

REPORT DOCUMENTATION PAGE

Form Approved
OMB NO. 0704-0188

Public Reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comment regarding this burden estimates or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188,) Washington, DC 20503.

1. AGENCY USE ONLY (Leave Blank)		2. REPORT DATE March 26, 2007	3. REPORT TYPE AND DATES COVERED Final Progress Report / March 13, 2006-December 12, 2006	
4. TITLE AND SUBTITLE Performance Benchmark for a Prismatic Flow Solver			5. FUNDING NUMBERS W911NF-06-1-0063	
6. AUTHOR(S) Bharat K. Soni and Yasushi Ito				
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) University of Alabama at Birmingham 701 20th St. S., AB 3 Birmingham, AL 35294-0103			8. PERFORMING ORGANIZATION REPORT NUMBER	
9. SPONSORING / MONITORING AGENCY NAME(S) AND ADDRESS(ES) U. S. Army Research Office P.O. Box 12211 Research Triangle Park, NC 27709-2211			10. SPONSORING / MONITORING AGENCY REPORT NUMBER 4 9 7 2 4 . 1 - E G	
11. SUPPLEMENTARY NOTES The views, opinions and/or findings contained in this report are those of the author(s) and should not be construed as an official Department of the Army position, policy or decision, unless so designated by other documentation.				
12 a. DISTRIBUTION / AVAILABILITY STATEMENT Approved for public release; distribution unlimited.			12 b. DISTRIBUTION CODE 	
13. ABSTRACT (Maximum 200 words) HPC Institute for Advanced Rotorcraft Modeling and Simulation (HI-ARMS) seeks the performance evaluation on various near-body/off-body spatial partitioning paradigm for the development of a computation engine for DoD's mission-critical design and analysis tasks. The purpose of this project is to benchmark the prismatic flow solver derived from the Tohoku University Aerodynamic Simulation (TAS) flow solver with a common test problem. The TAS code is an unstructured, implicit, compressible flow solver developed by Nakahashi et al. The full Reynolds-averaged Navier-Stokes (RANS) equations that retain the unsteady form are solved by a finite volume cell-vertex scheme. The control volumes are non-overlapping dual cells constructed around each node. The Harten-Lax-van Leer-Einfeldt-Wada (HLLW) Reimann solver is used for the numerical flux computations. The Lower-Upper-Symmetric Gauss-Seidel (LU-SGS) implicit method is used for time integration to reduce the computational time. A one-equation turbulence model by Goldberg and Ramakrishnan and the Spalart-Allmaras model are implemented to treat turbulent boundary layers. A grid reordering method is implemented for the cell-vertex scheme implementation for three-dimensional hybrid grids for the LU-SGS method. The deliverables are used as the basis for measuring the attributes of the prismatic flow solver and its potential to meet long-term HI-ARMS objectives and program requirements.				
14. SUBJECT TERMS Prismatic flow solver Reynolds-averaged Navier-Stokes (RANS) equations			15. NUMBER OF PAGES 4	
			16. PRICE CODE	
17. SECURITY CLASSIFICATION OR REPORT UNCLASSIFIED	18. SECURITY CLASSIFICATION ON THIS PAGE UNCLASSIFIED	19. SECURITY CLASSIFICATION OF ABSTRACT UNCLASSIFIED	20. LIMITATION OF ABSTRACT UL	

NSN 7540-01-280-5500

Standard Form 298 (Rev.2-89)
Prescribed by ANSI Std. Z39-18
298-102

Enclosure 1

Performance Benchmark for a Prismatic Flow Solver

Final Progress Report

Statement of the Problem Studied

With the simultaneous development of high performance computing (HPC) and improved mathematical/numerical algorithms, computational fluid dynamics (CFD) has rapidly emerged as an essential tool for engineering analysis and design. This trend has drastically changed the manner in which the underlying principles of science and engineering are applied to research, design, and development. However, improvement in accuracy, building confidence, increased throughput, and reduction in cost are the four key challenges to the use of CFD for a more significant role in the engineering design. In particular, the CPU time required to obtain a reliable solution of today's CFD solvers is too long for it to be useful in the short turn-around time in the design process. Solution throughput in the generation of aerodynamics, propulsion, and fluid dynamics simulations that involve parametric and sensitivity studies of design and analysis must be significantly improved at an affordable cost. HPC Institute for Advanced Rotorcraft Modeling and Simulation (HI-ARMS) seeks the performance evaluation on various near-body/off-body spatial partitioning paradigm for the development of a computation engine for DoD's mission-critical design and analysis tasks.

The Tohoku University Aerodynamic Simulation (TAS) flow solver, an unstructured, implicit, compressible flow solver originally developed by Nakahashi and Sharov, has been successfully used for many complex numerical simulations [1-3]. The full Reynolds-averaged Navier-Stokes (RANS) equations that retain the unsteady form are solved by a finite volume cell-vertex scheme. The control volumes are non-overlapping dual cells constructed around each node. The Harten-Lax-van Leer-Einfeldt-Wada (HLLEW) Riemann solver [4] is used for the numerical flux computations. The Lower-Upper-Symmetric Gauss-Seidel (LU-SGS) implicit method [1] is used for time integration to reduce the computational time. A one-equation turbulence model by Goldberg and Ramakrishnan [5] and the Spalart-Allmaras model [6] are implemented to treat turbulent boundary layers. A grid reordering method is used for the cell-vertex scheme implementation for three-dimensional hybrid grids for the LU-SGS method that achieved approximately 30 percent improvement compares to the method without reordering [1]. This flow solver accepts hybrid meshes, which are comprised of tetrahedra, prisms and pyramids, but it can be converted to a prism-only flow solver.

The purpose of the project is to benchmark the prismatic flow solver derived from the TAS flow solver by Nakahashi *et al.* with a common test problem. The deliverables are used as the basis for measuring the attributes of the prismatic flow solver and its potential to meet long-term HI-ARMS objectives and program requirements. We have performed computational simulations for a given configuration and flow conditions.

Benchmark Case

One benchmark case was analyzed. The test problem used in this evaluation study was the low-speed, viscous flow, over a sphere with a Reynolds number of 1 million. The development team in this activity was given the following:

- Surface triangulation of the sphere sufficient to accurately define the problem geometry and represent the spatial resolution intended for required tests.
- Required initial spacing for the volume grid ($y^+ = 1$).
- Distance of far-field outer boundary from wing surface.
- Free-stream conditions.
- Required temporal duration of the simulation.
- Maximum allowable time-step size that can be used in simulation.

A prismatic volume mesh of the problem geometry was provided by Dr. William Chang at NASA Ames Research Center (ARC) for use in this study.

References

1. Sharov, D. and Nakahashi, K., “Reordering of Hybrid Unstructured Grids for Lower-Upper Symmetric Gauss-Seidel Computations,” *AIAA Journal*, Vol. 36, No. 3, 1998, pp. 484-486.
2. Nakahashi, K., Togashi, F., Sharov, D., “An Intergrid-Boundary Definition Method for Overset Unstructured Grid Approach,” *AIAA Journal*, Vol. 38, No. 11, 2000, pp. 2077-2084.
3. Nakahashi, K., Ito, Y. and Togashi, F., “Some Challenges of Realistic Flow Simulations by Unstructured Grid CFD,” *International Journal for Numerical Methods in Fluids*, Vol. 43, Issue 6-7, 2003, pp. 769-783.
4. Obayashi, S. and Guruswamy, G. P., “Convergence Acceleration of an Aeroelastic Navier-Stokes Solver,” *AIAA Journal*, Vol. 33, No. 6, 1995, pp. 1134-1141.
5. Goldberg, U. C. and Ramakrishnan, S. V., “A Pointwise Version of Baldwin-Barth Turbulence Model,” *Computational Fluid Dynamics*, Vol. 1, 1993, pp.321-338.
6. Spalart, P. and Allmaras, S., “A One-Equation Turbulence Model for Aerodynamic Flows,” AIAA Paper 92-0439, 1992.

Summary of the Most Important Results

The prismatic flow solver was successfully derived from the sequential version of the original TAS flow solver. A volume mesh file format converter was developed to use the flow solver with the prismatic volume meshes provided by Dr. Chang of the NASA ARC.

Table 1. CPU times used in each step of the steady-state and time-accurate modes with one Intel Xeon 2.4GHz processor: a prismatic mesh for a sphere that has 1,259,766 nodes; the time-accurate simulation with 3 Newton sub-iterations

	Steady-state mode	Time-accurate mode
CPU time [s]	89.2 to 89.7	274 to 275

We have provided the following data/results to the HI-ARMS:

- Source codes and associated make files of
 - the prismatic flow solver
 - a wall-distance calculator (used with the Spalart-Allmaras turbulence model)
 - the volume mesh file format converter
- Volume meshes used to carry out required simulations in TAS flow solver format
- CPU times used in each step of the steady-state and time-accurate modes (Table 1).

- A brief document that summarizes the numerical methods and capabilities of the solver.
- A user manual of the flow solver and the volume mesh file converter

Publications and Technical Reports Supported under This Contract

None

Participating Scientific Personnel

- Dr. Bharat K. Soni, Dept. of Mechanical Engineering, University of Alabama at Birmingham
- Dr. Yasushi Ito, Dept. of Mechanical Engineering, University of Alabama at Birmingham
- Dr. Kazuhiro Nakahashi, Dept. of Aerospace Engineering, Tohoku University, Japan (subcontractor)

Report of Inventions

None

Bibliography

1. Mitsuhiro Murayama, Yasushi Ito, Kazuhiro Nakahashi, Kisa Matsushima and Toshiyuki Iwamiya, "Viscous Flow Computations of Aircraft with Changing Control Surface Deflection Using Unstructured Dynamic Meshes," *International Journal for Numerical Methods in Fluids*, Vol. 52, Issue 8, November 2006, pp. 925-940.
2. Fumiya Togashi, Yasushi Ito, Kazuhiro Nakahashi and Shigeru Obayashi, "Overset Unstructured Grids Method for Viscous Flow Computations," *AIAA Journal*, Vol. 44, No. 7, July 2006, pp.1617-1623.
3. Fumiya Togashi, Yasushi Ito, Kazuhiro Nakahashi and Obayashi, S. "Extensions of Overset Unstructured Grids to Multiple Bodies in Contact," *Journal of Aircraft*, Vol. 43, No. 1, January-February 2006, pp. 52-57.
4. Fumiya Togashi, Takeshi Fujita, Yasushi Ito, Kazuhiro Nakahashi and Yoshikazu Makino, "CFD Computations of NAL Experimental Airplane with Rocket Booster Using Overset Unstructured Grids," *International Journal for Numerical Methods in Fluids*, Vol. 48, Issue 8, July 2005, pp. 801-818.
5. Yasushi Ito and Kazuhiro Nakahashi, "Improvements in the Reliability and Quality of Unstructured Hybrid Mesh Generation," *International Journal for Numerical Methods in Fluids*, Vol. 45, Issue 1, May 2004, pp. 79-108.
6. Roy P. Koomullil, David S. Thompson, and Bharat K. Soni, "Iced airfoil simulation using generalized grids," *Applied Mathematics and Computation*, Vol. 46, 2003, pp. 319-330.
7. Bharat K. Soni, "High Fidelity Computational Simulations and Enabling Technologies Current Status and Future Directions," Invited Paper, TCN (Technologie per il Calco Numerico) Conference, October 2003.

8. Fumiya Togashi, Yasushi Ito, Mitsuhiro Murayama, Kazuhiro Nakahashi and Takuma Kato, "Numerical Simulation of Flow around an Insect in Flight," *Computational Fluid Dynamics Journal*, Vol. 12, No. 1, April 2003, pp. 98-106.
9. Takeshi Fujita, Yasushi Ito, Kazuhiro Nakahashi and Toshiyuki Iwamiya, "Computational Fluid Dynamics Evaluation of National Aerospace Laboratory Experimental Supersonic Airplane in Ascent," *Journal of Aircraft*, Vol. 39, No. 2, March 2002, pp. 359-364.
10. Fumiya Togashi, Kazuhiro Nakahashi, Yasushi Ito, Toshiyuki Iwamiya and Yuichi Shimbo, "Flow Simulation of NAL Experimental Supersonic Airplane/Booster Separation Using Overset Unstructured Grids," *Computers & Fluids*, Vol. 30, Issue 6, July 2001, pp. 673-688.
11. Bharat K. Soni, Roy P. Koomullil, David S. Thompson, and Hugh Thornburg, "Solution Adaptive Grid Strategies Based on Point Redistribution," *Comput. Methods Appl. Mech Engrg.*, Vol. 189, 2000, pp. 1183-1204.
12. Bharat K. Soni, "Grid Generation: Past, Present and Future," *Journal of Applied Numerical Mathematics*, Vol. 32, Issue 4, 2000, pp.261-269.
13. Zhilin Li, and Bharat K. Soni, "Fast and Accurate Numerical Approaches for Stefan Problems and Crystal Growth." *The Journal of Numerical Heat Transfer, Part B: Fundamentals*, Vol. 35, Issue 4, 1999, pp. 461-484.