11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
July 20 - 25, 2014, Barcelona, Spain

A PARALLEL AGGLOMERATION MULTIGRID METHOD FOR THE ACCELERATION OF COMPRESSIBLE FLOW COMPUTATIONS ON 3D HYBRID UNSTRUCTURED GRIDS

Georgios N. Lygidakis[1]* and Ioannis K. Nikolos[2]

[1,2] Technical University of Crete , School of Production Engineering and Management, University Campus, Chania, GR-73100, Greece
[1] email: glygidakis@isc.tuc.gr
[2] email: jnikolo@dpem.tuc.gr

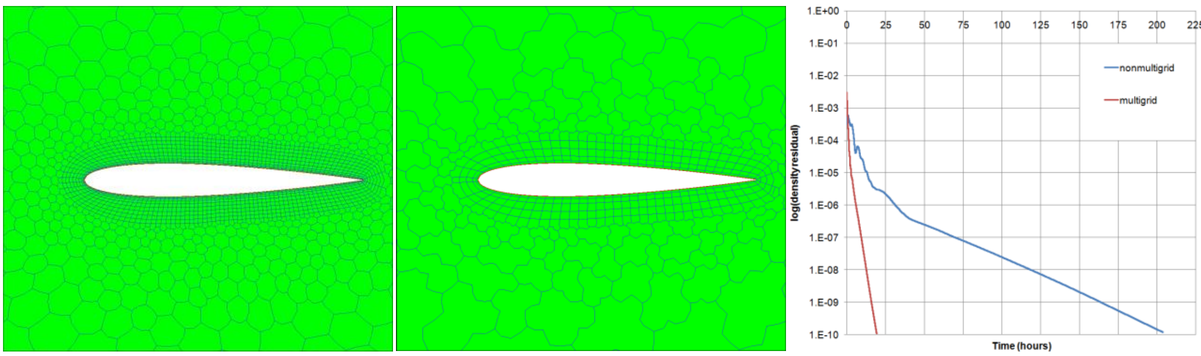
Key Words: *Compressible Flow, Multigrid, Isotropic/Directional Agglomeration, 3D Hybrid Unstructured Grids.*

During the last decades the 3D unstructured grids have become an important tool for Computational Fluid Dynamics (CFD) extending its applications to complex geometries, e.g. aircraft configurations. Despite their ability to discretize complex computational domains the corresponding solvers remain inferior in terms of efficiency compared to the structured mesh ones. One remedy to this shortcoming is the agglomeration multigrid method, based on the solution of the flow problem on successively coarser grids derived from the initial finest one through the agglomeration of the neighbouring control volumes [1, 2]. In this work a parallel agglomeration multigrid methodology is presented, developed to enhance an existing academic CFD code, named "Galatea", which employs the Reynolds-Averaged Navier-Stokes equations along with appropriate turbulence models to simulate inviscid and viscous laminar or turbulent compressible fluid flows.

The application of this multigrid method relies on the generation and the use of successively coarser grids derived by the fusion of the neighbouring control cells of the finer grid. The agglomeration is performed on a topology-preserving framework resembling the advancing front technique, while depending on the type of the initial finest grid (tetrahedral or hybrid) two main types of agglomerated meshes can be derived, an isotropic and a directional one [1, 2]. The procedure begins with the identification of the connection of every node to the computational boundaries, which will define its control volume agglomeration, e.g. a node belonging to three boundary-condition-type closures or three boundary slope discontinuities won't be agglomerated and will remain a singleton. Since the identification is complete, the main agglomeration process begins with the fusion of the viscous boundary control volumes, extends to the prismatic layers' ones (if exist) and continues to the rest interior and boundary volumes [2]. As this process is performed for a parallelized computational system, for every agglomeration level it initially takes into account only the "core" nodes of each partition; the "ghost" nodes of the overlapping area are then agglomerated, according to the fusion of their corresponding "core" nodes of the neighbouring partition. In Figure 1 the non-agglomerated (a) and the directionally agglomerated (b) volume grids of a rectangular wing with a NACA0012 airfoil are illustrated.

For the numerical solution of the flow and turbulence model equations the Full Approximation Scheme (FAS) is employed, incorporated in the Full Multigrid (FMG) scheme. The corrected equations are solved for the coarser grids similarly to the initial finest one, while the interaction between them is succeeded via the restriction of the variables and the fluxes from the finer to the coarser mesh as well as the prolongation of the variables' corrections from the coarser grid to the finer one. Considering the edge-based structure of the algorithm and the derived "virtual" edges of the coarser meshes the computation of their convective and their source terms is straightforward, implementing a spatially first order accurate scheme. For the computation of the velocity components' gradients and subsequently the calculation of the viscous fluxes, a nodal averaging technique is employed to the coarser levels; for the first (non-agglomerated) level the more accurate element-based edge-dual volume method is implemented. The FAS is achieved via a $V(v_1,v_2)$ cycle process, where v_1 denotes the number of iterative solutions before proceeding to the coarser level and v_2 the corresponding number after receiving the coarse grid correction; *V(1,0)* is employed for inviscid and laminar flows, while *V(3,3)* for turbulent ones.

The proposed algorithm has been validated against benchmark test cases, indicating the improvement of the computational efficiency with the implementation of the afore-mentioned technique, as it succeeded about *6* to *11* speed-up values depending on the type of the flow. In Figure 1(c) the convergence history for density, using the multigrid and the non-multigrid schemes, is illustrated for the laminar flow over a wing with a NACA0012 airfoil. The simulation was performed on a workstation with an AMD FX-8350 8-core Processor at *4.00* GHz, and a computational mesh consisted of about *320,000* nodes and divided in *4* partitions for parallelization.



(a) (b) (c)

Figure 1: Non-agglomerated volume grid of a wing with NACA0012 airfoil (a), directionally agglomerated volume grid (b) and density convergence history (c) for laminar flow over the same geometry.

REFERENCES

[1] G. Carre, L. Fournier and S. Lanteri, Parallel linear multigrid algorithms for the acceleration of compressible flow calculations, *Comput. Methods Appl. Mech. Engrg.*, Vol. **184**, pp. 427-448, 2000.

[2] H. Nishikawa and B. Diskin, Development and application of parallel agglomerated multigrid methods for complex geometries (AIAA 2011-3232), *20th AIAA Computational Fluid Dynamics Conference*, Honolulu, Hawaii, 27 - 30 June, 2011.