Defense Technical Information Center
Compilation Part Notice

ADP023738

TITLE: Time-Accurate Calculations of Free-Flight Aerodynamics of Maneuvering Projectiles

DISTRIBUTION: Approved for public release, distribution unlimited

This paper is part of the following report:

TITLE: Proceedings of the HPCMP Users Group Conference 2007. High Performance Computing Modernization Program: A Bridge to Future Defense held 18-21 June 2007 in Pittsburgh, Pennsylvania

To order the complete compilation report, use: ADA488707

The component part is provided here to allow users access to individually authored sections of proceedings, annals, symposia, etc. However, the component should be considered within the context of the overall compilation report and not as a stand-alone technical report.

The following component part numbers comprise the compilation report:
ADP023728 thru ADP023803

Time-Accurate Calculations of Free-Flight Aerodynamics of Maneuvering Projectiles

Jubaraj Sahu
US Army Research Laboratory (ARL), Aberdeen Proving Ground, MD
jubaraj.sahu@us.army.mil

Abstract

This paper describes a multidisciplinary computational study undertaken to model the flight trajectories and the free-flight aerodynamics of finned projectiles both with and without control maneuvers. Advanced computational capabilities both in computational fluid dynamics (CFD) and rigid body dynamics (RBD) have been successfully fully coupled on high performance computing (HPC) platforms for "Virtual Fly-Outs" of munitions similar to actual free flight tests in the aerodynamic experimental facilities. Time-accurate Navier-Stokes computations have been performed to compute the unsteady aerodynamics associated with the free flight of a finned projectile at a supersonic speed using an advanced scalable unstructured flow solver on a highly parallel Linux Cluster. Some results relating to the portability and the performance of the flow solver on the Linux clusters are also addressed. Computed positions and orientations of the projectile along the flight trajectory have been compared with actual data measured from free flight tests and are found to be generally in good agreement. Computed results obtained for another complex finned configuration with canard-control pitch-up maneuver in a virtual fly-out show the potential of these techniques for providing the actual time-dependent response of the flight vehicle and the resulting unsteady aerodynamics for maneuvering projectiles.

1. Introduction

As part of a Department of Defense (DoD) High Performance Computing Modernization Program (HPCMP) Grand Challenge Project, the US Army Research Laboratory (ARL) has recently focused on the development and application of state-of-the art numerical algorithms for large-scale simulations[1-3] to determine both steady and unsteady aerodynamics of projectiles with and without flow control. Our objective is to exploit CFD techniques on HPC platforms for design and analysis of guided projectiles whether the control forces are provided through canard control or jet control such as the micro adaptive flow control systems for steering spinning projectiles[3]. The idea is to determine if these control devices can provide the desired control authority for course correction for munitions. Knowledge of the detailed aerodynamics of maneuvering guided smart weapons is rather limited. Multidisciplinary computations can provide detailed fluid dynamic understanding of the unsteady aerodynamics processes involving the maneuvering flight of modern guided weapon systems. Such knowledge cannot be obtained by any other means. The computational technology involving CFD and RBD is now mature and can be used to determine the unsteady aerodynamics associated with control maneuvers. These multidisciplinary computations can lead to better experimental test designs and much better returns for full-scale flight tests. More importantly, they can provide physical insight of fluid mechanics processes that may not be gained from experimental techniques and flight tests.

Our goal is to perform real-time multidisciplinary coupled computational fluid dynamics/rigid body dynamics computations for the flight trajectory of a complex guided projectile system and fly it through a "virtual numerical tunnel" similar to what happens with the actual free flight of the projectile inside an aerodynamics experimental facility. Our initial attempt in the past has been to use a quasi-unsteady approach[4] in an effort to save computer time and resources. Recently progress has been made in the coupling of CFD and RBD to perform required time-accurate multidisciplinary simulations for moving body problems. Recent advances in computational aerodynamics and high performance computing make it possible to solve increasingly larger problems including these multidisciplinary computations at least for short time-of-flights. Complete time-accurate multidisciplinary simulations of entire trajectory for long-range guided munitions are still beyond the scope. A few highly optimized, parallelized codes[5,6] are available for

0-7695-3088-5/07 $25.00 © 2007 IEEE

such computations. Various techniques, including structured, unstructured, and overset methods can be employed to determine the unsteady aerodynamics of advanced guided projectiles and missiles. This involves real-time multi-disciplinary coupled CFD/RBD aerodynamics computations for the flight trajectory of a complex guided projectile system, much like a "Virtual Fly-Out" through the supercomputer. In the present study, a real time-accurate approach using unstructured methodology has been utilized to perform such complex computations for both a spinning projectile and a finned projectile. The maneuver of a projectile through flow control adds another complexity to the trajectory and the unsteady aerodynamics computations. Trajectory computations for the finned projectile have been accomplished both without and with aerodynamic flow control through the use of canard control surfaces. The emphasis in the present research is to provide insight into the time-dependent interaction of these canard maneuvers and the resulting unsteady flow fields and control forces that affect the flight dynamics response of the vehicle. In addition, the objective was to accomplish this through coupled CFD and RBD methods i.e. through virtual fly-outs.

Calculations for the finned configurations were performed using a scalable parallel Navier-Stokes flow solver, CFD++[7,8]. This flow solver is a multipurpose, unstructured, three-dimensional (3D), implicit, Navier-Stokes solver. The method used is scalable on SGI, IBM SP4, and newer computers such as the Linux Cluster. The method incorporates programming enhancements such as dynamic memory allocation and highly optimized cache management. It has been used extensively in the parallel high performance computing numerical simulations of projectile and missile programs of interest to the US Army. The code uses Riemann solvers to provide proper signal propagation physics, preconditioned forms of the governing equations for low speed flows, and pointwise turbulence models that do not require knowledge of distance to walls. The advanced CFD capability used in the present study solves the Navier-Stokes equations[9] and incorporates unsteady boundary conditions for simulation of the synthetic microjets[10,11] as well as a higher order hybrid Reynolds-Averaged Navier-Stokes (RANS)/Large Eddy Simulation (LES) turbulence model[12-15] for accurate numerical prediction of unsteady turbulent flows. Sahu[3] used these advanced techniques and performed numerical flow field computations for both steady and unsteady jets for a spinning projectile configuration at a low subsonic speed. The present numerical study is a big step forward which now includes numerical simulation of the actual fight paths of the finned projectiles both with and without flow control using coupled CFD/RBD techniques. The following sections describe the coupled numerical procedure and the computed results obtained for the finned bodies at supersonic speeds both with and without control maneuvers.

2. Computational Methodology

The complete set of 3D time-dependent Navier-Stokes equations[9] is solved in a time-accurate manner for simulations of unsteady flow fields associated with both spinning and finned projectiles during flight. The 3D time-dependent RANS equations are solved using the finite volume method[7,8]:

$$\frac{\partial}{\partial t}\int_V \mathbf{W}dV + \oint[\mathbf{F}-\mathbf{G}]\cdot dA = \int_V \mathbf{H}dV \qquad (1)$$

where **W** is the vector of conservative variables, **F** and **G** are the inviscid and viscous flux vectors, respectively, **H** is the vector of source terms, V is the cell volume, and A is the surface area of the cell face.

Second-order discretization was used for the flow variables and the turbulent viscosity equations. Two-equation[15] and higher order hybrid RANS/LES[7] turbulence models were used for the computation of turbulent flows. The hybrid model transitions smoothly between an LES calculation and a cubic k-ε model, depending on grid fineness. Dual time-stepping was used to achieve the desired time-accuracy.

Grid was actually moved to take into account the spinning motion of the projectile and grid velocity is assigned to each mesh point. This general capability can be tailored for many specific situations. For example, the grid point velocities can be specified to correspond to a spinning projectile. In this case, the grid speeds are assigned as if the grid is attached to the projectile and spinning with it. Similarly, to account for rigid body dynamics, the grid point velocities can be set as if the grid is attached to the rigid body with six-degrees-of-freedom (6DOF). For the rigid body dynamics, the coupling refers to the interaction between the aerodynamic forces/moments and the dynamic response of the projectile/body to these forces and moments. The forces and moments are computed every CFD time step and transferred to a 6DOF module which computes the body's response to the forces and moments. The response is converted into translational and rotational accelerations that are integrated to obtain translational and rotational velocities and integrated once more to obtain linear position and angular orientation. The 6DOF rigid body dynamics module uses quaternions to define the angular orientations. However, these are easily translated into Euler angles. From the dynamic response, the grid point locations and grid point velocities are set.

Typically we begin with a computation performed in "steady state mode" with the grid velocities prescribed to account only for the translational motion component of the complete set of initial conditions to be prescribed. At this stage we also impose the angular orientations from the initial conditions. The complete set of initial conditions includes both translational and rotational velocity components along with initial position and angular orientation.

3. Parallel Computational Aspects

The CFD++ computational fluid dynamics simulations software was designed from the outset to include three unification themes: unified physics, unified grid, and unified computing. The "unified computing" capability includes the ability to perform scalar and parallel simulations with great ease. The parallel processing capability in CFD++ was designed in the beginning to be able to run on a wide variety of hardware platforms and communications libraries including Message Passing Interface, Parallel Virtual Machine, and proprietary libraries of nCUBE, Intel Paragon etc. The code is compatible with and provides good performance on standard Ethernet (e.g., 100Mbit, 1Gbit, 10Gbit) as well as high performance communications channels of Myrinet and Infiniband etc.

It is very easy to use CFD++ on any number of CPUs in parallel. The mesh files, restart and plot files) that are needed/generated for single CPU runs are identical to those associated with multi CPU runs. One can switch the use of an arbitrary number of CPUs at any time. The only extra file required is a domain-decomposition file, which defines the association between cell number and which CPU should consider that cell as its "native" cell. Depending on the number of CPUs being employed, the corresponding domain decomposition is utilized. The software suite includes several domain decomposition tools and it is also fully compatible with the METIS tool developed at the University of Minnesota. The code runs in parallel on many parallel computers including those from Silicon Graphics, IBM, Compaq (DEC and HP), as well as on PC workstation clusters. Excellent performance (see Figure 1 for the timings on a 4-million mesh) has been observed up to 64 processors on the Silicon Graphics O3K (400 MHz), the IBM SP P3 (375 MHz), the IBM SP P4(1.7GHz), and the Linux PC cluster (3.06 GHz). Computed results on the new Linux PC cluster seem to show 2- to 4-fold reduction in CPU time for number of processors larger than 16. Similar good performance is also achieved on the Linux PC cluster for a larger 12 million mesh (see Figure 2) up to 128 processors.

The computational algorithms implemented in CFD++ synergistically help achieve all three unification goals. In particular, the parallel processing capability is fully compatible with all types of meshes including structured and unstructured, steady and moving and deforming, single and multi block, patched and overset types. Inter-CPU communications are included at the fine grid level as well as all the multigrid levels to help ensure high degree of robustness consistently observed in using CFD++, independent of the number of CPUs being employed.

4. Results

4.1. Finned Projectile CFD with 6DOF

Time-accurate numerical computations were also performed using Navier-Stokes and coupled 6DOF methods to predict the flow field and aerodynamic coefficients, and the flight paths of a finned projectile at supersonic speeds.

The supersonic projectile modeled in this study is an ogive-cylinder-finned configuration (see Figure 3). The length of the projectile is 121 mm and the diameter is 13 mm. Four fins are located on the back end of the projectile. Each fin is 22.3 mm long and 10.16 mm thick. The computational mesh for the 25 mm projectile model is a C-grid (see Figure 4) consisting of seven zones. The first zone encompasses the entire projectile body, from the tip of the nose to the end of the fins. In general, most of the grid points are clustered in the afterbody fin region. Figure 4 shows a 3D view of the full projectile mesh. The total number of grid points is 1.5 million for the full grid.

The first step here was to obtain the steady state results for this projectile at a given initial supersonic velocity. Also imposed were the angular orientations at this stage. Corresponding converged steady state solution was then used as the starting condition along with the other initial conditions for the computation of coupled CFD/RBD runs. Numerical computations have been made for these cases at an initial velocities 1,037 and 1,034 m/s depending on whether the simulations were started from the muzzle or a small distance away from it. The corresponding initial angles of attack were, $\alpha = 0.5°$ or 4.9° and initial spin rates were 2,800 or 2,500 rad/s, respectively.

Figure 5 shows the computed z-distance as a function of x (the range). The computed results are shown in solid lines and are compared with the data measured from actual flight tests. For the computed results the aerodynamic forces and moments were completely obtained through CFD. One simulation started from the gun muzzle and the other from the first station away from the muzzle where the actual data was measured. The first station was located about 4.9 m from the muzzle. Both sets of results are generally found to be in good agreement with the measured data, although there is a small discrepancy between the two sets of computed results. The z-distance is found to increase with increasing x-distance.

Figure 6 shows the variation of the Euler pitch angle with distance traveled. Both the amplitude and frequency in the Euler pitch angle variation are predicted very well by the computed results and match extremely well with the data from the flight tests. Both sets of computations, whether it started from the muzzle or the first station away from the muzzle, yield essentially the same result. One can also clearly see the amplitude damped out as the projectile flies down range i.e., with increasing x-distance.

4.2. Complex Projectile in Supersonic Flight with Canard Maneuver

Another case considered in the study is a complex canard-controlled finned projectile. Here, the control maneuver is achieved by the two horizontal canards in the nose section (Figures 7–9). Unstructured Chimera overlapping grids were used (see Figure 7) and solutions have been obtained for several canard deflection cases. Figure 8 shows the computed pressure contours at $M = 3.0$ and $\alpha = 0°$ for a canard deflection of 20 deg. Although not shown here, this produces lift that can be used to obtain increased range. A typical result is shown in Figure 9 for the canard deflection of 20° (high pressure region shown in red and low pressure region in blue).

Some results for a "pitch maneuver" are shown in Figures 10 and 11. In this case, the two horizontal canards are rotated down 10° in 0.01 sec, held there for the next 0.01 sec, and then deflected back to their horizontal positions in the next 0.01 sec. This maneuver generates a lot of lift force (see Figure 10) until the end of this virtual fly-out simulation (time = 0.145 sec). This result in the nose of the projectile pitching up and the z-distance of the center of gravity of the projectile increasing from 0 to 0.5 meter (see Figure 11). Also, shown in this figure is the Euler pitch angle which goes from 0 to a peak value of about −16° (nose-up corresponds to negative Euler pitch angle). These results clearly show a large effect on the time-dependent response of the projectile subject to a canard maneuver.

5. Concluding Remarks

This paper describes a new coupled CFD/RBD computational study undertaken to determine the flight aerodynamics of finned projectiles using a scalable unstructured flow solver on various parallel computers such as the IBM, and the Linux Cluster. Advanced scalable Navier-Stokes computational techniques were employed to compute the time-accurate aerodynamics associated with the free flight of the finned projectiles at supersonic velocities both with and without control maneuvers. High parallel efficiency is achieved for the real time-accurate unsteady computations. For the finned configuration without control maneuver, coupled CFD/RBD calculations show the flight trajectory and the unsteady aerodynamics associated with its flight. Computed positions and orientations of the projectile have been compared with actual data measured from free flight tests and are found to be generally in very good agreement. Computed results obtained for another complex finned configuration with canard-control pitch-up maneuver in a virtual fly-out show the potential of these techniques for providing the actual time-dependent response of the flight vehicle and the resulting unsteady aerodynamics for maneuvering projectiles.

This work demonstrates a coupled method to accurately predict the time-accurate unsteady aerodynamics and the flight trajectories of projectiles at various speeds using HPC on highly parallel supercomputers. The present CFD/RBD simulations clearly show the capability of the coupled approach and form the basis for future multidisciplinary, time-dependent computations of advanced maneuvering munitions. The results show the potential of HPC coupled CFD and RBD simulations on parallel machines to provide insight into the unsteady flow fields leading to improve designs and accurate prediction of flight trajectories both with and without control maneuvers.

References

1. Sahu, J., D. Pressel, K.R. Heavey, and C.J. Nietubicz, "Parallel Application of a Navier-Stokes Solver for Projectile Aerodynamics." *Proceedings of Parallel CFD'97 Meeting*, Manchester, England, May 1997.

2. Sahu, J., H.L. Edge, S. Dinavahi, and B. Soni, "Progress on Unsteady Aerodynamics of Maneuvering Munitions." *Users Group Meeting Proceedings,* Albuquerque, NM, June 2000.

3. Sahu, J., "Unsteady Numerical Simulations of Subsonic Flow over a Projectile with Jet Interaction." *AIAA paper no. 2003-1352*, 41st Aerospace Sciences Meeting, Reno, NV, 6–9 Jan 2003.

4. Sahu, J., H.L. Edge, J. DeSpirito, K.R. Heavey, S.V. Ramakrishnan, and S.P.G. Dinavahi, "Applications of Computational Fluid Dynamics to Advanced Guided Munitions." *AIAA Paper No. 2001-0799*, 39th AIAA Aerospace Sciences Meeting Reno, Nevada, 8–12 January 2001.

5. Peroomian, O. and Chakravarthy S., "A 'Grid-Transparent' Methodology for CFD." *AIAA Paper 97-0724*, Jan. 1997.

6. Meakin, R. and Gomez, R., "On Adaptive Refinement and Overset Structured Grids." *AIAA Paper No. 971858-CP*, 1997.

7. Batten, P., U. Goldberg, and S. Chakravarthy, "Sub-grid Turbulence Modeling for Unsteady Flow with Acoustic

Resonance." *AIAA Paper 00-0473*, 38th AIAA Aerospace Sciences Meeting, Reno, NV, January 2000.

8. Peroomian, O., S. Chakravarthy, S. Palaniswamy, and U. Goldberg, "Convergence Acceleration for Unified-Grid Formulation Using Preconditioned Implicit Relaxation." *AIAA Paper 98-0116*, 1998.

9. Pulliam, T.H. and J.L. Steger, "On Implicit Finite-Difference Simulations of Three- Dimensional Flow" *AIAA Journal*, vol. 18, no. 2, pp. 159–167, February 1982

10. Smith, B.L. and A. Glezer, "The Formation and Evolution of Synthetic Jets." *Journal of Physics of Fluids*, vol. 10, No. 9, September 1998

11. Amitay, M., V. Kibens, D. Parekh, and A. Glezer, "The Dynamics of Flow Reattachment over a Thick Airfoil Controlled by Synthetic Jet Actuators." *AIAA Paper No. 99-1001*, January 1999

12. Arunajatesan, S. and N. Sinha, "Towards Hybrid LES-RANS Computations of Cavity Flowfields." *AIAA Paper No. 2000-0401*, January 2000.

13. Sandberg, R.D. and H.F. Fasel, "Application of a New Flow Simulation Methodology for Supersonic Axisymmmetric Wakes." *AIAA Paper No. 2004-0067.*

14. Kawai, S. and K. Fujii, "Computational Study of Supersonic Base Flow using LES/RANS Hybrid Methodology." *AIAA Paper No. 2004-68.*

15. Goldberg, U., O. Peroomian, and S. Chakravarthy, "A Wall-Distance-Free K-E Model with Enhanced Near-Wall Treatment." *ASME Journal of Fluids Engineering*, Vol. 120, pp. 457–462, 1998

16. Sahu, J., "Unsteady CFD Modeling of Aerodynamic Flow Control over a Spinning Body with Synthetic Jet." *AIAA Paper 2004-0747*, Reno, NV, 5–8 January 2004.

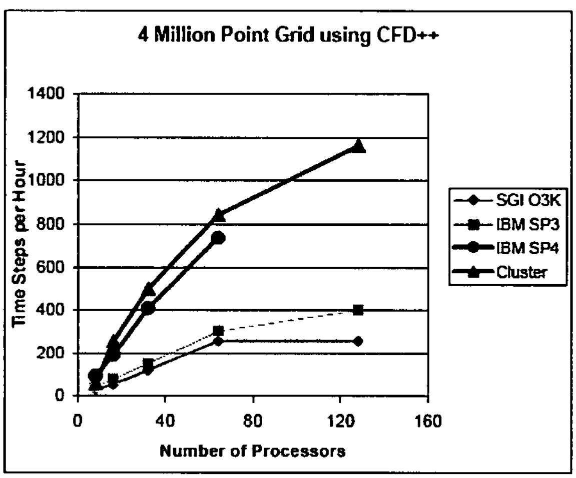

Figure 1. Parallel speedups (4-million grid)

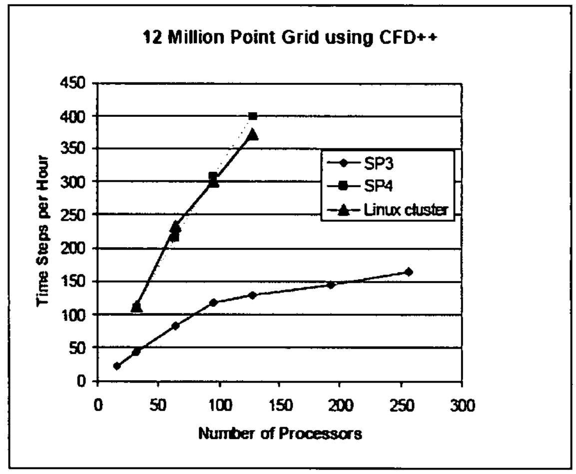

Figure 2. Parallel speedups (12-million grid)

Figure 3. Computational finned configuration

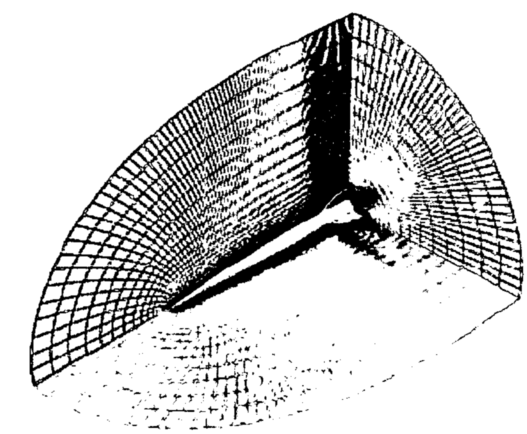

Figure 4. Computational grid

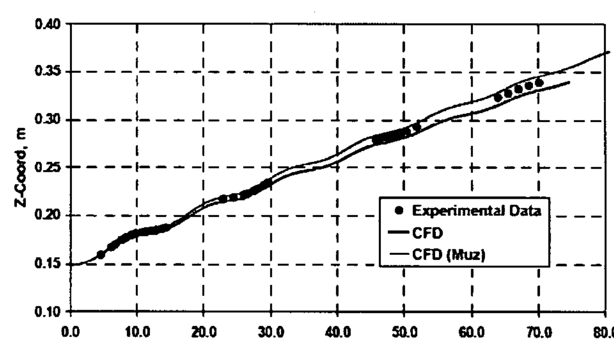

Figure 5. Computed z-distance vs. range

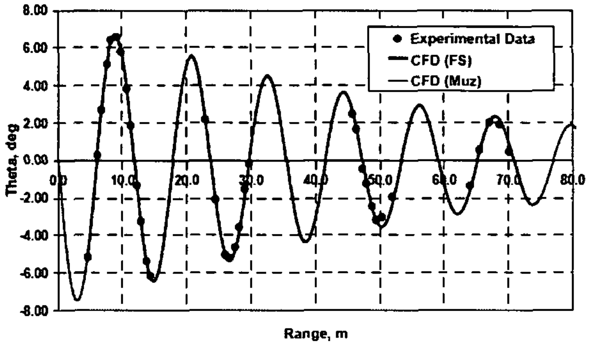

Figure 6. Euler pitch angle distance vs. range.

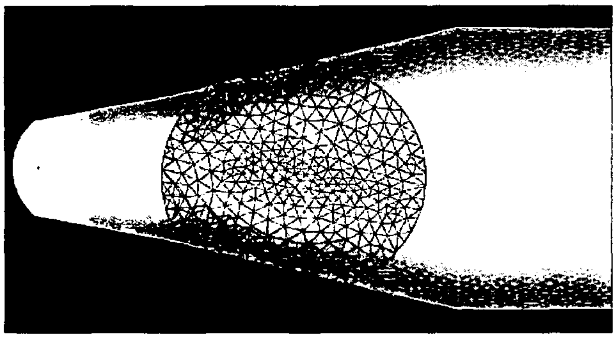

Figure 7. Unstructured Chimera mesh in the nose region (side view)

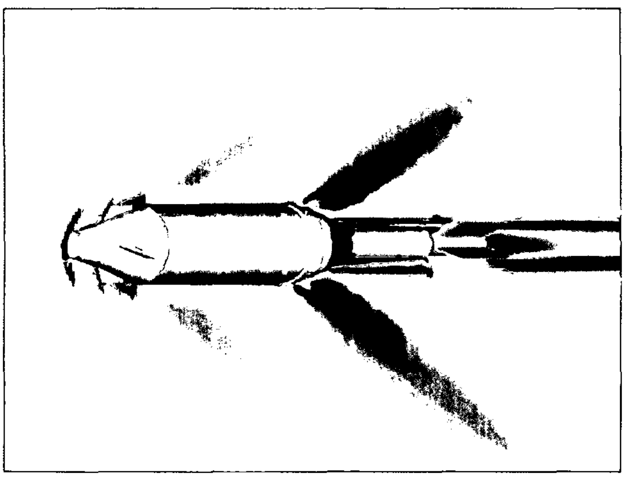

Figure 8. Computed pressure contours, M = 3.0, α = 0°

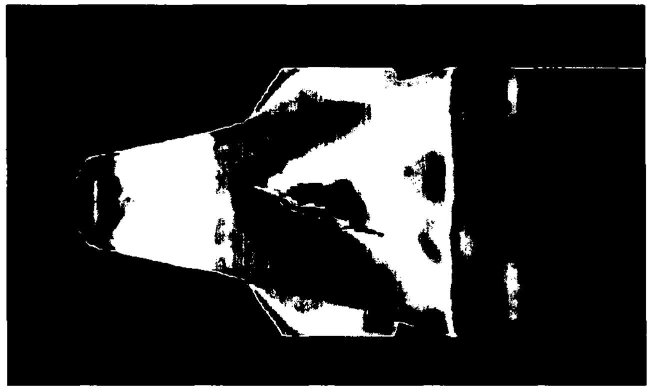

Figure 9. Computed surface pressure contours in the nose section, M = 3.0, α = 0°

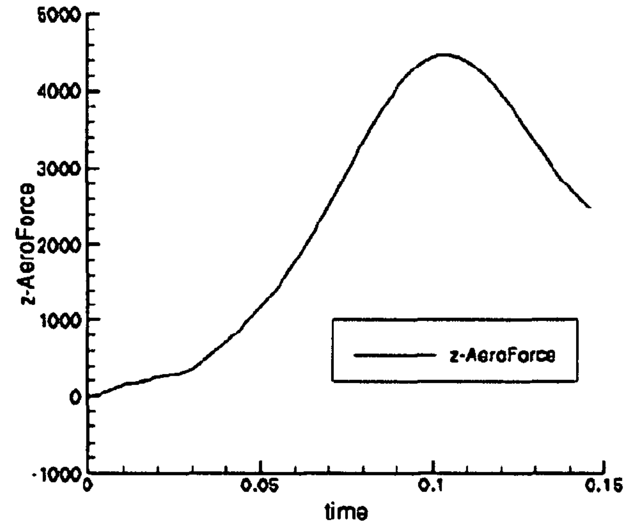

Figure 10. Time history of lift force

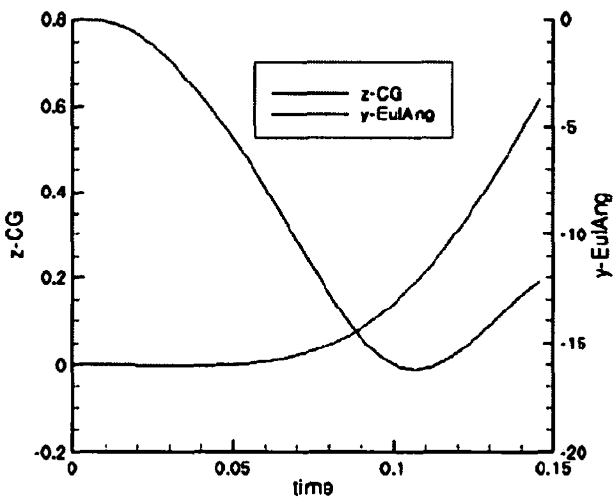

Figure 11. Time history of z-distance (center of gravity) and Euler pitch angle