8th. World Congress on Computational Mechanics (WCCM8)
5th. European Congress on Computational Methods in Applied Sciences and Engineering (ECCOMAS 2008)
June 30 – July 5, 2008
Venice, Italy

A HYBRID FINITE VOLUME/ELEMENT METHOD FOR LOW AND HIGH REYNOLDS NUMBER INCOMPRESSIBLE FLOW

***Shuangzhang Tu[1] and Shahrouz Aliabadi[2]**

[1] Jackson State University
P.O. Box 17098, Jackson, MS 39217, USA
shuangzhang.tu@jsums.edu

[2] Jackson State University
1230 Raymond Rd, MS 39204, USA
saliabadi@jsums.edu

Key Words: *Incompressible flows, hybrid finite volume/element method, unstructured flow solver.*

ABSTRACT

We have successfully developed an implicit hybrid finite volume/element solver for both low and high Reynolds number incompressible flows. The solver is based on the segregated pressure correction or projection method on staggered unstructured meshes. An intermediate velocity field is first obtained by solving the original momentum equations with the matrix-free implicit cell-centered finite volume method. The pressure Poisson equation is solved by the node-based Galerkin finite element method for an auxiliary variable. The auxiliary variable is used to update the velocity field and the pressure field. The pressure field is carefully updated by taking into account the velocity divergence field. This updating strategy can be rigorously proved to be able to eliminate the unphysical pressure boundary layer and are crucial for the correct temporal convergence rate (Figure 1). Our current staggered-mesh scheme is distinct from other conventional ones in that we store the velocity components at cell centers and the auxiliary variable at vertices. For high Reynolds number turbulent flows, we loosely couple the Spalart-Allmaras Detached Eddy Simulation (SA-DES) turbulence model. The same matrix-free finite volume method as the one used for momentum equations is used to solve the turbulence equation. We will address some important implementation issues for high-Reynolds number flows where highly stretched elements are typically used to ensure the solver's robustness. Several numerical examples will be presented to demonstrate the accuracy and efficiency of our current solver for both low (Figure 2) and high (Figure 3) Reynolds number flows.

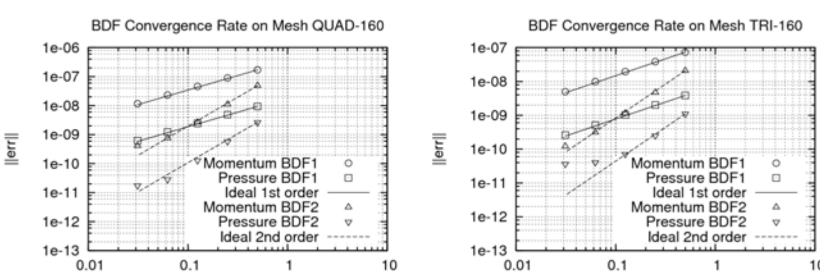

Figure 1. Rate of temporal convergence. Left: quadrilateral mesh. Right: triangular mesh.

Figure 2. Vorticity of a low Reynolds number flow (Re = 100).

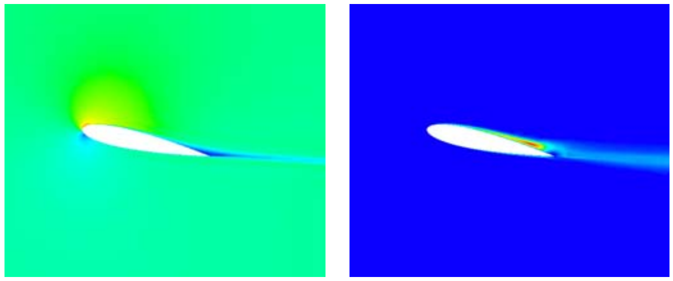

Figure 3. High Reynolds number turbulent flow (Re = 1.5e+6, AoA = 12 deg). Left: velocity magnitude field; right: turbulent viscosity field.

We will present the scalability and parallel performance of CaMEL flow solvers based on finite element and finite volume methods. The sustained computational speed is measured more than one teraflop using unstructured meshes with two billion tetrahedral elements on Cray X1E with 256 processors. Many 3D numerical examples will be presented including aerodynamics of Formula-1 race car. (see Figure 4)

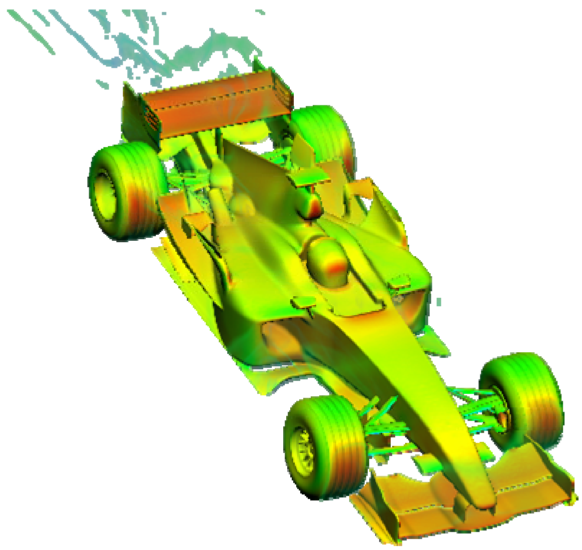

Figure 4. Pressure distribution on Formula-1 race car.

REFERENCES

[1] J.L. Guermond, P. Minev, and J. Shen. "An overview of projection methods for incompressible flows," *Computer Methods in Applied Mechanics and Engineering*, Vol. 195, pp. 6011–6045, (2006)

[2] S. Tu and S. Aliabadi, "Development of a hybrid finite volume/element solver for incompressible flows on unstructured meshes", *International Journal of Numerical Methods in Fluids*, Vol. 55, No. 2, pp. 177–203, (2007)