Highly Parallel Implicit CFD applied to GPU Computing with Unstructured Tetrahedral Grids

Matthew Smith* **and Yi-Hsin Lin**

* Department of Mechanical Engineering, NCKU, Taiwan. Email: msmith@mail.ncku.edu.tw

Abstract

The Finite Volume Method (FVM) has become one of the cornerstone techniques for the numerical solution to the Euler and Navier-Stokes equations. These approaches, falling within the larger family of Computational Fluid Dynamics (CFD) solvers, are generally computationally intensive for three dimensional problems. In order to reduce the computational time required to obtain a solution, parallel computing is often employed to share the computational load. In this instance, we employ Graphics Processing Units (GPUs) to accelerate the computation by sharing the workload across the large number of streaming cores on the GPU device. Due to the need for simulation of complex and often arbitrary three dimensional geometries, the presented solver is written for application on an unstructured tetrahedral mesh. In order to accelerate the transient computation, in addition to allowing a steady state solution approach, the solver is written for solution using an implicit scheme. The non-linear governing equations are used directly to form a residual function, which are then solved using the Newton-Raphson technique. The residual-Jacobian inverse is computed using the BiConjugate Gradient (or BiCG) method for each Newton-Raphson iteration. Each part of the computation is parallel – from the Jacobian computation, its associated inverse and the Newton-Raphson update stage. The results demonstrate a numerical dispersion associated with the use of an implicit solver which scales with the local CFL number, with larger transient time step sizes leading to increased dispersion. The performance and scalability of the approach is reviewed and compared to a more traditional explicit computation using the GPU.