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 (HLLEW) 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.
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) Reimann 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

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>
<thead>
<tr>
<th>CPU time [s]</th>
<th>Steady-state mode</th>
<th>Time-accurate mode</th>
</tr>
</thead>
<tbody>
<tr>
<td>89.2 to 89.7</td>
<td>274 to 275</td>
<td></td>
</tr>
</tbody>
</table>

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


