Computations of flow field over Apollo and OREX reentry modules at high speed

R C Mehta*

School of Mechanical and Aerospace Engineering, Nanyang Technological University, Singapore 639 798, Singapore

Received 11 May 2007; accepted 18 August 2008

The paper describes a numerical simulation of the viscous flow past the Apollo and the OREX (Orbital Reentry EXperiments) configurations for freestream Mach numbers range of 1.2-5.0. The flow fields over the reentry module are obtained by solving time-dependent, axisymmetric, compressible Navier-Stokes equations. The fluid mechanics equations are discretized in spatial coordinate employing a finite volume method, which reduces the governing equations to semi-discretized ordinary differential equations. Temporal integration is carried out using a two-stage Runge-Kutta time-stepping scheme. A local time-stepping is used to obtain the steady state solution. The numerical simulation is done on a single-blocked structured grid. The flow field features around the reentry capsules such as bow shock wave, sonic line, expansion fan and recirculating flow in the base-shell region are well captured by the present numerical computation. A low pressure is observed immediately downstream of the base which is characterized by a low-speed recirculating flow region, which can be attributed to fill-up in the growing space between the shock wave and the reentry module.

The effects of the module geometrical parameters, such as radius of the spherical cap radius, shoulder radius, cone angle and back shell inclination angle on the flow field, which will provide a useful input for the optimization of the reentry module.

The primary design consideration of reentry capsules requires large spherical nose radius of their forebody that gives high aerodynamic drag and a short body length for reducing the total structure weight and the ballistic coefficient. The forebody shape of reentry capsules can be selected either employing a spherical cap, or a combination of spherical nose with cone, or a spherical blunt cone/flare configuration. To ensure the deployment of parachute in the transonic region, the flow field past the capsule must be known at supersonic speeds. The flow field in the wake region of a reentry capsule is complex due to the expansion at the shoulder and the base-shell. The bow shock wave is detached from the blunt fore body and is having a mixed subsonic-supersonic region between them. The surface pressure distribution, the location of the sonic line and the shock stand-off distance on the spherical cap have been analytically calculated at very high speeds with an adiabatic index near to unity which gives a singular point at 60° from the stagnation point. The flow-field over the reentry capsule becomes further complicated due to the presence of corner at the shoulder and the base shell of the reentry module. The flow is curved in the direction of the freestream on the spherical cap of the capsule and the pressure from the shock wave to the body surface equals to the centrifugal force due to the curvature of the flow. The pressure coefficient behind the shock wave depends on the surface slope of the fore body of the reentry module. The pressure relief due to curvature depends on the local air mass, the velocity, and the radius of the curvature of the spherical cap. The shape of the shock and detachment distance depends on the geometry of the body and on the freestream Mach number. The analytical approach to study the high-speed flow-past the blunt-body is considerably difficult and complex. The flow field features over the reentry capsules can be delineated through numerical simulation at high speeds.

A large number of computational fluid dynamics simulations has been performed for aerobraking and reentry capsules. Allen and Cheng have carried out the numerical solution of Navier-Stokes equations in the near wake region of the reentry module, which confirms the mechanism of flow separation as, observed experimentally. Base drag represents the loss in recovery of pressure over the base of the capsule. The supersonic and hypersonic laminar flow over a slender cone has been numerically calculated by Tai and Kao. A summary of developments relating to the base pressure prediction is reported in the review paper of Lamb and Oberkampf. An aerodynamic analysis of the Commercial Experiment Transport (COMET) reentry capsule carried out by Wood et al. by solving the laminar thin layer Navier-Stokes equations flow
solver LAURA. The flow field past blunt and short reentry capsule has been analyzed in order to understand the mechanism of the instability at supersonic speeds due to decay of base pressure. Yamamoto et al. have computed flow field over the OREX reentry module in conjunction with the in-depth the thermal analysis of thermal protection system and results were compared with the flight data. Tam has used LUSGS implicit scheme for flow computation over On-Axis Biconic and Aeroassist Flight Experiment (AFE) reentry vehicles. Liever et al. solved the flow field over Beagle reentry capsule. The flow field and the heat flux computation over the Mars pathfinder vehicle has been numerically carried out by Haas along with fore body and wake flow structure during atmospheric entry of the spacecraft.

The literature survey shows that the fore body shape of the reentry capsules can be classified either using as a spherical cap, or a combination of the spherical cap with cone. In the present work, numerical studies were undertaken for a freestream supersonic Mach numbers of 1.2-5.0. The numerical simulation to solve the axisymmetric laminar compressible unsteady Navier-Stokes equations is by employing a two-stage Runge-Kutta time-stepping scheme. The numerical scheme is second order accurate in space and time. The numerical simulation is carried out on a monoblock structured grid. Surface pressure and forebody aerodynamic drag on the Apollo and the OREX (Orbital Reentry EXperiments) configurations are computed numerically, which will give a systematic understanding of the flow features at supersonic Mach numbers and varying geometrical parameters of the reentry modules. The objective of the present note is to provide an insight into the flow field such as the separated zone and vortex formation for two different kinds of reentry modules. The effects of the module geometrical parameters, such as radius of the spherical cap radius, shoulder radius, cone angle and back shell inclination angle on the flow field, which will provide a useful input for the optimization of the reentry module.

Problem Definition and Approach

Governing fluid dynamics equations
The time-dependent axisymmetric compressible Navier-Stokes equations are written in integral form with the ideal gas law for solution augmenting the system of equations. The coefficient of molecular viscosity is computed according to Sutherland’s law. The flow is assumed to be laminar, which is consistent with the numerical simulation of.

Numerical algorithm
The flow field code employs a finite volume discretization technique. Using a finite-volume approach, the governing equations are discretized in space starting from an integral formulation without any intermediate mapping. The spatial and temporal terms are decoupled using the method of lines. The spatial computational domain is divided into a number of non-uniform and non-overlapping quadrilateral grids. A cell-centred scheme is used to store the flow variables. On each cell face the convective and diffusive fluxes are calculated after computing the necessary flow quantities at the face centre. These quantities are obtained by a simple averaging of adjacent cell-centre values of dependent variables. The numerical procedure reduces to central differencing on a rectangular and smooth grid. The entire spatial discretization scheme is second-order accurate. In viscous calculations, the dissipative properties are present due to diffusive terms. Away from the shear layer regions, the physical diffusion is generally not sufficient to prevent the odd-even point decoupling of centered numerical schemes. Thus, to maintain numerical stability and to prevent numerical oscillations in the vicinity of shocks or stagnation points, artificial terms are included as blend of a Laplacian and biharmonic operator in a manner analogous to the second and fourth differences. Artificial dissipation terms are added explicitly to prevent numerical oscillations near shock waves to damp high frequency undamped modes.

Temporal integration is performed using two-stage Runge-Kutta time stepping scheme of Jameson et al. The artificial dissipation terms are evaluated only at the first stage. The two-stage Runge-Kutta time-stepping method is second order accurate in time for a linear system of one-dimensional equations. A conservative choice of the Courant-Friedrichs-Lewy number, CFL = 0.8 is made to obtain a stable numerical solution. A local time-step is used to obtain steady-state solution.

Initial and boundary conditions
Conditions corresponding to supersonic freestream Mach numbers are given as an initial condition in Table 1. The boundary conditions are as follows: All variables are extrapolated at the outer boundary, and a...
no-slip condition is used as wall boundary condition. An isothermal wall condition is considered for the surface of the reentry configuration. The wall temperature is prescribed as 231 K. A symmetry condition is applied on the centre line ahead and downstream of the reentry capsule.

Geometrical details of reentry modules
The dimensional details of the Apollo and the OREX modules, shown in Fig. 1 are of axisymmetric designs. The Apollo capsule has a spherical blunt nose diameter of $D = 3.95$ m, spherical nose radius of $R_N = 4.595$ m and a shoulder radius of $R_C = 0.186$ m. The back shell has an inclination angle, $\alpha_B = 32.5^\circ$ relative to the vehicle’s axis of symmetry as depicted in Fig. 1a. The overall length of the module is $L = 3.522$ m.

The OREX has a spherical nose cap and a conical section of diameter $D = 3.4$ m with an apex half-angle $\alpha_N = 50^\circ$ as depicted in Fig. 1b. The outer edge of the vehicle has a rounded edge of $R_C = 0.01$ m and the rear of the vehicle is made up of a conical panel with an apex half angle $\alpha_B = 75^\circ$ as measured from the clockwise direction. The overall length of the OREX module is $L = 1.508$ m.

Computational grid
One of the controlling factors for the numerical simulation is proper grid arrangement. In order to initiate the numerical simulation of the flow along the reentry module, the physical space is discretized into non-uniform spaced grid points. These body-oriented grids are generated using a finite element method in conjunction with homotopy scheme. The typical computational space of the reentry module is defined by a number of grid points in a cylindrical coordinate system. Using these surface points as the reference nodes, the normal coordinate is then described by exponentially structured field points, extending onwards up to an outer computational boundary. The stretching of grid points in the normal direction is obtained using the exponentially stretching relation.

These grids are generated in an orderly manner. Grid independence tests were carried out, taking into consideration the effect of the computational domain, the stretching factor to control the grid density near the wall, and the number of grid points in the axial direction.
and normal directions. A rigorous grid refinement study with successive doubling of the number of cells in each direction is carried out. The present numerical analysis is carried out on $132 \times 62$ grid points. Figure 2 displays the enlarged view of the mono-block structured grid over the Apollo and the OREX reentry configurations. This spatial resolution is adequate for fine resolution of the boundary layer and the complex flow field. The finer mesh near the wall helps to resolve the viscous effects. The coarse grid helps reducing the computer time. The grid-stretching factor is selected as 5, and the outer boundary of the computational domain is maintained as 1.5-2.5 times maximum diameter of the reentry module. In the downstream direction the computational boundary is about 6-9 times the diameter of the module, $D$. The nature of the flow fields examined in this study is generally quasi-steady. The grid arrangement is found to give a relative difference of about ±1.5% in the computation of drag coefficient. The convergence criterion is based on the difference in density values $\rho$ at any of the grid points, between two successive iterations $|\rho^{n+1} - \rho^n| \leq 10^{-5}$ where $n$ is time-step counter.

**Results and Discussion**

The numerical procedure mentioned in the previous section is applied to simulate the flow field over the Apollo and the OREX reentry capsules for freestream Mach numbers in the range of 1.2-5.0, and for freestream Reynolds numbers ranging from $1.967 \times 10^7 - 8.198 \times 10^7/m$, based on the trajectory conditions as given in Table 1.

**Flow characteristics**

Figures 3 and 4 show the closed view of the velocity vector plots over the Apollo and the OREX at $M_\infty = 1.2-5.0$. It can be seen from the vector plots that the bow shock wave follows the body contour relatively close to the fore body. A separated flow can...
be observed in the base region of the reentry capsules. The flow around the capsule is divided into two regions inside and outside of the recirculation, and the shear layer separates the regions. The flow field is very complex because of the back-shell geometry. The wake flow field, immediately behind the capsule base, exhibits vortex flow behaviours. The formation of the bow shock wave on the fore body of the OREX capsule depends on geometrical parameters such as spherical cap radius and the apex cone angle, and the value of the freestream Mach number. The bow shock wave moves close to the fore body with the increasing freestream of the Mach number, i.e., stand-off distance between bow shock wave and the fore body decreases with increasing of the freestream Mach number. The approaching boundary layer separates at the corner and the free shear layer is formed in the wake region. The wake flow also shows a vortex attached to the corner with a large recirculation, which depends on spherical nose radius, apex cone angle, back-shell inclination angle and freestream Mach number. The separation point moves downstream from the shoulder towards the base with increasing $M_\infty$. Similar flow field features were observed in the analysis of the bulbous payload shroud of the heat shield of the launch vehicle.

Computed Mach contour plots around the Apollo and the OREX for $M_\infty = 1.2-5.0$ are depicted in Figs 5 and 6. The velocity vector plots show the formation of vortices at the corner region of the capsule for $M_\infty \leq 3$. Characteristic features of the flow field around the blunt body at supersonic Mach numbers, such as bow shock wave ahead of the capsule, the wake, and the recompression shock waves emanating from the

![Fig. 4—Closed-up view of the vector plot over OREX reentry capsule](image-url)
Fig. 5 — Mach contour over Apollo

Fig. 6 — Mach contours over OREX
shoulder point, are seen in the Mach contour plots. The bow shock wave following the body contour and the fore body is entirely subsonic up to the corner point of the Apollo and the OREX modules, where the sonic line is located. The Mach contour plots reveal many interesting flow features of the reentry capsule. The flow expands at the base corner and is followed by the recompression shock downstream of the base, which realigns the flow. The flow then develops in the trailing wake. As observed in the figures, vortices are generated at the capsule surface and are then moving, changing location with freestream Mach number. One can also see the strong vortex flow over the shoulder of the capsule at freestream Mach number 1.2 and 2.0. The flow may become unsteady at supersonic Mach number due to the formation of the vortices. Note, however, that the use of a fixed CFL number in the present numerical flow simulation leads to a local time step size, which differs throughout the flow domain. The local time stepping scheme gives rapid convergence for steady-flow problem but cannot compute time accurate behaviour. Rapid expansion around the fore body corners produces high Mach numbers in the outer inviscid region of the wake.

**Surface pressure distribution**

Figures 7 and 8 display the pressure coefficient \( C_p = 2(\bar{p}/\bar{p}_\infty - 1)/(\gamma M_\infty^2) \) variation along the model surface for the Apollo and the OREX for \( M_\infty = 1.2-5.0 \). The \( s/D = 0 \) location is the stagnation point, where \( s \) represents surface arc distance length. \( D \) is the maximum diameter of the capsule. The variation of pressure coefficient on the spherical region decreases gradually for the Apollo and the OREX capsules whereas in the conical region of the OREX it remains constant. The pressure coefficient falls on the sphere-cone junction and remains constant over the cone for the OREX and the sonic point moves to the corner of the blunt bodies and affects the pressure distribution throughout the subsonic flow.

In the case of the OREX with \( \alpha_N = 50^\circ \), the pressure coefficient shows over expanded flow. A sudden drop of pressure coefficient is observed on the shoulder of the module followed by the negative pressure coefficient variation in the base region. A low pressure is formed immediately downstream of the base, which is characterized by a low speed recirculating flow region, which can be attributed to fill up the growing space between the shock wave and body. In the base region, the pressure coefficient is decreasing with increasing freestream Mach number. The effect of the corner radius on the pressure coefficient is higher. The \( C_p \) variation depends on the geometry of the capsules. The value of \( C_p \) in the back-shell region of the OREX is very low as compared with the Apollo module at \( M_\infty = 1.2 \), which shows the influence of \( \alpha_B \). A wavy pattern is observed in the pressure distribution in the base region, which
may be attributed to the unsteady nature of the flow in the \( s/D = 0.5 \) to 1.425 for the Apollo and \( s/D = 0.6 \) to 1.12 for the OREX at \( M_{\infty} = 1.2 \). A low pressure is observed immediately downstream of the base which is characterized by a low-speed recirculating flow region, which can be attributed to filling of the growing space between the shock wave and the reentry module.

Pressure wave drag is calculated by integrating the pressure distribution on the body surface excluding the base pressure. The forebody aerodynamic drag \( C_D \) is given in Table 2 for the Apollo and the OREX for \( M_{\infty} = 1.2-5.0 \). For the calculation of \( C_D \), the reference area is the maximum cross-section area of the capsule. The base pressure is somewhat constant. The value of \( C_D \) for the Apollo capsule compared to the OREX at \( M_{\infty} = 1.2 \) and 2.0. Then, the \( C_D \) becomes higher for the Apollo capsule compared to the OREX module at high Mach numbers. Thus the value of \( C_D \) depends on the fore body geometry.

**Conclusions**

The flow field over the Apollo and the OREX reentry capsules is computed by solving compressible, laminar, and time-dependent axisymmetric Navier-Stokes equations. A single-block structured, axisymmetric, finite volume code solves the governing fluid dynamics equations using two-stage Runge-Kutta time stepping scheme with local time stepping in order to accelerate the convergence for obtaining a steady state solution. All the essential flow field features are fairly well captured such as bow shock wave, expansion on the corner, recompression shock wave and recirculation flow in the base region. The Apollo and the OREX capsules have the sonic line over the fore body shoulder. The pressure coefficient distribution along the surface of the capsules and integrated value of the pressure coefficient are important aerodynamic parameters for designing the capsule configuration. The flow field visualization of the separation region helps in a systematic understanding of the flow field features under various freestream Mach number. A low pressure is observed immediately downstream of the base which is characterized by a low-speed recirculating flow region, which can be attributed to filling of the growing space between the shock wave and the reentry module.

**References**