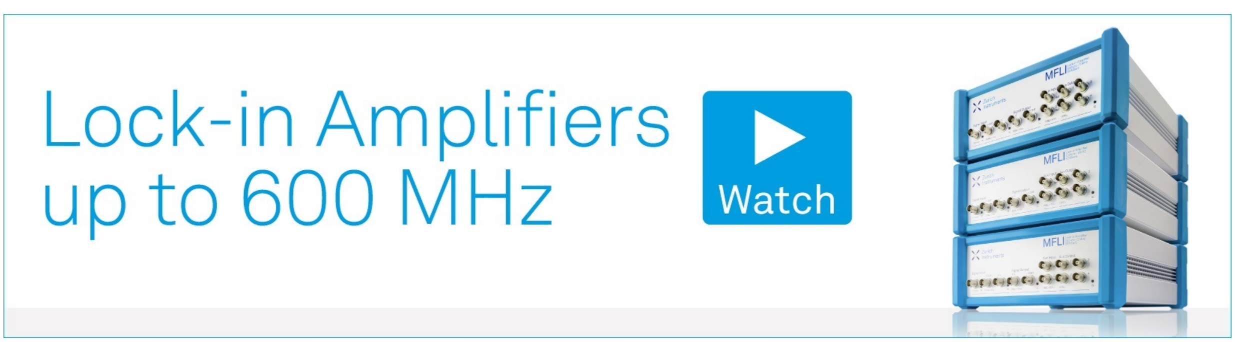
Lift coefficient calculation using a geometric/solution adaptive Navier Stokes solver on two-dimensional cartesian grids for compressible and turbulent flows

Cite as: AIP Conference Proceedings **1889**, 020018 (2017); https://doi.org/10.1063/1.5004352
Published Online: 25 September 2017

Emre Kara, Ahmet İhsan Kutlar, and Mehmet Haluk Aksel



Lift Coefficient Calculation Using a Geometric/Solution Adaptive Navier Stokes Solver on Two-Dimensional Cartesian Grids for Compressible and Turbulent Flows

Emre Kara[1,a)], Ahmet İhsan Kutlar[1,b)] and Mehmet Haluk Aksel[2,c)]

[1]*Mechanical Engineering Department, Faculty of Engineering, University of Gaziantep, Gaziantep, Turkey*
[2] *Mechanical Engineering Department, Faculty of Engineering, Middle East Technical University, Ankara, Turkey*

[a)]Corresponding author: emrekara@gantep.edu.tr
[b)]aikutlar@gantep.edu.tr
[c)]aksel@metu.edu.tr

Abstract. In this study, two-dimensional geometric and solution adaptive refinement/coarsening scheme codes are generated by the use of Cartesian grid generation techniques. In the solution of compressible, turbulent flows one-equation Spalart-Allmaras turbulence model is implemented. The performance of the flow solver is tested on the case of high Reynolds number, steady flow around NACA 0012 airfoil. The lift coefficient solution for the airfoil at a real-life-flight Reynolds number is compared with the experimental study in literature.

INTRODUCTION

In this study, two-dimensional geometric and solution adaptive refinement/coarsening scheme codes are generated by the use of Cartesian grid generation techniques [1]. The main program is named **GeULER-NaTURe** (cartesian-**G**rid-generator-with-**eULER**-**Na**vier-Stokes-**TUR**bulent-flow-solv**e**r). In the solution of compressible, turbulent flows one-equation Spalart-Allmaras turbulence model is implemented. The performance of GeULER-NaTURe flow solver [2] is tested on the case of high Reynolds number, steady, two-dimensional flow around NACA 0012 airfoil. GeULER-NaTURe lift coefficient solution for NACA 0012 airfoil at a fairly high and real-life-flight Reynolds number of $9x10^6$ and Mach number of 0.799 at the corrected angle-of-attack of 2.26º (corrected for interference of the wind tunnel wall) is compared with the experimental study in literature.

A SHORT LITERATURE SURVEY

There are lots of scientific studies regarding force coefficient calculations using Cartesian grid techniques. Some of the recent ones are as follows:

Sang and Yu [3] analyzed numerically high-lift aerodynamics of wing/body model with omni-tree Cartesian grids. This search described the work of the CFD improvement in the numerical simulation of high-lift configuration with respect to the Cartesian grid method and the Euler flow solver. The multi-zone technique reduced the difficulties of generating Cartesian grid to a large extent, while the face-to-face technique greatly simplified the numerical simulation of geometry discontinuity.

Hashimoto et al. [4] developed an automatic grid generator called HexaGrid that produces Cartesian/prism hybrid grids in three-dimensional space. The target of Hexagrid was to predict aerodynamic coefficients, i.e. lift and drag. The Cartesian grid part of the generator uses refinement box (similar to box adaptation applied in the current

36th Meeting of Departments of Fluid Mechanics and Thermodynamics
AIP Conf. Proc. 1889, 020018-1–020018-5; https://doi.org/10.1063/1.5004352
Published by AIP Publishing. 978-0-7354-1572-0/$30.00

project), solver part stored data in a structure called Alternating Digital Tree (ADT) which is the common name of 2^n-tree data structures. Grid smoothing and cell splitting Cartesian methods were employed for increased grid quality. The Euler/full Navier-Stokes equations were solved on the grid by cell-vertex FVM. Numerical flux computations were accomplished with the help of Harten-Lax-van Leer-Einfeldt-Wada (HLLEW) method. Time integration was carried by Lower/Upper Symmetric Gauss Seidel (LU-SGS) method. Limiters for the second-order accurate realization were Venkatakrishnan Limiter and Unstructured MUSCL scheme (U-MUSCL). Second order turbulence model used was Spalart-Allmaras model.

GOVERNING EQUATIONS

Considering an arbitrary control volume V, the conservative integral form of Reynolds-Averaged Navier-Stokes equation closed with Spalart-Allmaras (SA) turbulence model is generated using Gauss divergence theorem as follows:

$$\frac{\partial}{\partial t}\int_V \boldsymbol{Q}\mathrm{d}V + \int_A (\mathbf{F}\cdot\mathbf{n})\mathrm{d}A = \int_A (\mathbf{G}\cdot\mathbf{n})\mathrm{d}A + \int_V \mathrm{S}\,\mathrm{d}V \tag{1}$$

where V is the cell volume, S is the source term, **F** and **G** are the inviscid and viscous flux vectors, respectively. **Q** represents the vector of any conserved variable such as density, ρ, x-, y- and z-velocity components, u, v and w, respectively, total energy, E or SA working variable, $\tilde{\nu}$.

It is a widespread practice to calculate lift coefficients, c_l, on the surfaces of an embedded boundary and compare them with the experimental results. The governing equation is shown in the following:

$$c_l = \frac{F_{norm}\cos\alpha - F_{axial}\sin\alpha}{\frac{1}{2}\rho_\infty {U_\infty}^2} \tag{2}$$

where F_{norm} is the normal force and F_{axial} is the axial force acting on the solid body. The explicit computation of the forces can be found in the study of Walsh [6]. Non-dimensional free stream values as initial guesses, denoted by subscript $_\infty$, are given below:

$$u_\infty{}' = M_\infty \sin\theta\,;\;\; v_\infty{}' = M_\infty \cos\theta\,;\; w_\infty{}' = 0; \qquad E_\infty{}' = \frac{1}{\gamma(\gamma-1)}\frac{{M_\infty}^2}{2},$$

$$T_\infty{}' = 1;\; \rho_\infty{}' = 1; \qquad p_\infty{}' = \frac{1}{\gamma}; \quad c_\infty{}' = \sqrt{\frac{p_f{}'\gamma}{\rho_f{}'}} = 1 \tag{3}$$

Appropriate boundary conditions are crucial for physically relevant solutions of RANS-SA equations. In external flows, there are two types of boundary conditions; far-field and solid wall. A far-field boundary condition is far away from the solid wall of concern and uniform conditions are introduced as far-field values. No-slip condition at the wall is required in the case of viscous and turbulent flows. These conditions are forced by using ghost-cells for each cut cell.

Far field boundary conditions may be applied by extrapolating the dependent variable $\tilde{\nu}_\infty{}'$ from the exterior cells (ghost cells). In this study, a trivial solution is applied for turbulence equations for fully-turbulent conditions:

$$\tilde{\nu}_\infty{}' = 0.1\nu_\infty{}' \tag{4}$$

In turbulent flows, initial guess of $\tilde{\nu}'$ is the most crucial one. Setting it to a small value of $\tilde{\nu}' < 1$ may result in a laminar flow simulation. If $\tilde{\nu}'$ is set to a larger value such as 3, then the computation diverges. It is generally safe to set initial guess of $\tilde{\nu}_\infty{}'$ [6].

The no-slip condition, introduced as the first boundary condition, has the following definition: The fluid velocity (U) adjacent to the solid wall is equal to the velocity of the wall. This can be expressed as:

$$U = U_{wall} \text{ on the wall} \tag{5}$$

Secondly, a Dirichlet condition on temperature is defined as:

$$T = T_{wall} \text{ on the wall} \tag{6}$$

Thirdly, a Neumann condition on the gradient of temperature is defined as:

$$-\gamma\frac{\partial T}{\partial n} = q_{wall} \text{ on the wall} \tag{7}$$

Since the current study is dealing with external turbulent flows with high Reynolds numbers around embedded boundaries, wall-bounded flow approach should be followed to limit the resolution of the fine grids in the turbulent boundary layer. In this region, the wall function is employed for the coarse grid utilization ($30 \leq y^+ \leq 150$) instead of fine grids ($y^+ < 5$), thus eliminates stiffness problem owing to redundant refinement.

RESULTS

In the solution of turbulent flows one-equation SA turbulence model implemented. The performance of GeULER-NaTURe flow solver in predicting lift coefficient is tested on the case of high Reynolds number, steady, two-dimensional flow around NACA 0012 airfoil. The experimental data from NASA's Langley 8-Foot Transonic Pressure Tunnel [7] is used to validate the adaptive Cartesian grid generation methodology for turbulent flow. Because of tunnel's large span-chord ratio and small side-wall-boundary-layer effects, the data of this experimental study is accepted as a benchmark test case for predicting separated flows. GeULER-NaTURe solution of chord-wise pressure distributions for NACA 0012 airfoil at a fairly high and real-life-flight Reynolds number of 9×10^6 and Mach number of 0.799 at the corrected angle-of-attack of 2.26º (corrected for interference of the wind tunnel wall) is compared with the experimental study of Harris [7]. The input parameters fed to the solver are tabulated in the Table-1 below. Eventual mesh constructed around the geometry is shown in Fig. 1.

TABLE 1. Input file for transonic, turbulent flow around NACA 0012

Cartesian Grid	
Outer boundary size factor	18
Number of successive divisions to generate uniform grid	4
Boundary size factor for box-adaptation	2.5 & 1.5
The size of small cells wrt maximum body dimensions	0.005, 0.01, 0.02, 0.05
Maximum level of cell	20
Boundary Conditions	
Mach number	0.799
Angle of attack (º)	2.26
Reynolds number	9×10^6
Prandtl number (laminar and turbulent)	0.72, 0.90
Specific heat ratio of the fluid	1.4
Free stream temperature in Kelvin	293.15
Solution Parameters	
Solution scheme for the flux	AUSM
Multiplication criteria for refinement	0.05
Number of refinement and coarsening cycle	2
Order of the solver	2nd
Convergence Parameter	
Residual exponent for the limit of convergence	-5

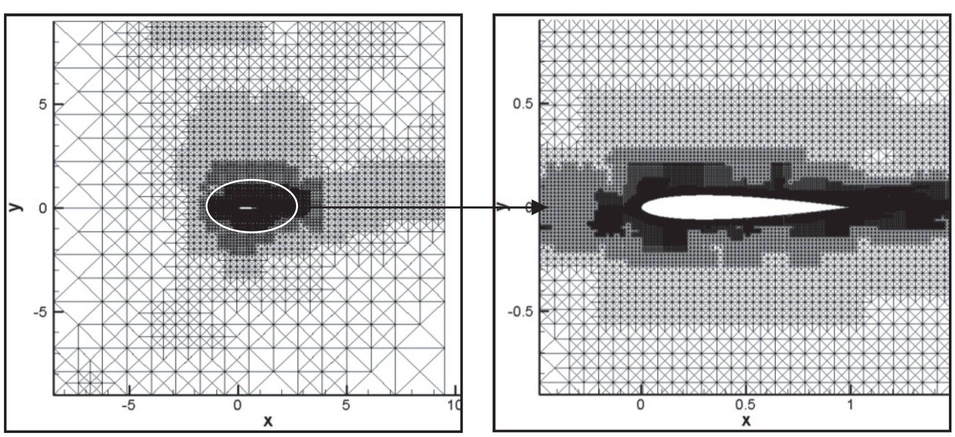


FIGURE 1. Mesh generation after three level solution adaptation around NACA 0012 airfoil

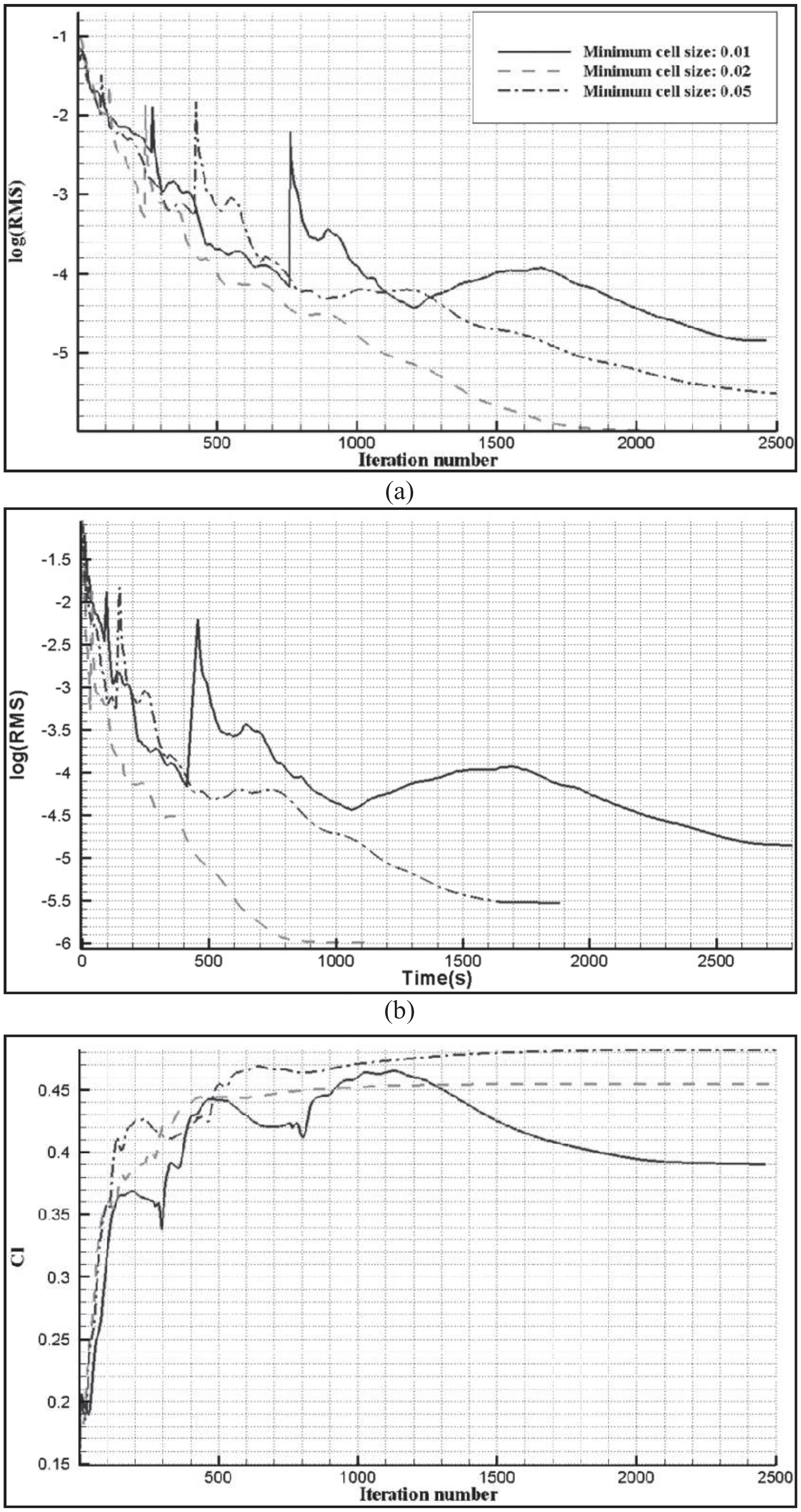


FIGURE 2. Transonic, turbulent test cast case of e of NACA 0012:(a) Convergence histories, (b) residuals versus Cus CPU time and (c) convergence histories of the lift coefficient; $M_\infty = 0.799$, $\theta = 2.26°$; Re = $9x10^6$

Figures 2 (a) and 2 (b) show the residual of the density and the convergence history versus CPU time, respectively. As seen in Figure 2 (c), for initial cell dimensions of 0.05 and 0.02, lift coefficient converges around 0.48 and 0.45, respectively. For the initial cell dimension of 0.01, it converges around 0.39 which is the exact experimental result of Harris [7].

CONCLUSION

In predicting lift coefficient, the performance of the GeULER-NaTURe flow solver with the addition of one-equation SA turbulence model is investigated by computing high Reynolds number steady flow around NACA 0012 airfoil at a fairly high and real-life-flight Reynolds number of 9×10^6 and Mach number of 0.799 with the corrected angle-of-attack of 2.26º. GeULER-NaTURe solutions are compared with the experimental benchmark study. For the initial cell dimension of 0.01, lift coefficient converges around 0.39 which is the exact experimental result of Harris.

SA turbulence model with tripping terms, modified one-equation turbulence models or two-equation turbulence models can be integrated into GeULER-NaTURe flow solver for predicting shock location comparatively well and solving lift coefficient faster as a future work.

REFERENCES

1. E. Kara, A. İ. Kutlar and M. H. Aksel, Progress in Computational Fluid Dynamics **16(3)**, 131–145 (2016).
2. E. Kara, A. İ. Kutlar and M. H. Aksel, "Solution refinement effectiveness of multi-grid accelerated, Cartesian grid based Navier Stokes Solver on compressible and laminar flows," in *International Youth Science Forum "Litteris et Artibus" (2016)* (Lviv, 2016), Vol. 8, pp. 306–307.
3. W. Sang and J. Yu, Aerospace Science and Technology **15(5)**, 375–380 (2011).
4. A. Hashimoto, K. Murakami, T. Aoyama and P. Lahur, AIAA-2009-1365 (2009).
5. P. Wong, "A compressible Navier-Stokes flow solver using the Newton-Krylov method on unstructured grids," Ph.D. thesis, University of Toronto, 2006.
6. P. C. Walsh, "Adaptive solution of viscous aerodynamic flows using unstructured grids," Ph.D. thesis, University of Toronto, 1998.
7. C. D. Harris, NASA Report, TM-81927, 1981.