XX Fluid Mechanics Conference KKMP2012,
Gliwice, 17-20 September 2012

STRUCTURE MESH GENERATOR WITH ANISOTROPIC ADAPTATION FOR BOUNDARY LAYER REGION

Piotr SZAŁTYS[†1], Jerzy MAJEWSKI[†2], Jacek ROKICKI[†3]
[†]Institute of Aeronautics and Applied Mechanics, Warsaw University of Technology
24 Nowowiejska Str., 00-665 Warsaw, Poland
[1]*pszaltys@meil.pw.edu.pl*, [2]*jmajewski@meil.pw.edu.pl*, [3]*jack@meil.pw.edu.pl*

Key words: structure mesh generator, boundary layer mesh, anisotropic mesh adaptation

ABSTRACT

This paper is related to the problems of grid generation and anisotropic adaptation for the flows with high Reynolds number. In such flows one of the critical issues is to create a proper mesh in a boundary layer region. Due to very small thickness of boundary layer in turbulent flows a proper spacing in the direction normal to the surface should be applied. Resultant cells are characterized by large aspect ratio. It is preferred by solvers to use the discretization given by quadrilateral or prismatic elements in such cases.

Presented algorithm consists of several steps. Initial grid is generated with user defined spacing then the flow problem is solved. In the next step the error estimator is processing the solution in order to obtained better quality spacing for the new grid. Estimator is based on mixed gradient – hessian approach [2]. As a result estimator computes a metric field which is used to create a new mesh. Structural mesh generator is using the frontal algebraic method [1]. The thickness of following layers is defined by metric field in direction normal to the surface. In order to stop the generation of structural mesh the several criteria is used: maximum number of layer, maximum aspect ratio, maximum height of elements. Finally a boundary of resulting structured mesh is extracted.

In the last step the unstructured grid generator is applied in order to fill the domain bounded by the extracted boundary of the structural mesh and a far field surface. Than the both grids are coupled together. Resulting grid is used by the solver in order to obtain a solution in the next iteration of adaption.

Described algorithm was applied in two dimensional case of laminar flow around NACA0012 airfoil. The laminar flow was chosen in order to obtain a relatively thick boundary layer which is better visible on the solution diagrams. The final hybrid grid with obtained velocity field after four iterations of adaptation is showed on Figure 1. The two-dimensional turbulent flow was also computed for the high lift multi-element airfoil L1T2. The velocity field with final grid after eight iteration of adaptation is showed on Figure 2. The adaptation algorithm was also applied in three-dimensional case. An example of initial structured mesh for ONERA m6 airfoil is showed on Figure 3.

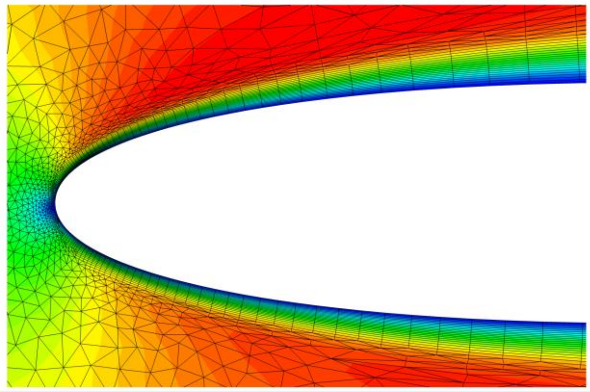

Figure 1. Hybrid grid with velocity field, laminar flow. NACA0012

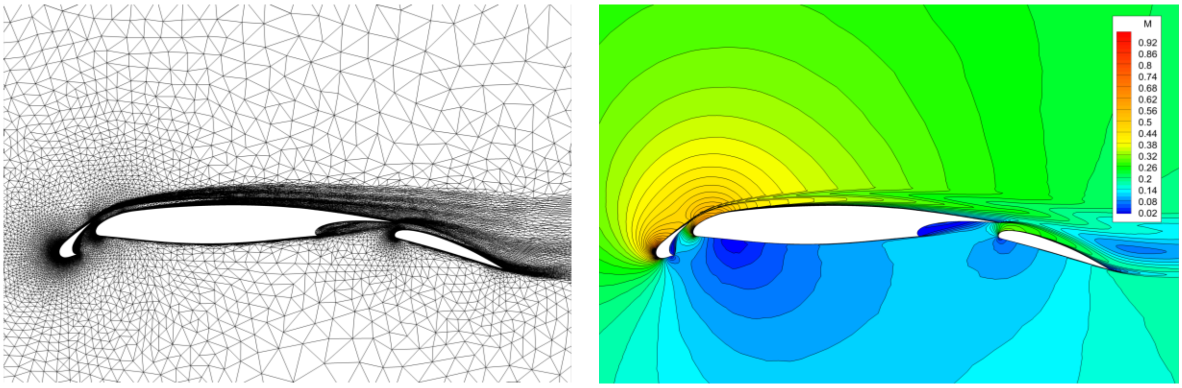

Figure 2. Grid and Mach number field for L1T2 airfoil after 8 adaptation steps.

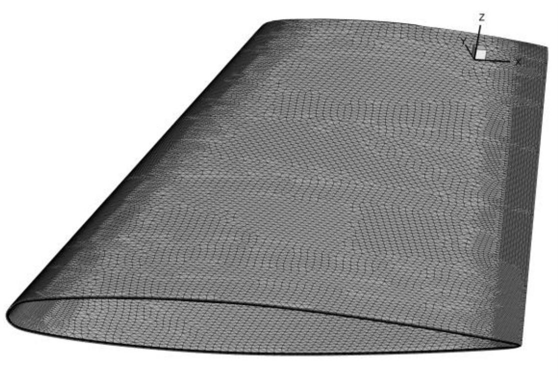

Figure 3. Inital prism mesh in boundary layer region. ONERA M6

References

[1] Athanasiadis., A. (2004): *Three-Dimensional Hybrid Grid Generation with Application to High Reynolds Number Viscous Flows*., PhD thesis, Université Libre de Bruxelles, Chaussée deWaterloo, 72, 1640 Rhode-St-Genèse, Belgium

[2] Majewski J., Rokicki. J. (2010): *Anisotropic mesh adaptation in the presence of complex boundaries*. In Kroll N., Bieler H., Deconinck H., Couaillier V., van der Ven H., and Sorensen K., editors, ADIGMA - *A European Initiative on the Development of Adaptive Higher-Order Variational Methods for Aerospace Applications*, volume 113 of Notes on Numerical Fluid Mechanics and Multidisciplinary Design, pages 441–453. Springer Berlin / Heidelberg, 2010. 10.1007/978-3-642-03707-8 31.