



## The Study of the Lift Coefficient of NACA-0012 Airfoil Subjected to an Incompressible Media by Using the Cartesian Cut Cell Method

Ramadan Ghmati<sup>\*1</sup>, and Mehdi Ghatas<sup>2</sup>

<sup>1</sup>Marines and Offshore Engineering Department-Faculty of Engineering-Tripoli University, Libya

<sup>2</sup>Mechanical Engineering Department-Faculty of Engineering-Almerghib University, Libya

\*elgamate@yahoo.com , mehdi\_gat@yahoo.com

### ABSTRACT

In order to shorten product design time, engineers need to create high quality meshes within a few days or hours. Newer automated techniques have been published to tackle this need, the Cartesian Cut Cell is one of them. This study is focused on the effect of different Cut Cell meshing strategies on the accuracy of aerodynamic performance predictions. The method can be described as a methodology in which Cut Cells are applied to the geometry utilizing a process involving rectangular/hexagonal cells on a regular lattice cutting through the geometry. The Cut Cell meshing is a general purpose designed for ANSYS FLUENT, making use of Workbench to construct the airfoil shape and the mesh. The results obtained for NACA-0012 are computed using two models available in ANSYS FLUENT, namely the Eq. 2  $k-\omega$  SST and the Eq. 3  $k-k\ell-\omega$  models. The three-dimensional numerical simulations were created for steady incompressible flow around NACA-0012 shape. Lift coefficient, Boundary layer thickness, mesh expansion ratio, and mesh density variation parameters were investigated. For this application both models produce good lift results.  $k-k\ell-\omega$  produce better lift and the results are close to the measured data. The Cut Cell method showed a very good agreement between Computational Fluid Dynamics results and experimental data. This work illustrated that the Cartesian Cut Cell method has the ability to generate high quality mesh, which captures the details of the viscous boundary layer easily. The future work is to use more sophisticated turbulence models and mesh refinement for air craft wing with flap.

**Keywords:** Cartesian; Cut Cell; Coefficient; lift; Meshing; predications.

## 1. INTRODUCTION:

In the pursuit of increased airplane operating profits, the main goal for airframe manufacturers is to optimize the performance of their product. Customers seek good quality service in terms of minimum cost and reasonable flight time, while aviation manufacturers look to increase their profit margin in today's competitive market. Therefore, manufacturers need to define airplane wing configurations that possess the qualities essential to achieve those goals.

Defining airplane configurations is an overwhelmingly challenging task. Subsequently, carrying out research to increase the accuracy of modeling turbulent flow with extensive separations, the flow near the airfoil edge, trailing viscous wakes and the merging boundary layer becomes inevitable to develop such airplane configurations. With advances in computing power and Computational Fluid Dynamics (CFD) algorithms, the complexity of the simulated engineering models has significantly increased. This fact encourages engineers to rely on CFD simulations for testing and refining new technological ideas. A host of approaches to CFD were proposed over the course of the last three decades or so. The work of Purvis and Burkhalter in 1979 [1], and Wedan and South in 1983 [2], marked the advent of the Cartesian Cut Cell method. Their work involved utilizing the finite volume method for the full potential equations. In [3] the authors were able to successfully apply it to the shallow water equations in two and three dimensions. In each cell, the process of cutting the geometry is based on a linear piecewise cut. Clarke et al. [4], extended this work to the 2D Euler equations. This extension involved the addition of an agglomeration procedure in which smaller cells were treated as adjacent cells in a way that would not limit the specified time step. The work demonstrated some agreement relative to an analytical airfoil solution at the leading edge. The Euler finite volume method was extended to 3D by Gaffney et al. [5]. Their extended approach continued to use linear cuts and the small volume cut cell agglomeration procedure. Later, significant interest in the adaptive Cartesian Cut algorithms for problems associated with complex geometries was seen. In this regard, Aftosmis et al. [6], reported a new methodology to handle rapid and robust Cartesian mesh generation pertaining to component based geometry. The mesh is generated via a cell division process applied to the initial uniform grid. Yang et al. [7,8], suggested a methodology for computing compressible flows. The algorithm uses the Cartesian Cut Cell approach in conjunction with a multi-dimensional high resolution upwind finite volume scheme. The algorithm is versatile in the sense that it is able to cope with static as well as moving body problems, in which the associated geometry is complex. Yang et al. in [9], extended the work reported in [6,7], specifically, the Cartesian cut cell method to the three-dimensional case. Their extension covers the static and moving body problems. In 2000, Tucker and Pan [10], applied the Cartesian Cut Cell method to model incompressible laminar flow. The procedure involved cutting out solid bodies or boundaries in the flow domain. Three benchmarks were used to qualify this hybrid approach. Shortly afterwards, Causon et al. [11], reported a new approach

in which the focus was on calculating shallow water flows in the presence of moving physical boundaries. methodology was qualified using a problem involving a ship hull moving at supercritical speed and conjectural landslide events in which the material dips suddenly into a quiescent shallow lake and a fiord. Murnam et al. [12], proposed a supersonic missile system with a small number of synchronous canard control surfaces. An automated inviscid Cartesian method was used in designing the missile system. Simulations for total motion were carried out for the canard dither schedules pertaining to level flight, pitch and yaw maneuvers. High resolution viscous simulations and other experimental data were used to validate the time dependent dynamic simulations which were utilized to determine dynamic stability derivatives. Wang et al. [13], proposed a finite volume methodology to tackle electromagnetic wave dynamics in the time domain. A host of test cases with existing analytical solutions were used to qualify the computational electromagnetic solver. Murnam et al. [14], proposed a methodology to simulate impermeable boundaries within a fixed Cartesian mesh.

The scheme lowers the frequency at which the geometry is intersected with the Cartesian volume within a full simulation. Ingram et al. [15], reported a Cartesian Cut Cell based method which was regarded as an alternative to the traditional boundary fitted grid methods.

Not much work has been done involving the full cell-based methods for full Navier Stokes equations. In 1994, Tau [16], presented a two-dimensional approach for solving the Navier-Stokes equation on a staggered grid. A decade later, the method was extended by Kirkpatrick et al. [17], to solve the Navier-Stokes equations on a non-uniform staggered grid in three dimensions involving curved boundaries. The methodology was examined using a flow through a channel placed oblique relative to the grid as well as a flow past a cylinder at  $Re = 40$ . The results obtained seem within good agreement relative to experimental data obtained for this flow. Dorge and Verstappen [18], presented a method to solve the unsteady incompressible Navier-Stokes equations, in which the problem domain was of arbitrarily shaped boundaries. The method was examined using an incompressible unsteady flow around a circular cylinder in which  $Re$  is 100. Rosatti et al. [19], presented methods based on radial basis functions associated with a high degree of approximation. Shortly afterward, Rosatti et al. [20], suggested extending the shallow water semi-implicit models on staggered Cartesian meshes to account for the existence of Cut Cells at the computational space boundaries. A host of simulations were carried out to assess the accuracy of the environmental flow models. Chung [21], reported a Cartesian grid-based methodology with Cut Cells intended for simulating two dimensional unsteady viscous incompressible flows associated with arbitrarily shaped rigid bodies. Ji et al. [22], suggested a numerical methodology for solving, in 2D, the variable coefficients of the Poisson equation in which the interface is irregular, and the coefficients as well as the very solution might not be continuous everywhere across the solution space. Popsecu et al. [23], attempted to simulate the sound waves generated by oscillating baffled pistons. The wave equations in Cartesian coordinates along with cut cells were used along with a compact finite volume scheme to implement spatial discretization. Sang and Li [24], presented a methodology for handling the computations associated with the complex flow fields around three-dimensional high lift configurations. The performance of the methodology was compared with experimental data. Hsu [25], examined the numerical performance pertaining to the explicit Cartesian methodologies in the context of compressible flows. Pattinson et al. [26], and Pattinson [27], reported a Cut Cell-based methodology that is

based on multi-grid and non-conforming Cartesian mesh methodology to model inviscid compressible and non-compressible flows.

The specific objective of this work is to assess the numerical capability of the Cartesian Cut Cell method for predicting the lift coefficient when applied to NACA-0012. The rationale behind choosing the Cartesian Cut Cell method is that it is a new computational technique for mesh generation. This task is the crucial part of the numerical solution procedure in many engineering problems. The work presents model and mesh scenarios and the deviation between numerical solutions and experimental data.

## 2. GOVERNING EQUATIONS:

To date, there has been relatively little work focusing on the Cartesian Cut Cell method for the full Navier-Stokes equations. In the present work, the authors implemented a Cut Cell method based approach to solve the Navier-Stokes equations on an unstructured mesh in three dimensions. The use of unstructured grids in the solution of the Navier-Stokes equations is attractive as it exhibits a whole host of advantages. Accordingly, the approach is popular in the context of solving incompressible flow problems. Among the advantages the approach offers, is the capability of overcoming numerical problems associated with pressure velocity coupling. This problem, normally, occurs when a collocated grid is used [30,31].

$$\frac{\partial(\bar{u}_i)}{\partial x_i} = 0 \quad (1)$$

$$\bar{u}_j \frac{\partial(\bar{u}_i)}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \bar{P}}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mathbf{v} \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \overline{u'_i u'_j} \right] \quad (2)$$

Where  $\bar{u}_i$  are the Cartesian components of the mean flow velocity vector  $\mathbf{u} = (u_1, u_2, u_3)$ ;  $x_i$  are the Cartesian coordinates of the position vector  $\mathbf{x}=(x_1, x_2, x_3)$ ;  $\bar{P}$  is the mean flow pressure;  $\overline{u'_i u'_j}$  are Reynolds stresses or turbulent stresses,  $\rho$  is the fluid density and  $\mathbf{v}$  is the kinematic viscosity of the fluid.

Steady state incompressible flow around a airfoil shape with high Reynolds number was the basic assumption for the simulations.

- The equations discretized in space using finite volume formulation on a Cartesian grid. The second order upwind scheme is used for convective terms.
- The resulting discretized transport equation for a general variable  $\phi$  using information at three time levels,  $n+1$ ,  $n$  and  $n-1$  written as [32]:

$$\begin{aligned}
 A_p^{n+1} \Phi_p^{n+1} = & \sum_{nb} (A_{nb}^{n+1} \Phi_{nb}^{n+1}) + S_{imp} \Phi_p^{n+1} + S_{exp} \Phi_p^{n+1} \\
 & + \left[ \sum_{nb} (A_{nb}^n \Phi_{nb}^n) - A_p^n \Phi_p^n + S_{imp} \Phi_p^n + S_{exp}^n \right] \\
 & + \left[ \sum_{nb} (A_{nb}^{n-1} \Phi_{nb}^{n-1}) - A_p^{n-1} \Phi_p^{n-1} + S_{imp} \Phi_p^{n-1} + S_{exp}^{n-1} \right] \quad (3)
 \end{aligned}$$

Where nb is a generic subscript for neighbor cell; and  $S_{imp}$  and  $S_{exp}$  contain any further implicit and explicit sources respectively. Neighboring cells are labeled using the following convention: East (E), North (N) and Up (U) corresponding to the positive  $x_1, x_2, x_3$  directions and West (W), south (S) and Down (D) to the negative directions. The subscript P refers to the cell for which the flux is being calculated.

### 3. CUTCELL METHOD:

One of the principle difficulties in solving a system of nonlinear partial differential equations in complex geometries involves the setting up of a suitable computational domain. If the system of PDE's has a domain of smooth behavior and other rapid variation, the problem of designing the mesh is compounded. The problem of solving a system of PDE's in 3D incompressible flow can be divided into two major tasks, the construction of a suitable mesh and the specification of the PDE's. In the present work, the authors addressed the issue of solving a system of PDE's using what has become known as a Cut Cell Method. Cut Cell Cartesian meshing is a general purpose meshing designed for ANSYS FLUENT [32]. Make use of workbench to construct the airfoil shape and the Cut Cell mesh. For a very simple mesh, a mesh specification is necessary, the mesh can then be refined based upon the specified criteria.

The framework is divided into four main sections: geometry construction, mesh generation, and results visualization. The first step in solving the problem is to get a geometry on which the simulation is to take place. The chosen geometry is constructed from basic geometrical entities such as points, lines, curves, planes, etc. The various boundary options that are to be set include inlet, outlet, wall, and symmetry. The Computational domain for this study consists of a NACA-0012 contained within a far field shape. The mesh section parses the mesh specification file, which is the required mesh to model the flow around the object.

A Cut Cell meshing is a patch independent volume meshing approach with no necessity for manual geometry creation or breakdown, thus reducing the turnaround time necessary to construct the lattice. The Cut Cell algorithm is appropriate for a great range of functions, and due to the huge fraction of hex cells in the mesh, often creates enhanced results compared to tetrahedral schemes. The general advantage of the CutCell mesh procedure is that it captures the size function values which are calculated. The lattice is then adaptively distinguished based on the local size function values. The cells intersected by the geometry are identified for projection. The edges intersected by the geometry were recognized and mesh edges to be

recovered are determined and used to build mesh faces. Once the mesh faces were identified, cells were modified to recover these faces. [See Figure (1)] The boundary mesh is identified and divided on the original geometry. In the Cut Cell mesh approach the global inflation controls should be set first. It is essential to set inflation controls in advance of the Cut Cell meshing approach. If the global inflation was set after the generation of the Cut Cell, the re-mesh feature would be activated.

The outcome of the cutting algorithm was a Cut Cell mesh of a computational domain obtained from the original background mesh by removing elements completely contained in the geometry [28].

Depending upon the geometry, and flow details, different mesh densities are essential to capture finer details of flow at regions of interest in the domain. The output of the cutting algorithm is a Cut Cell mesh of a computational domain obtained from the original background mesh by removing elements completely contained in the geometry, Figure (2) illustrates a generated Cut Cell meshing.

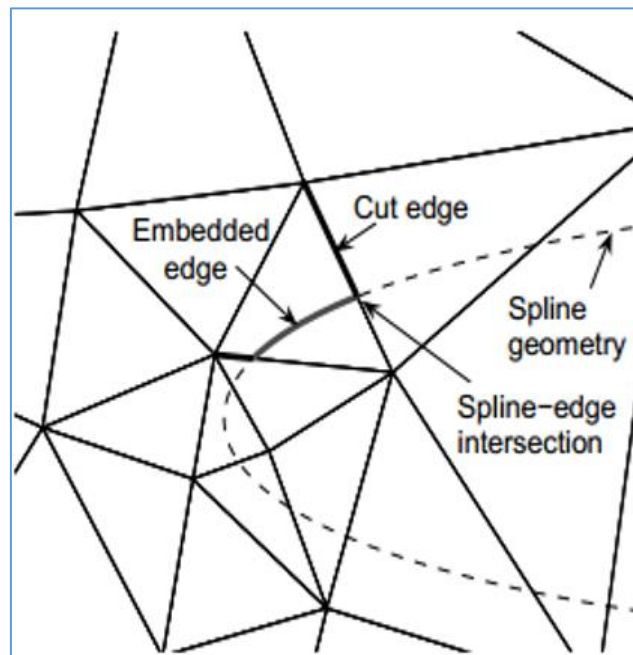


Figure 1: Intersection between a background mesh and an airfoil [27]

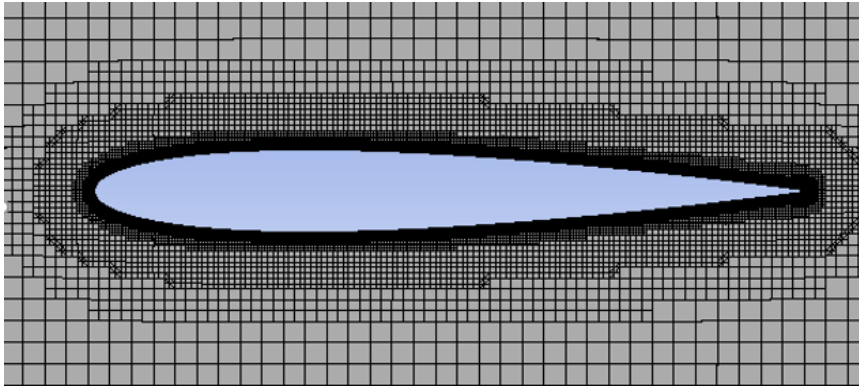


Figure 2: NACA0012Cut Cell Meshing

#### 4. SIMULATION AND RESULTS:

The meshed file was imported into the CFD package Fluent to employ the steady incompressible flow simulations. The Boundary Layer was modeled to be fully turbulent and sst  $k-\omega$  (Eq. 2) and transition  $k-k\ell$  omega (Eq. 3) [34] were used for this work. A Navier-Stokes solution was computed around a NACA-0012 in a single value of the free stream Mach number,  $M_\infty=0.15$ ,  $Re=6$  million and angles of attack of  $0^\circ$ ,  $2^\circ$ ,  $4^\circ$ ,  $6^\circ$ ,  $8^\circ$  and  $10^\circ$ . The Reynolds number is based on the airfoil chord,  $L$ , the free stream velocity,  $U_\infty$ , and kinematic viscosity,  $\nu_\infty$ . The simulations were run for different mesh specifications and different angles of attack to demonstrate the accuracy the of Cut Cell Method in predicting the performance of aerodynamic forces. The results were validated against the available experimental data [29], but it should be noted that the deviation between experimental data and simulation results increases at higher angles of attack. The lift caused by the force perpendicular to the flow is

$$C_L = \frac{F_L}{\frac{1}{2}\rho U_\infty^2 A} \quad (4)$$

where  $A$  is the area of the airfoil,  $C_L$  is the lift coefficient and  $U_\infty$  is the free stream velocity. The authors presented the lift as Tables and Figures to show the simulation results. The computational results were compared to the experimental data obtained from the wind tunnel. Both the simulation work and experimental data were run on the same conditions.

Table (1) illustrates the simulation and experimental lift Coefficients [29] for different angle of attack for lift coefficients prediction for both sst  $k-\omega$  (Eq. 2) and transition  $k-k\ell$  omega (Eq. 3) models. It can be seen from columns two, three, and four the good agreement between the computational and the experimental results for both models for different mesh size to predict the lift coefficient. The error percent of both  $k-\omega$  and  $k-k\ell$  omega are tabulated in Table (1). One can see the accuracy of the used models for predicting the lift coefficient. The simulation and experimental data for lift coefficients are plotted on Figure (3). For 2 degree angle of attack several cases were run for different parameters such as; boundary layer thickness, inflation options and meshes size. Table (2) shows the different mesh size and the results

plotted on Figures (4-5). We can determine that 9 B.L.T (Boundary Layer thickness), fine mesh and Last Aspect Ratio inflation option demonstrated good results compared to experimental data for lift predication.

## 5. CONCLUSIONS:

The most important consideration in aerodynamic design is the accuracy of the turbulent model utilized for simulating complex turbulent flows. Along with computer and numerical simulation techniques, turbulence modeling has improved over the last decade to match the challenges of analyzing current aerodynamic systems. The Cut Cell numerical method was validated in three-dimensional viscous steady incompressible flow utilizing NACA-0012. The present results obtained for NACA-0012 are computed using the two models available in ANSYS FLUENT, namely sst k- $\omega$  (Eq. 2) and transition k-kl omega (Eq. 3) models. It was shown in this study that the Cut Cell mesh method has the ability to generate high quality mesh which captures the details of the viscous boundary layer easily. Future work will focus on using the Cut Cell Method for more sophisticated turbulence models and mesh refinement to predict the lift coefficients for high left devices with flap.

Table 1: The simulation and experimental lift Coefficients for different angle of attack for lift coefficients prediction

Alpha	k-kl omega	K-omega	EXP.	k-kl omega error %	K-omega error %
0	0.0000	0.0001	0.0000	0.0000	0.0000
2	0.2190	0.1855	0.2200	-0.4517	-15.7045
4	0.4383	0.4004	0.4400	-0.3864	-9.0091
6	0.6604	0.6450	0.6500	1.6015	-0.7769
8	0.8751	0.8459	0.8700	0.5862	-2.7678
10	1.0901	1.0433	1.0700	1.8785	-2.4953

Table 2: Three different mesh sizes were tested for angle of attack equal 2 deg.

	Element size of wing soft faces	Body sizing	Number of elements
Coarse	10000	100000	6701569
Fine	1.8 mm	500 mm	8655456
X-fine	1.8	80	13805201



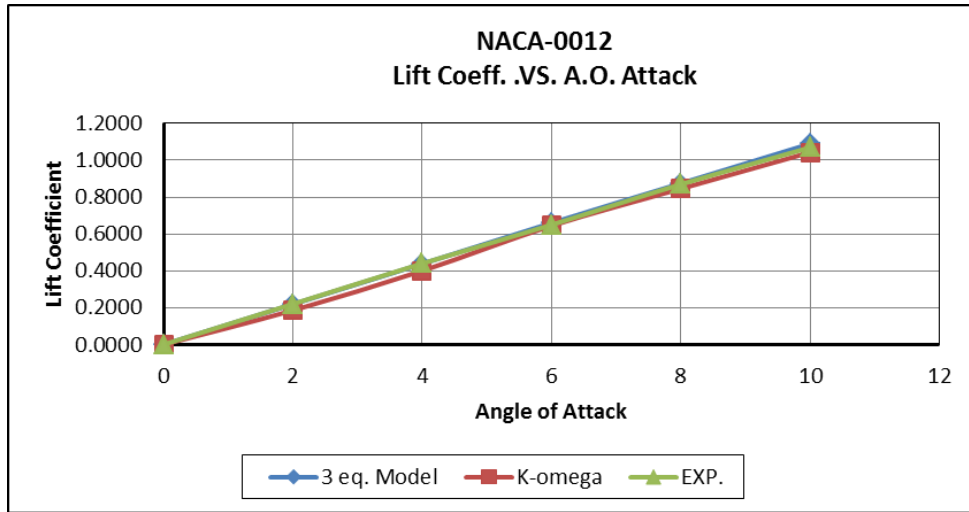


Figure 3: Lift Coefficients for .VS. Angle of attack

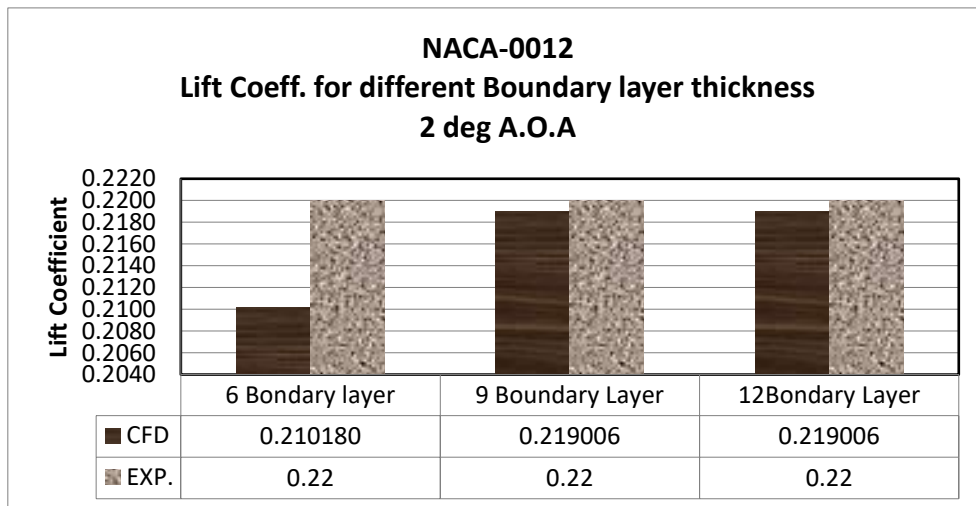


Figure 4: Lift coefficients for different Boundary layer thickness.

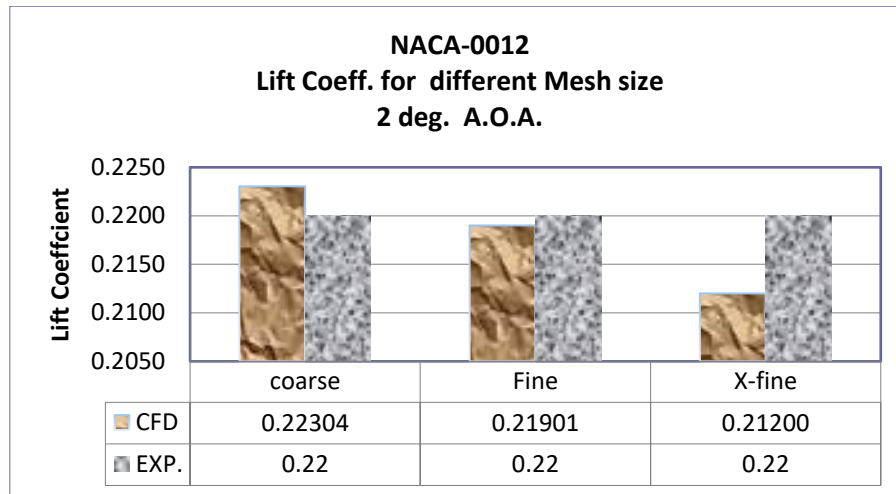


Figure 5: Lift coefficients for different mesh size.

## REFERENCES

1. J. W. Purvis and J. E. Burkhalter , Prediction of critical Mach number for store configurations, AIAA J., 17(11) (1979), 1170-1177.
2. B. Wedan and J. C. South, A method for solving the transonic full potential equation for general configurations , Computational Fluid Dynamics Conference, 6th, Danvers, MA; United States; 13-15 July (1983) 515-526.
3. K. J. Fidkowski, A simplex Cut cell adaptive method for high order discretizations of the compressible Navier Stokes equations, DE Thesis, Massachusetts Institute of Technology , Jun(2007).
4. D. K. Clarke, M. D. Salas and A. Hassan, Euler calculation for multi element air foils using Cartesian grids, American Institute of Aeronautics and Astronautics, Aerospace Sciences Meeting, 23rd, Reno, NV; United States; 14- Jan. (1985).
5. R. L. Gaffney, M. D. Salas and H. A. Hassan, Euler Calculations for wings using Cartesian grids, AIAA(1987) 1987-0356.
6. M. J. Aftosmis , M. J. Berger and J. E. Melton , Robust and efficient Cartesian mesh Generation for component based geometry, AIAA 97-0196, Jan. (1997). Also AIAA J. 36(6): June (1998), 952-960.
7. G. Yang, D. M. Causon, Ingram, R. Saunders and P. Batten, A Cartesian cut cell method for compressible flows part A: Static body problems, Aeronautical J. 2119 Jan. (1997).
8. G. Yang, D. M. Causon, Ingram, R. Saunders and P. Batten, A Cartesian cut cell method for compressible flows part B: Static body problems, Aeronautical J. 2120 Jan. (1997).
9. G. Yang, D. M. Causon and D. M. Ingram, Calculation of compressible flows about complex moving geometries using a three dimensional Cartesian cut cell method , Int. J. Numer. Meth. Fluids, 33 : 1121-1151 (2000).

10. P. G. Tucker and Z. Pan, A Cartesian cut cell method for incompressible flow, *J. of Appl. Mathematical Modeling*, 24(2000) : 591-606.
11. D. M. Causon, D. M. Ingram and C. G. Mingham, A Cartesian cut cell method for shallow water flows with moving boundaries, *Advances in water Resources* 24(2001) , 899-911.
12. S. M. Murman, M. J. Aftosmis and M. J. Berger, Numerical Simulation of Rolling Airframes using a Multi-level Cartesian method, AIAA-2002-2798.
13. Z. J. Wang, A. J. Przekwas and Y. Liu, A FV-TD electromagnetic solver using adaptive Cartesian grids , *J. Computer Phys. Commun.* 148 (2002)17-29.
14. S. M. Murman, M. J. Aftosmis and M. J. Berger, Implicit Approaches for moving boundaries in a 3D Cartesian Method, AIAA-2003-1119, January 6-9.
15. D. M. Ingram, D. M. Causon and C. G. Mingham, Development in Cartesian cut cell methods , *Mathematics and Computers in Simulation* 61(2003) 561-572.
16. E. Y. Tau, A 2nd order projection method for incompressible Navier-Stokes equations in arbitrary domains, *J. Comp. Phys.* 115(1994) 147-152.
17. M. P. Kirkpatrick, S. W. Armfield , and J. H. Kent , A representation of curved boundaries for the solution of the Navier-Stokes equations on a staggered three dimensional Cartesian grid , *J. Comput. Phys.* 184(2003) 1-36.
18. M. Droge and R. Verstappen, A new symmetry preserving Cartesian grid method for computing flow past arbitrarily shaped objects, *Int. J. Meth. Fluids*( 2005) 47:979-985.
19. G. Rosatti, R. Chemotti and L. Bonaventura, Higher order interpolation methods for semi-Lagrangian models of mobile-bed hydrodynamics on Cartesian grids with cut cells, *Int. J. Numer. Meth. Fluids* 2005 47:1263-1275.
20. G. Rosatti, D. Cesari and L. Bonaventura, Semi-implicit, semi-Lagrangian modeling for environmental problems on staggered Cartesian grids with cut cells, *J. of Comput. Phys.* 204(2005) 353-377.
21. M. Chung , Cartesian cut cell approach for simulating incompressible flows with rigid bodies of arbitrary shape , *J. of Computers & Fluid* 35(2006) 607-623
22. H. Ji, F. Lien and E. Yee , An efficient second order accurate cut cell method for solving the variable coefficient Poission equation with jump conditions on irregular domains, *aint. J. Numer. Fluids* 52(2006) 723-748.
23. M. Popescu, C-F Tai, W. Shyy, Cartesian Cut Cell Methods with local grid refinement for wave Computations, AIAA 2006-2522.
24. W. Sang and F. Li , Numerical Simulations for Transport Aircraft High-Lift Configurations using Cartesian Grid Methods , *J. Aircraft* 43(4) 2006,
25. K. Hsu , Numerical performances of explicit Cartesian methods for compressible moving boundary flow problems , *ASME* 2006.
26. J. Pattinson, A. G. Malan and J. P. Meyer, A cut-cell non-conforming Cartesian mesh method for compressible and incompressible flow , *Int. J. Meth. Engng* 72(2007) 1332-1354.
27. K. J. Fidkowski, A simplex Cut cell adaptive method for high order discretizations of the compressible Navier Stokes equations, DE Thesis, Massachusetts Institute of Technology , Jun(2007).
28. ANSYS Meshing User's Guide, Release 13.0, Nov. 2010.
29. Abbot and Doenhoff, Theory of Wing Section, 1959 edition.
30. H. K. Versteeg and W. Malaiasekers, An introduction to Computational Fluid Dynamics, 2<sup>nd</sup> edition.
31. J. H. Ferziger and M. Peric, Computational Fluid Dynamics, 3<sup>rd</sup> edition.
32. M. P. Kirkpatrick , S. W. Armfield , and J. H. Kent , A representation of curved boundaries for the solution of the Navier-Stokes equations on a staggered three dimensional Cartesian grid , *J. of Comput. Phys.*184(2003) 1-36.

33. <https://www.sciencedirect.com/book/9780081021644/engineeringanalysis-with-ansys-software>.
34. D. K. Walters and D. Cokljat, A three-equation eddy-viscosity model for reynolds-averaged navier-stokes simulations of transitional flows. J. of Fluids Eng. 130(8) 2008.

## NOMCLATURE

D	Down	$\bar{u}$	Mean Velocity
E	East	$\overline{u_i u_j}$	Reynolds Stresses
LES	Large Eddy Simulation	U	UP
N	North	AOA	Angle of Attack
nb	Neighbor cell	S <sub>exp</sub>	Explicit source
S	South	S <sub>imp</sub>	Implicit source
SST	Shear-Stress Transport		

## دراسة لتقدير معامل قوة الرفع للجنيح NACA-0012 يتعرض لوسائط غير انضغاطية باستخدام طريقة الخلية الكارتيزية المقطوعة

\*<sup>1</sup>رمضان القماطي ، <sup>2</sup>المهدي القطوس

<sup>1</sup>قسم الهندسة البحرية والمنصات العائمة، كلية الهندسة ، جامعة طرابلس ، ليبيا  
<sup>2</sup>قسم الهندسة الميكانيكية، كلية الهندسة ، جامعة المرقب ، ليبيا

\*elgamate@yahoo.com ، mehdi\_gat@yahoo.com

## المخلص

في ظل التطور لميكانيكا الموائع الحسابية التي تستخدم الطرق العددية لحل المسائل التي تتضمن سريان الموائع وتحليل نتائج المحاكاة باستخدام الحواسيب لإجراء الحسابات اللازمة للتوصل الى الحلول التقريبية لحل معادلات نافيرستوكس ، يحتاج المهندسون إلى إنشاء شبكات عالية الجودة من الشبكات العنقودية حول الأشكال الهندسية التي قد تستغرق أيام أو ساعات قليلة. تم استحداث عدة طرق لحل هذه المشكلة وتعتبر الخلية الكارتيزية المقطوعة هي احدي هذه التقنيات (Cartesian Cut Cell Method)، تركز الدراسة الحالية على مدى تأثير استراتيجيات تشابك الخلايا الكارتيزية المقطوعة المختلفة على دقة الأداء الديناميكي في الهواء الجوي. يمكن وصف طريقة الخلية الكارتيزية المقطوعة بأنها

منهجية للخلايا المقطوعة على الاشكال الهندسية باستخدام عملية تتضمن قطع خلايا مستطيلة / سداسية على شبكة الاشكال الهندسية، من اجل تحقيق الهدف من هذه الدراسة استخدم برنامج المحاكاة ANSYS FLUENT لتصميم شكل الجنيح وشبكة Cartesian Cut Cell، لقد استعمل نموذجين للسريان المضطرب المستقر الغير انضغاطي المتاحة في البرنامج و هما model k- $\omega$  (Eq. 2) و model k-kl- $\omega$  (Eq. 3)، تمت عملية المحاكاة الرقمية ثلاثية الأبعاد حول الجنيح NACA-0012 وفحص معامل الرفع، سمك الطبقة المتاخمة، ومتغيرات تباين كثافة الشبكة. أظهر استخدام طريقة الخلية الكارتيزية المقطوعة اتفاقاً جيداً جداً مع نتائج ديناميكا السوائل الحسابية والبيانات التجريبية ، أوضحت هذه الدراسة أن طريقة الخلية الكارتيزية المقطوعة لديها القدرة على إنشاء شبكة عالية الجودة تلتقط تفاصيل الطبقة المتاخمة للزجة بسهولة، في المستقبل سيتم توسيع نطاق استخدام هذه التقنية لنماذج أكثر تعقيداً للتنبؤ بمعاملات الرفع.

**الكلمات المفتاحية:** الكرتيزية; الخلية المقطوعة; معامل; الرفع; الشبكة; تقدير.