Adaptive remeshing for industrial unsteady CFD

- ADMOS 2015 -

A. Limare*,†, H. Borouchaki† and P. Brenner*

* Airbus Defence & Space
66, route de Verneuil, 78133 Les Mureaux, France
E-mails: alexandre.limare@astrium.eads.net
pierre.brenner@astrium.eads.net

†GAMMA3
Université de Technologie de Troyes
12, Rue Marie Curie, CS 42060, 10004 Troyes Cedex, France
Email : houman.borouchaki@utt.fr

ABSTRACT

FLUSEPA is an advanced simulation tool which performs a large panel of aerodynamic studies. It is the unstructured finite-volume solver developed by Airbus Defence & Space to calculate compressible, multidimensional, unsteady, viscous and reactive flows around bodies in relative motion. The numerical strategy in FLUSEPA is designed for highly compressible flow and keeps its accuracy regardless of the grid. According the desired accuracy, a second-order accurate shock-capturing scheme is generally used for RANS and URANS simulations and a fourth order accurate vortex-centred scheme is used for hybrid RANS/LES simulations[1].

In this paper we introduce an adaptive meshing approach to accurately represent unsteady flows in FLUSEPA. The meshing strategy is based on a multi-overlapping grid intersection which is conservative and allows to quickly and properly mesh 3D complex geometries. It can be seen as a CHIMERA strategy without interpolation. Each part of the bodies is meshed independently and immersed in background grids. This technique will be largely described for it differs from the commonly used interpolation-based CHIMERA methods.

The adaptive mesh construction is based on a simple 2-1 balanced octree approach. Different criteria for refinement will be tested: one relying on the Ducros[2] sensor, another is a basic hessian and the last one is based on the resolved kinetic energy. The aim of this study is to obtain a versatile module for industrial applications combining different criteria. The test case is a transonic turbulent flow around a square cylinder at a Mach number of 0.9 and Reynolds number of 4.10^5 based on the experimental conditions of Nakagawa[3]. Computations are carried out using a CFL of 0.8 and on a Cartesian grid. The study will be focused on the interaction between the von Karman eddy street and the shock wave.

REFERENCES

[1] G. Pont, P. Cinnella, J-C Robinet and P. Brenner, “Development of numerical schemes for hybrid RANS/LES modelling in an industrial CFD solver”, *AIAA Conference*, (2013).

[2] F. Ducros, V. Ferrand, F. Nicoud, C. Weber and D. Darracq, “Large eddy simulation of the shock/turbulence interaction”, *Journal of Computational Physics*, Vol. **152**, pp. 517-549, (1999).

[3] T. Nagakawa, “Vortex shedding behind a square cylinder in transonic flows”, *Journal of fluid Dynamics*, Vol. **178**, pp. 303–323, (1987).