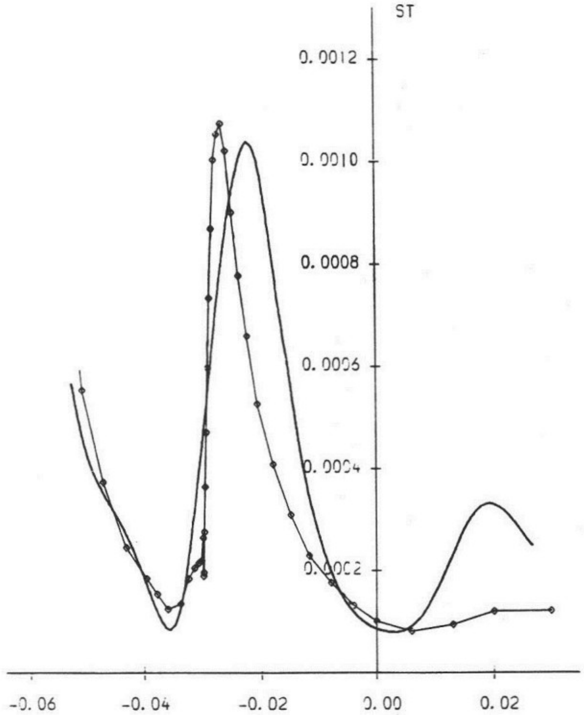

Fig. 17 St number on the leeward direction (— experiment, -∘- calculation)

in excellent agreement with the experimental results. Fig. 16 displays the calculated and the experimental Stanton number on the plane of symmetry on the windward direction. The comparison of Stanton number on the leeward side is made in Fig. 17. The computed and experimental results agree very well indeed. In the simulation of hypersonic flow, the heat transfer rate is usually very difficult to predict correctly especially in the viscous separated flow domain. It is shown that the current numerical simulation has excellent agreement with the experiment even on the leeward side. It is illuminating to plot the temperature and velocity profiles on the plane of symmetry as done in Fig. 18. From this figure it is convincing that sufficient grid points exist in the viscous regions to expect good resolution, and that these regions are simulated very satisfactorily.

No experimental data on the skin friction coefficient is available. However, for convenience of comparison with other calculation, and to facilitate the identification of separated flow regions, the Cf on the plane of symmetry is plotted in Fig.19. The separation and reattachment point can be easily determined from this figure.

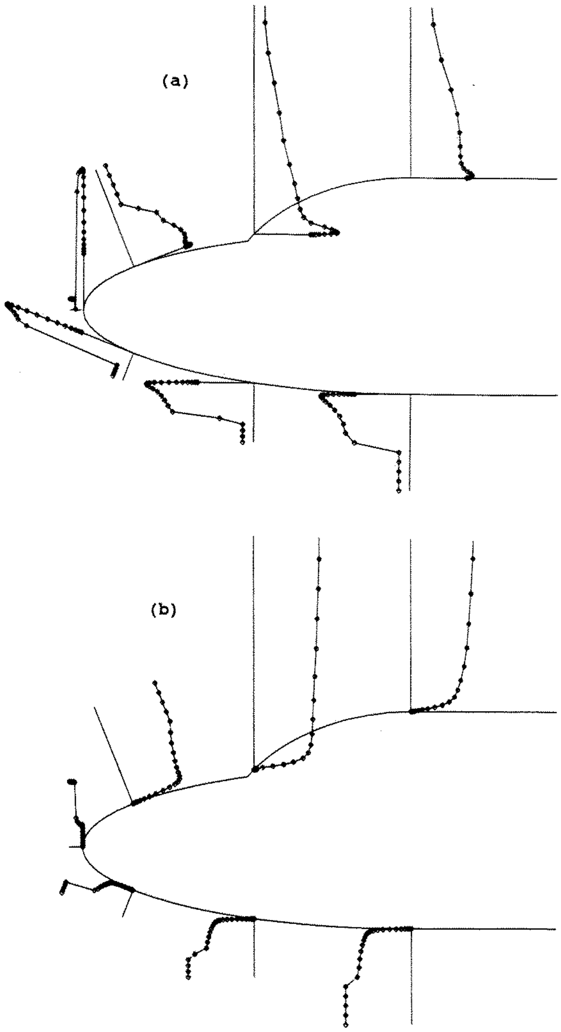

Fig. 18 Temperature (a) and velocity (b) profile on the plane of symmetry

Conclusions

Numerical experiments have been carried out to model the flow over generalized shapes representing a spaceplane during re-entry flight. Reduced Navier-Stokes (N-S) solvers such as based on assumptions of locally conical flow have proved a valuable tool for explaining flows of of this type. A sensitivity study carried out with such a code has allowed a suitable choice of mesh size to have been made and had identified Osher's flux difference splitting (FDS) scheme to be a superior approach amongst other N-S solvers. The high quality of the output has allowed a more detailed understanding of the flow fields

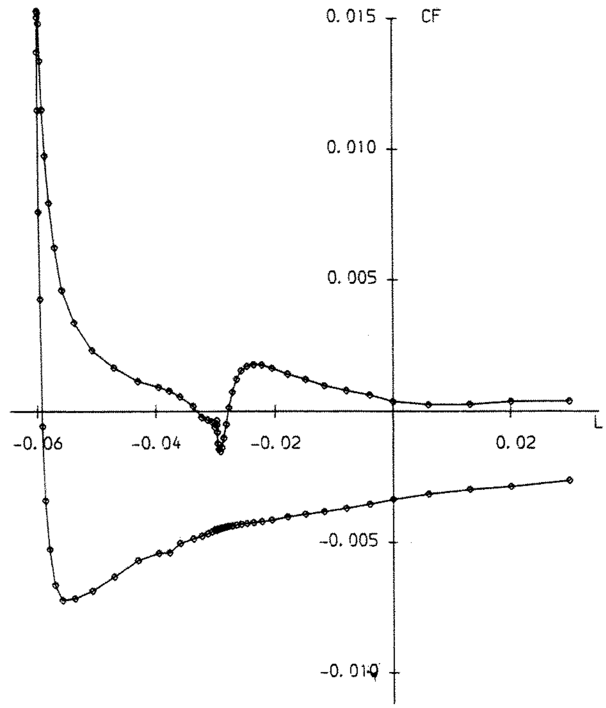

Fig. 19 Computed Cf on the plane of symmetry

predicted than achieved previously. An extension of the Osher FDS scheme with additional acceleration techniques has enable the complex flow over a blunt leading edged delta wing to be calculated with accuracy and high resolution. A grid generator scheme based on transfinite interpolation techniques has proved successful in generating adequate meshes.

A comprehensive study of total variation diminishing (TVD) schemes which were developed to avoid spurious oscillations normally expected near shock waves, have been shown to be appropriate, with modification, to tackle the prediction of flows with extensive viscous regions. Such modified TVD schemes have been applied to solve the full 3D N-S equations on a double ellipsoid shape representing the nose and canopy of a space plane shape at M=8.15, angle of attack of 30 degrees and a Reynolds number of 1670000. Comparisons of these predictions with surface pressure and heat transfer from wind tunnel tests have proved excellent. Detailed temperature and velocity profiles, flow field plots and simulated surface flow visualisation pictures have proved useful in understanding the flow behaviour.

It is concluded overall that numerical experiments based on validated N-S solvers are proving as valuable as experimental techniques towards the understanding of complex flow phenomena. These techniques show promise in providing the design tool for aerospace vehicles in the future.

Acknowledgement

Support is gratefully acknowledged, for various parts of the work, from SERC, SBFSS (administered by the British Council), MOD (Dr. T.A. Holbeche, RAE Farnborough) and B. Ae (through Mr. Philip Varty).

References

1. Qin, N. and Richards, B.E.
Sparse quasi-Newton method for high resolution schemes. 8th GAMM Conference on Numerical Methods Louvain, Sept. 1987. Notes of Numerical Fluid Mechanics Vol. 20. pp.310-317 Vieweg, Braunschweig, 1988.

2. Qin, N. and Richards, B.E.
Sparce quasi-Newton Method for Navier-Stokes solution. 7th GAMM Conference on Numerical Methods in Fluid Mechanics, Delft, Sept. 1989. Notes on Numerical Fluid Mechanics Vieweg Braunscheig 1990.

3. Qin, N., Scriba, K.W. and Richards, B.E.
Shock-shock, shock-vortex interaction and aerodynamic heating in hypersonic corner flow. International Conference on Hypersonic Aerodynamics, Manchester, U.K. Sept. 1989.

4. Wang, Z.J. and Richards, B.E.
High resolution schemes for steady inviscid and viscous flow. Numeta 90. Vol. 2, pp. 995-1002. Elsevier Applied Science 1990.

5. Jiang, D. and Richards, B.E.
Numerical solution of the Navier-Stokes equations for transonic separated flow. Royal Aeronautical Society Conference: The Prediction and Exploitation of Separated Flow, 18-20 April, 1989, Paper 9.

6. Jiang, D.C. and Richards, B.E.
Hypersonic Viscous Flow over Two-Dimensional ramps. Workshop on Hypersonic Flows for Re-entry Problems. January 22-25, 1990, Antibes (France). Vol. 3, pp. 147-162.

7. Anderson, J.M.
Implementation of equilibrium air models within the framework of an existing Euler code. Unpublished note, Sept., 1989.

8. Wang, Z.
Numerical simulation of 3D hypersonic flow using high resolution schemes. Ph.D. Thesis, University of Glasgow, May 1990.

9. Shu, C. and Richards, B.E.
Multidomain solutions of incompressible flow around complex geometries by generalised differential quadrature. Journal of Computational Physics (under review).

10. Qin, N. and Richards, B.E.
A parabolised Navier-Stokes code combining high resolution with fast convergence. AGCFM Workshop organised by the Royal Aerospace Establishment, University of Surrey, Sept. 1989.

11. Qin, N. and Richards, B.E.
PNS solution using a sparse quasi-Newton method for fast convergence. 12th International Conference on Numerical Methods in Fluid Dynamics, Oxford. Springer Verlag 1990.

12. Scriba, K.W.
Unpublished paper, 1990.

13. Aymer, D., Alziary Th., Carlomagno, G. and De Luca L.,
Experimental study of flow over a double ellipsoid. Workshop on Hypersonic Flows for Re-entry Problems. January 22-25, 1990, Antibes (France). Vol. 4, pp. 1-10.

14. Hummel, D.
Experimental investigation on the blunt bodies and corner configurations in hypersonic flow. AGARD-CP-428, 1987, paper 6.