PARALLEL CFD SIMULATIONS OF PROJECTILE FLOW FIELDS WITH MICROJETS

Jubaraj Sahu
Karen R. Heavey
U.S. Army Research Laboratory
Aberdeen Proving Ground, Maryland 21005-5066

ABSTRACT

As part of a Department of Defense Grand Challenge Project, advanced high performance computing (HPC) time-accurate computational fluid dynamics (CFD) techniques have been developed and applied to a new area of aerodynamic research on microjets for control of small and medium caliber projectiles. This paper describes a computational study undertaken to determine the aerodynamic effect of flow control in the afterbody regions of spin-stabilyzed projectiles at subsonic and low transonic speeds using an advanced scalable unstructured flow solver on various parallel computers such as the IBM SP4 and Linux Cluster. High parallel efficiency is achieved for both steady and time-accurate unsteady flow field simulations using advanced scalable Navier-Stokes computational techniques. Results relating to the code's portability and its performance on the Linux clusters are also addressed. Numerical simulations with the unsteady microjets show the jets to substantially alter the flow field both near the jet and the base region of the projectile that in turn affects the forces and moments even at zero degree angle of attack. The results have shown the potential of HPC CFD simulations on parallel machines to provide insight into the jet interaction flow fields leading to improve designs.

INTRODUCTION

As part of a DOD High Performance Computing (HPC) grand challenge project, the U.S. Army Research Laboratory (ARL) has recently focused on the development and application of state-of-the art numerical algorithms for large-scale simulations[1,2] to determine both steady and unsteady aerodynamics of projectiles with flow control. One of the objectives is to exploit computational fluid dynamics (CFD) techniques on HPC platforms for design and analysis of micro adaptive flow control (MAFC) [3,4] systems for steering spinning projectiles for infantry operations. The idea is to determine if the MAFC using microjets[1] can provide the desired control authority for course correction for munitions. These very tiny (of the order of 0.3mm) micro-jet actuators have been shown to successfully modify subsonic flow characteristics. The microjets (fluid being pumped in and out of the jet cavity at a high frequency of the order of 1000 Hz) are control devices with zero net mass flux and are intended to produce the desired control of the flow field through momentum effects. The current research effort is aimed at advancing the state-of-the-art in CFD and advanced flow visualization to accurately predict and provide a crucial understanding of the complex flow physics associated with both steady and unsteady aerodynamics of this new class of tiny micro-jets for control of modern projectile configurations.

The control of the trajectory of a spinning projectile is achieved by altering the pressure distribution on the projectile through forced asymmetric flow separation using the Coanda effect. Time-accurate unsteady CFD computations have been performed to predict and characterize the unsteady nature of the microjet interaction flow field produced on the projectile for various yaw and spin rates for fully viscous turbulent flow conditions. Turbulence was modeled using both Reyonlds-Averaged Navier-Stokes (RANS) and the hybrid RANS/LES (Large Eddy Simulation) [5,6] approaches. Calculations for this projectile were performed using a scalable parallel Navier-Stokes flow solver, CFD++.[7,8] CFD++ flow solver is a multipurpose, unstructured, 3-D, implicit, Navier-Stokes solver. It provides scalable performance on a number of parallel computer architectures and includes 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 U.S. Army. Numerical solutions must capture with high fidelity all the flow structures associated the microjet interactions in a time-dependent fashion. It is imperative that these flow structures near the "tiny" microjets be modeled accurately with particular attention to turbulence modeling so as not to dissipate the developing flow structure in the immediate vicinity of the jet due to numerical issues. Modeling of these microjets requires tremendous grid resolution coupled with highly specialized boundary conditions for the jet activation and the use of a hybrid RANS/LES approach permitting local resolution of the unsteady turbulent flow with high fidelity.

COMPUTATIONAL METHODOLOGY

Three-dimensional (3-D) time-dependent Reynolds-averaged Navier-Stokes (RANS) equations are solved using the finite volume method [7]:

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

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 [8] and higher order hybrid RANS/LES [6] turbulence models were used for the computation of turbulent flows. The hybrid RANS/LES approach based on Limited Numerical Scales (LNS) is well suited to the simulation of unsteady flows and contains no additional empirical constants beyond those appearing in the original RANS and LES sub-grid models. With this method a regular RANS-type grid is used except in isolated flow regions where denser, LES-type mesh is used to resolve critical unsteady flow features. 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. In addition, special jet boundary conditions were developed and used for numerical modeling of microjets. Grid was actually moved to take into account the spinning motion of the projectile.

PARALLEL COMPUTATIONAL ISSUES

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 MPI, PVM, 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 Silicon Graphics O3K

(400 MHz), IBM SP P3 (375 MHz), IBM SP P4(1.7GHz), and 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.

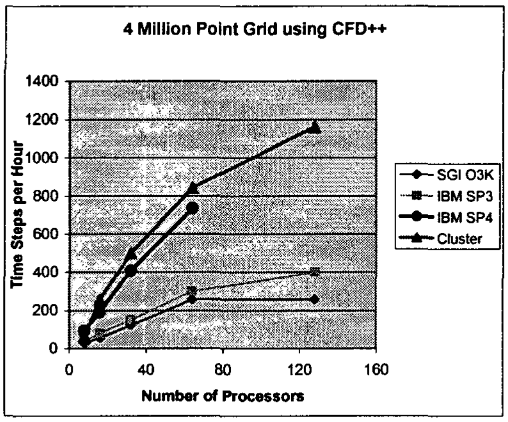

Figure 1. Parallel Speedups (4-million grid).

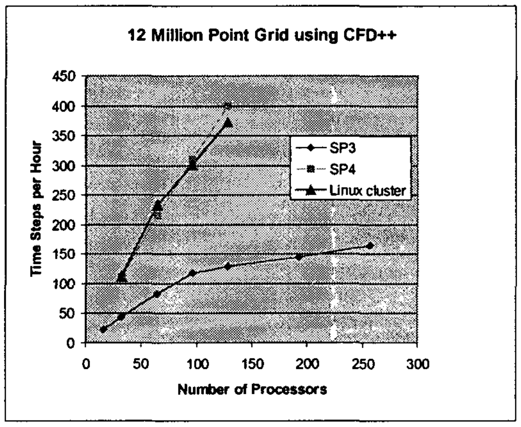

Figure 2. Parallel Speedups (12-million grid).

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.

RESULTS

Time-accurate unsteady numerical computations using advanced viscous Navier-Stokes methods were performed to predict the flow field and aerodynamic coefficients on two projectiles. Limited experimental data [10,11] exists and was used to validate the unsteady CFD results.

A projectile used in this study is a 1.8-caliber ogive-cylinder configuration (see Figure 3). The first step here was to obtain converged solution for the projectile without the jet. Converged jet-off solution was then used as the starting condition for the computation of time-accurate unsteady flow field for the projectile with microjets. The jet conditions were specified at the exit of the jet for the unsteady (sinusoidal variation in jet velocity) jets. The jet width was 0.32 mm, the jet slot half-angle was 18°, and the peak jet velocitiy used was 31 m/s operating at a frequency of 1000 Hz. Numerical computations have been made for these jet cases at subsonic Mach numbers, $M = 0.11$, and at an angle of attack, $\alpha = 0°$. The unsteady simulation took thousands of hours of CPU time on an IBM SP4 and Linux Cluster parallel computers running with 32–64 processors.

Many flow field solutions resulting from the simulation of a large number of microjet operations were obtained and saved at regular intermittent time-intervals to gain insight into the physical phenomenon resulting from the microjet interactions. The unsteady jets were found to break up the shear layer coming over the step in front of the base of the projectile. This in turn substantially altered the flow field (making it highly unsteady) both near the jet and in the wake region, resulted in asymmetric pressure distributions, and produced the required forces and moments even at zero degree angle-of-attack. The particles emanating from the jet interact with the wake flow (Fig. 4) making it highly unsteady. More importantly, the break up of the shear layer is clearly evidenced by the particles clustered in regions of flow gradients or vorticity.

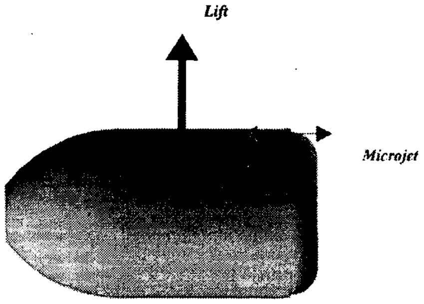

Figure 3. Projectile geometry.

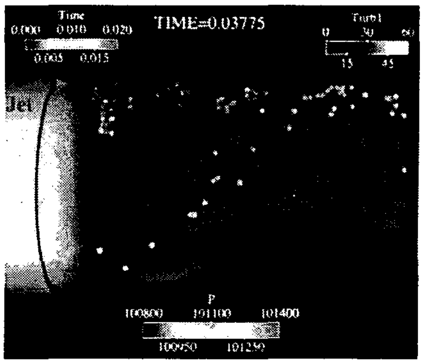

Figure 4. Particle Traces, M = 0.11, α = 0°.

These results were first obtained for a non-spinning case where microjet was activated all the time. For realistic situations (e.g. a spinning case), the jet needs to be activated only for part of the spin cycle and is off for the remainder of the spin cycle. Such an on-off jet actuation was simulated for the non-spinning case where the jet was on for 40 ms and off for 120 ms. The jet actuation was repeated for more cycles subsequently. Computed result for this case is shown in Fig. 5 and is compared with the experimental data obtained at Georgia Tech Research Institute (GTRI). Both results show the same trend of the lift force generated due to the jet actuation. When the jet is off, the mean lift force is zero and the usual wake oscillations that are very similar between the computation and experiment can be seen.

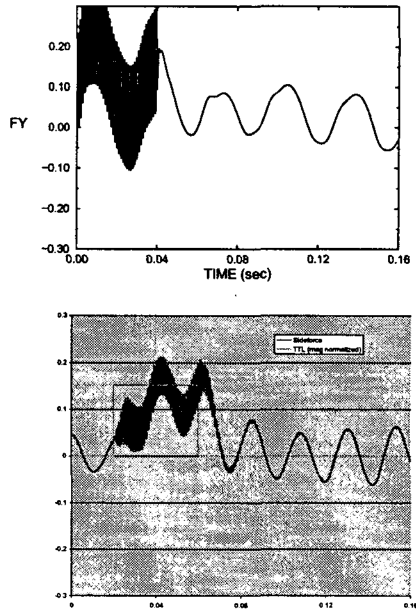

Figure 5. Lift force distribution, computation (top), experiment (bottom).

The net lift force (Fy) was determined from the actual time histories of the highly unsteady lift force. An example of the time-dependent aerodynamic forces resulting from the jet interaction for the non-spinning case and computed with the new hybrid RANS/LES turbulence approach is shown in Fig. 6. Results were obtained both with and without the cavity modeled and the effect of the cavity modeling was assessed. Similar results (not shown here) were also obtained for an actual spinning case (here the jet is turned on and off during spin cycle).

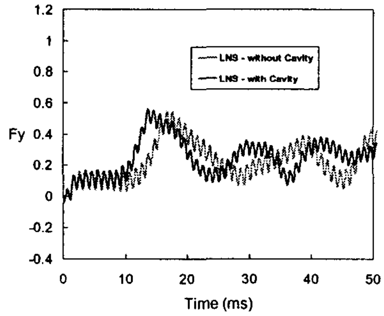

Figure 6. Time history of lift force.

Computations have also been carried out on another small caliber configuration at a high subsonic speed, Mach = 0.76 and α = 0. Results have been obtained for a micro-combustion-jet located in the cylinder section of projectile. Figures 7 and 8 show the computed Mach number contours. Fig 7 shows the mach contours in the vicinity of the projectile including the low speed flow in the near wake. An expanded view of the computed contours near the microjet (Fig. 8) shows the extent of microjet interaction with the free stream flow as well as the upstream and downstream effects. Other jet locations are also being considered and results for these cases will be included in the full paper. The effect of jet conditions has also been studied. An example of the results illustrating the effect of different jet pressures is shown in Fig. 9. Increasing the jet pressures clearly produces larger extent of jet interaction effects both upstream and downstream of the jet location. These changes strongly affect the level of MAFC force that is produced by the microjets for control of the projectile.

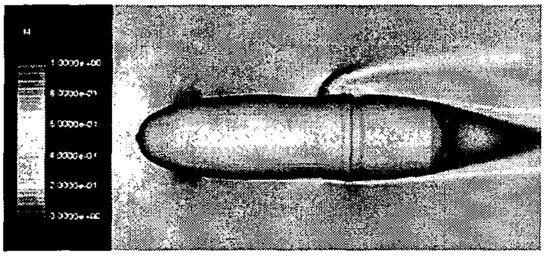

Figure 7. Mach contours.

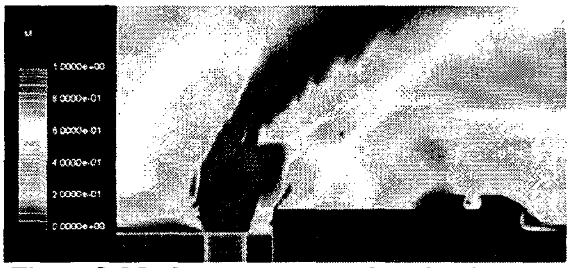

Figure 8. Mach contours near the microjet.

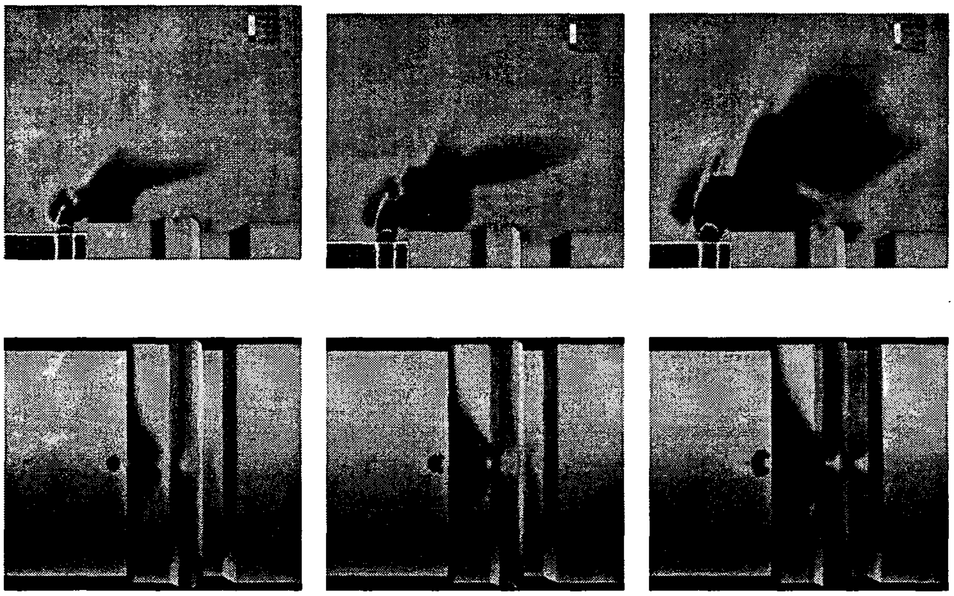

Figure 9. Pressure contours for different jet pressures 3, 6, and 9 atm (left to right).

CONCLUDING REMARKS

This paper describes a computational study undertaken to determine the aerodynamic effect of flow control in the afterbody regions of spinning projectiles at subsonic and low transonic speeds using a scalable unstructured flow solver on various parallel computers such as the IBM SP4 and Linux Cluster. Advanced scalable Navier-Stokes computational techniques were used to obtain numerical solutions for both steady and unsteady aerodynamic flow fields. High parallel efficiency is achieved for both steady and time-accurate unsteady cases. Numerical results show the effect of the jet on the flow field and on the aerodynamic coefficients for the unsteady jet interaction flow fields. The unsteady jet is shown to substantially alter the flow field both near the jet and the base region of the projectile that in turn affects the forces and moments even at zero degree angle of attack. The results have shown the potential of high performance computing computational fluid dynamics simulations on parallel machines to provide insight into the jet interaction flow fields leading to improve designs.

REFERENCES

1. J. Sahu, K. R. Heavey, and E. N. Ferry, "Computational Fluid Dynamics for Multiple Projectile Configurations", *Proceedings of the 3rd Overset Composite Grid and Solution Technology Symposium*, Los Alamos, NM, October 1996
2. J. Sahu, K. R. Heavey, and C. J. Nietubicz, "Time-Dependent Navier-Stokes Computations for Submunitions in Relative Motion", *6th International Symposium on Computational Fluid Dynamics*, Lake Tahoe, NV, September 1995
3. B. L. Smith and A. Glezer, "The Formation and Evolution of Synthetic Jets." *Journal of Physics of Fluids*, vol. 10, No. 9, September 1998
4. M. Amitay, 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
5. S. Arunajatesan and N. Sinha, "Towards Hybrid LES-RANS Computations of Cavity Flowfields", AIAA Paper No. 2000-0401, January 2000
6. P. Batten, 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
7. O. Peroomian, S. Chakravarthy, S. Palaniswamy, and U. Goldberg, "Convergence Acceleration for Unified-Grid Formulation Using Preconditioned Implicit Relaxation." AIAA Paper 98-0116, 1998
8. U. Goldberg, O. Peroomian, and S. Chakravarthy, "A Wall-Distance-Free K-E Model With Enhanced Near-Wall Treatment" *ASME Journal of Fluids Engineering*, Vol. 120, 457-462, 1998
9. T. H. Pulliam and J. L. Steger, "98dOn Implicit Finite-Difference Simulations of Three- Dimensional Flow" *AIAA Journal*, vol. 18, no. 2, pp. 159–167, February 1982
10. C. Rinehart, J. M. McMichael, and A. Glezer, "Synthetic Jet-Based Lift Generation and Circulation Control on Axisymmetric Bodies." AIAA Paper No. 2002-3168
11. McMichael, J., GTRI, Private Communications.
12. J. Sahu, "Unsteady Numerical Simulations of Subsonic Flow over a Projectile with Jet Interaction" AIAA Paper 2003-1352, Reno, NV, 6-9 January 2003