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

TOWARDS A MODULAR APPROACH FOR UNSTRUCTURED SHOCK-FITTING

R. Pepe [1], A. Bonfiglioli[1], R. Paciorri[2], A. Lani[3], J. G. Mena[3] C. F. Olliver-Gooch[4]

[1] Università della Basilicata , 85100, Potenza, Italy aldo.bonfiglioli@unibas.it
[2] Università di Roma, 00184, Roma, Italy, renato.paciorri@uniroma1,it
[3] von Karman Institute, 1640 Sint-Genesius-Rode, Belgium, lani@vki.ac.be
[4] University of British Columbia, V6T 1Z4, Vancouver, Canada, cfog@mech.ubc.ca

Key words: *flow with shocks, shock-fitting, unstructured meshes.*

The *unstructured*-grid version of the shock-fitting technique made its first appearance in 2009 [1] and has been developed since then: it has been successfully applied to 2D steady flows featuring interacting shocks [2] and 3D steady flows with multiple, though not interacting shocks [3].
Unstructured shock-fitting retains features of both the "boundary"(algorithmically simple) and "floating"(topologically more versatile) variants of the algorithm that had been used in the structured grid setting: the fitted shock fronts are interior *boundaries* that are free to *float* over a background mesh which is locally re-meshed to adapt to the actual position of the shock front; a shock-capturing solver is used to discretize the governing PDEs in smooth regions of the flowfield.
The use of unstructured grids allows to relieve most of the algorithmic difficulties encountered in the structured grid setting whilst keeping the key feature of the "traditional"shock-fitting discretization, i.e. accuracy on coarse meshes.
Modularity is one of the key features of this newly developed unstructured shock-fitting algorithm. Indeed, since the fitted shock fronts behave as interior boundaries, the CFD solver can be used as a "black box"by the shock fitting algorithm and the same consideration applies to the algorithm that is used for local re-meshing around the shock-front. For instance, the technique had originally been developed using the in-house `eulfs` CFD solver [4] and different public domain codes for volumetric and surface remeshing, such as: triangle [5] in 2D and tetgen [6] and Yams [7] in 3D. Very recently, the CFD solver has been replaced with the COOLFluiD[8, 9] code developed at VKI. Thanks to the high modularity of the shock-fitting technique, this task has been accomplished at a very limited coding effort. Figure 1, which refers to the supersonic flow past a circular cylinder, shows pressure iso-contours computed by CoolFluid in shock-capturing and shock-fitting mode on the same coarse mesh. The comparison clearly highlights the improvements in solution quality that shock-fitting enables.

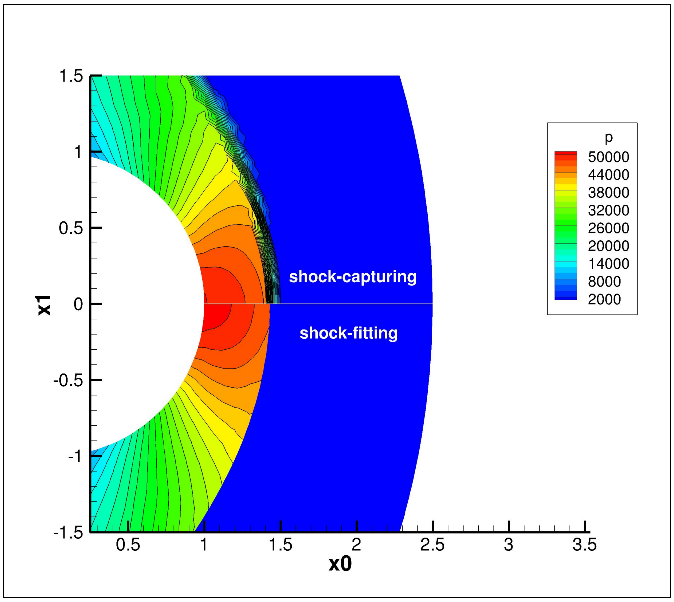

Figure 1: Supersonic flow past a 2D circular cylinder

In the near future, we plan to use the local remeshing technique specifically designed by Zaide and Olliver-Gooch [10] to remesh the shock region. This will allow to improve the performance of the shock-fitting technique and to broaden its range of application, especially in the field of 3D flows.

In the full paper the integration of the shock-fitting algorithm with the COOLFluid solver and the new mesh generation subroutines will be discussed and more examples of flow computations will be shown.

REFERENCES

[1] R. Paciorri and A. Bonfiglioli. A shock-fitting technique for 2D unstructured grids. *Computers and Fluids* Vol.38,no.3,715–726, 2009.

[2] R. Paciorri and A. Bonfiglioli. Shock interaction computations on unstructured, two-dimensional grids using a shock-fitting technique. *Journal of Computational Physics* Vol.230, no.8, 3155–3177, 2011.

[3] A. Bonfiglioli and M. Grottadaurea and R. Paciorri and F. Sabetta. An unstructured, three-dimensional, shock-fitting solver for hypersonic flows. *Computers and Fluids* Vol.73,no.0,162–174, 2013.

[4] A. Bonfiglioli. Fluctuation splitting schemes for the compressible and incompressible Euler and Navier-Stokes equations. *International Journal of Computational Fluid Dynamics* Vol.14, no.1, 21–39, 2000 .

[5] J. R. Shewchuk. Triangle: Engineering a 2D Quality Mesh Generator and Delaunay Triangulator. *Lecture Notes in Computer Science* Vol.1148,203–222, 1996.

[6] H. Si. TetGen a quality tetrahedral mesh generator and a 3D Delaunay triangulator. 2011. http://tetgen.berlios.de/.

[7] P.J. Frey. Yams: a fully automatic adaptive isotropic surface remeshing procedure. *Rapport technique n° 0252*, Institut National de Recherche en Informatique et en Automatique (I.N.R.I.A.) ISSN 0249-0803, 2001, http://www.ann.jussieu.fr/ frey/software.html.

[8] A. Lani and T. Quintino and D. Kimpe and H. Deconinck, *The COOLFluiD Framework - Design Solutions for High-Performance Object Oriented Scientific Computing Software*, ICCS 2005, Atlanta (GA), LNCS 3514, Vol.1, pp. 281-286, Springer-Verlag, 2005.

[9] A. Lani and N. Villedieu and K. Bensassi and L. Kapa and M. Panesi and M. S. Yalim , *COOLFluiD: an open computational platform for multi-physics simulation*, 21st AIAA CFD Conference, AIAA 2013-2589, San Diego, June 2013.

[10] D. W. Zaide and C. F. Ollivier-Gooch. Inserting a Curve into an Existing Two Dimensional Unstructured Mesh. *Proceedings of the 22nd International Meshing Roundtable* ,93-107, 2014.