The Investigation of the Lift Coefficient of the NACA-0012 Airfoil in an Incompressible Medium by Means of the Cartesian Cut Cell Approach

G Veerabhadrappa [1], Mahesh [2], Ravi M Tilavalli [3]

[1] *Department of Mechanical Engineering, Government polytechnic Karatagi, Karnataka, India.*

[2] *Department of Mechanical Engineering, Government polytechnic Aurad, Karnataka, India.*

[3] *Department of Mechanical Engineering, Government polytechnic Harihara, Karnataka, India.*

ABSTRACT

Engineers must be able to generate high-quality meshes in a matter of days or hours if they want to reduce the time it takes to develop products. The Cartesian Cut Cell is one of many recently reported automated methods that aim to meet this need. The impact of various Cut Cell meshing procedures on the precision of aerodynamic performance forecasts is the primary focus of this work. Cut Cells, a technique using rectangular and hexagonal cells on a regular lattice that cuts through the geometry, are applied to the geometry in this way. Utilizing Workbench, the Cut Cell meshing is a general-purpose tool for ANSYS FLUENT that builds the airfoil form and mesh. The NACA-0012 form was the basis for the three-dimensional numerical models of steady incompressible flow. The parameters for change in mesh density, boundary layer thickness, lift coefficient, and mesh expansion ratio were examined. In this case, both models are effective in raising the target. The findings are in good agreement with the measured data, and k-kl-ö produces superior lift. The findings from Computational Fluid Dynamics and the experimental data were found to be highly concordant using the Cut Cell approach. The results showed that the Cartesian Cut Cell approach can produce a high-quality mesh that can readily capture the intricacies of the viscous boundary layer. More advanced turbulence models and mesh refinement for flap-equipped aircraft wings will be the focus of future efforts.

Keywords: Lift Coefficient, Airfoil, Cut Cell, Computational Fluid Dynamics, ANSYS FLUENT.

1. INTRODUCTION

The primary objective of airframe producers is to maximize the performance of their product in order to improve aircraft operational profitability. Manufacturers in the aviation industry want to expand their profit margin in today's competitive market, while customers desire affordable, acceptable travel times and excellent quality service. Consequently, manufacturers must specify wing layouts for airplanes that meet those requirements.

Attempting to define aircraft configurations is an enormously difficult undertaking. Research into improving the accuracy of turbulent flow models with large separations, flow close to the airfoil edge, trailing viscous wakes, and merging boundary layers is therefore necessary for the development of such aircraft designs. The computational capacity and methodologies for Computational Fluid Dynamics (CFD) have allowed for a dramatic rise in the complexity of engineering models that may be simulated. Because of this, CFD simulations are becoming more important for engineers to use while developing and evaluating new technological concepts. There have been a plethora of CFD methods developed within the past 30 years. Wedan and South introduced the Cartesian Cut Cell technique in 1983 [2] and Purvis and Burkhalater in 1979 [1]. Their task was to solve the whole potential equations using the finite volume approach. The authors of [3] used it to solve two- and three-dimensional shallow water equations with success. A linear piecewise cut is the basis for cutting the geometry in each cell. This approach was expanded to include the 2D Euler equations by Clarke et al. [4]. An agglomeration process was used into this expansion to accommodate smaller cells as nearby ones without limiting the required time step. In comparison to the state-of-the-art analytical airfoil solution, the study showed some agreement. A three-dimensional version of the Euler finite volume approach was developed by Gaffney et al. [5]. The small volume cut cell aggregation method and linear cuts persisted in their expanded approach. Appropriate adaptive Cartesian Cut algorithms for complicated geometry problems later attracted a lot of attention. The authors Aftosmis et al. [6] introduced a novel approach to component-based geometry that can generate Cartesian meshes quickly and reliably. Applying a cell division technique to the basic uniform grid generates the mesh. A technique for calculating compressible flows was proposed by Yang et al. [7,8]. In this procedure, a multidimensional high resolution upwind finite volume scheme is used in combination with the Cartesian Cut Cell approach. The algorithm's adaptability lies in its ability to handle difficult geometry-related static and moving body challenges. By applying the Cartesian cut cell approach to the three-dimensional example, Yang et al. [9] expanded upon the work presented in [6,7]. Their expansion addresses issues with both stationary and mobile bodies. The Cartesian Cut Cell technique was used to describe incompressible laminar flow by Tucker and Pan [10] in the year 2000. Part of the process included removing obstacles or borders from the flow domain. This hybrid technique was qualified using three benchmarks. Not long after that, a fresh method for determining shallow water flows with physically shifting boundaries was published by Causon et al. [11]. The approach was tested on a scenario stating that a ship's hull traveling at supercritical speed may experience hypothetical landslide occurrences where the material would plunge abruptly into a calm, shallow lake and a fiord. An arrangement of a few simultaneous canard control surfaces was suggested by Murnam et al. [12] for use in a system of supersonic missiles. The missile system was designed using an automated inviscid Cartesian technique. The canard dither sequences for level flight, pitch, and yaw motions were simulated for total motion. The time-dependent dynamic simulations used to calculate dynamic stability derivatives were validated using high-resolution viscous simulations in addition to other experimental data. A finite volume approach was suggested by Wang et al. [13] to deal with time domain electromagnetic wave dynamics. To certify the computational electromagnetic solver, a large number of test cases were used, each of which has an analytical solution already in place. An approach to mimicking impermeable barriers within a static Cartesian mesh was put out by Murnam et al. [14].

The plan reduces the number of times in a complete simulation that the geometry intersects with the Cartesian volume. An alternative to the conventional boundary fitted grid techniques was described by Ingram et al. [15] and it was based on Cartesian cut cells.

The entire cell-based approaches to full Navier Stokes equations have not been extensively studied. A two-dimensional method for solving the Navier-Stokes equation on a staggered grid was proposed by Tau [16] in 1994. The Navier-Stokes equations on a non-uniform staggered grid in three dimensions with curving edges were expanded a decade later by Kirkpatrick et al. [17]. The technique was tested with a flow past a cylinder at Re=40 and a flow via a channel that was angled with respect to the grid. When compared to experimental data collected for this flow, the findings appear to be within excellent agreement. When the problem area had arbitrarily formed borders, Dorge and Verstappen [18] offered a way to solve the unstable incompressible Navier-Stokes equations. To test the approach, we used a circular cylinder with a Reynolds number of 100 and an incompressible, unstable flow around it. Methods using radial basis functions linked to a high degree of approximation were introduced by Rosatti et al. [19]. Not long after that, Rosatti et al. [20] proposed adding Cut Cells at the computational space borders to the shallow water semi-implicit models using staggered Cartesian meshes. To ensure the environmental flow models were accurate, a large number of simulations were run.

For the purpose of modeling two-dimensional unstable viscous incompressible flows linked to arbitrarily formed rigid structures, Chung [21] described a system based on a Cartesian grid that uses cut cells. An approach to numerically solving the Poisson equation with variable coefficients in two dimensions was proposed by Ji et al. [22]. This approach accounts for cases where the interface is not perfectly smooth and where neither the coefficients nor the solution are necessarily continuous across the solution space. The goal of Popsecu et al. [23] was to model the behavior of oscillating baffled pistons in order to create artificial sound waves. In order to apply spatial discretization, a compact finite volume scheme was employed in conjunction with cut cells and wave equations expressed in Cartesian coordinates.

In order to manage the calculations related to the complicated flow fields around three-dimensional high lift systems, Sang and Li [24] developed a technique. Experimental data was used to compare the methodology's performance. Numerical performance of explicit Cartesian methods for compressible flows was studied by Hsu [25]. The authors Pattinson [27] and et al. [26] described a Cut Cell-based approach to modeling inviscid compressible and non-compressible flows using multi-grid and non-conforming Cartesian mesh methodologies. Using NACA-0012 as an example, this study aims to determine how well the Cartesian Cut Cell technique can forecast the lift coefficient numerically. The novel computational methodology for mesh production, Cartesian Cut Cell, was the deciding factor in its selection. The numerical solution process for many engineering issues relies on this activity. Discrepancies between numerical solutions and experimental data, as well as model and mesh situations, are detailed in the article.

2. CUTCELL METHOD

The establishment of an appropriate computing domain is one of the key obstacles to solving a system of nonlinear partial differential equations in complicated geometries.

The challenge of building the mesh is intensified if the system of partial differential equations exhibits domains of smooth behavior and other types of fast variation. There are two main steps to solving a system of PDEs in 3D incompressible flow: first, building an appropriate mesh; and second, specifying the PDEs. Solving a system of partial differential equations (PDEs) using the now-popular Cut Cell Method was the focus of the authors' current effort. One general-purpose meshing method developed for ANSYS FLUENT is cut cell cartesian meshing [32]. Use the workbench to build the cut cell mesh and the airfoil form. Even with a basic mesh, it's important to have a specification so that it may be fine-tuned according to the requirements.

Geometry building, mesh production, results display, and results analysis are the four primary parts of the system. Obtaining a geometry to run the simulation on is the first stage in finding a solution to the challenge. Various geometrical building blocks, including points, lines, curves, planes, etc., make up the selected geometry. Inlet, outlet, wall, and symmetry are the available boundary possibilities. This study's computational domain is a far field shape including a NACA-0012. The mesh specification file must be parsed in the mesh section in order to simulate the object's flow.

The construction time of the lattice may be reduced using a cut cell meshing method, which is a patch independent volume meshing methodology that does not need human geometry development or breakdown. Because of the large proportion of hex cells in the mesh, the Cut Cell method often produces better results than tetrahedral schemes, and it is suitable for a wide variety of tasks. The CutCell mesh process captures the estimated size function values, which is a general benefit. Next, the values of the local size function are used to adaptively differentiate the lattice. In order to project, we need to find the cells that the geometry intersects. In order to construct mesh faces, we first identified the edges that the geometry intersected with, and then we calculated which edges of the mesh needed to be recovered. After the mesh faces were located, the cells were adjusted such that they could be recovered. Look at Figure (1). Based on the initial geometry, the boundary mesh is located and segmented. It is recommended to begin by setting the global inflation controls in the Cut Cell mesh technique. Prior to the Cut Cell meshing procedure, it is critical to establish inflation controls. You may enable the remesh functionality by setting the global inflation after the Cut Cell is generated.

The algorithm's output, a computational domain Cut Cell mesh, was created by extracting the original background mesh and deleting all geometry-containing parts [28]. To capture greater details of flow at domain areas of interest, varying mesh densities are necessary, depending on the geometry and flow features. Figure (2) shows the result of a created Cut Cell meshing, which is a computational domain's output from the initial backdrop mesh that has had all geometry-containing components removed.

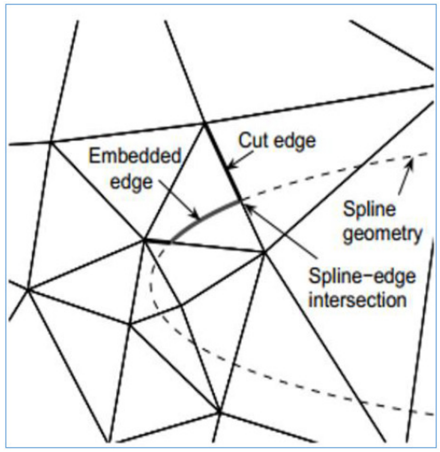

Figure 1: Intersection between a background mesh and an airfoil [27]

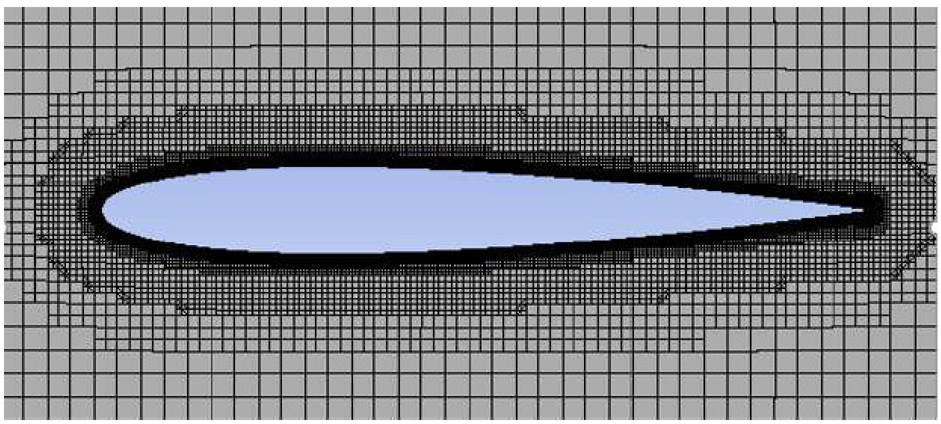

Figure 2: NACA0012 Cut Cell Meshing

3. SIMULATION AND RESULTS

The meshed file was imported into the CFD program Fluent to conduct steady incompressible flow simulations. The Boundary Layer was simulated as completely turbulent, using the sst k-ω model and the transition k-kl omega model [34] for this study. A computational solution using the Navier-Stokes equations was obtained for a NACA-0012 airfoil. The solution was calculated for a single value of the free stream Mach number, M_∞=0.15, and a Reynolds number of 6 million. The angles of attack considered were 0º, 2º, 4º, 6º, 8º, and 10º. The Reynolds number is calculated using the airfoil chord length, L, the free stream velocity, U_∞, and the kinematic viscosity, v_∞. The simulations were conducted using various mesh parameters and angles of attack to showcase the precision of the Cut Cell Method in forecasting aerodynamic forces. The findings were verified using the existing experimental data [29], however, it is important to acknowledge that the discrepancy between the experimental data and simulation results becomes more pronounced at larger angles of attack. The lift generated by the force acting perpendicular to the direction of the flow is

$$C_L = \frac{F_L}{\frac{1}{2}\rho U_\infty^2 A}$$

The variables in the equation are as follows: A represents the area of the airfoil, C_L represents the lift coefficient, and U_∞ represents the free stream velocity.

The authors used Tables and Figures to visually display the lift and demonstrate the simulation findings. The computational findings were compared to the empirical data acquired from the wind tunnel.

Both the modeling and experimental data were conducted under identical circumstances.

Table (1) presents the simulated and experimental lift coefficients [29] for various angles of attack. The table compares the predicted lift coefficients for both the sst k- ω and transition k-kl omega models. The columns two, three, and four clearly demonstrate the strong correlation between the computational and experimental findings for both models, across various mesh sizes, in predicting the lift coefficient. The error percentages for both k-ω and k-kl omega are shown in Table (1).

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

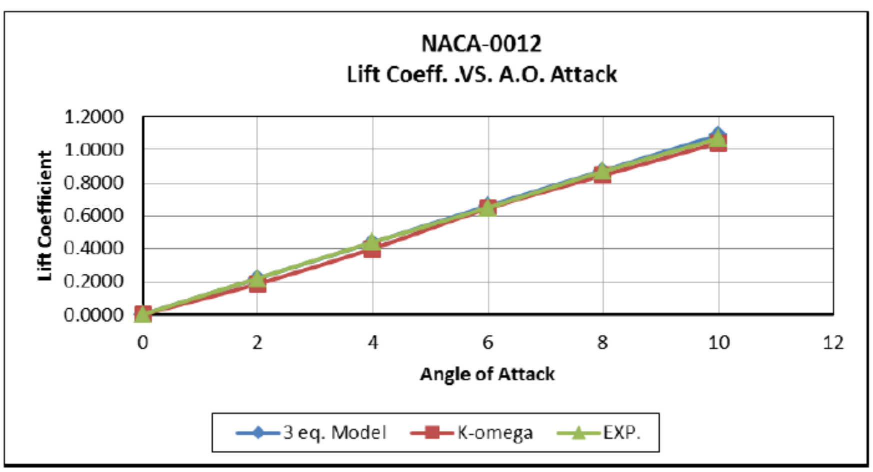

Figure 3: Lift Coefficients for v/s AOA

The precision of the used models in forecasting the lift coefficient is evident. The lift coefficients obtained from the simulation and experimental data are shown on Figure (3). Various scenarios were conducted using varied parameters, including boundary layer thickness, inflation choices, and mesh size, at a 2-degree angle of attack. Table 2 displays the various mesh sizes and the corresponding findings depicted on Figures 4 and 5. Based on our analysis, it is evident that the 9 B.L.T (Boundary Layer thickness), fine mesh, and Last Aspect Ratio inflating option yielded favorable outcomes when compared to the experimental data for lift prediction.

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

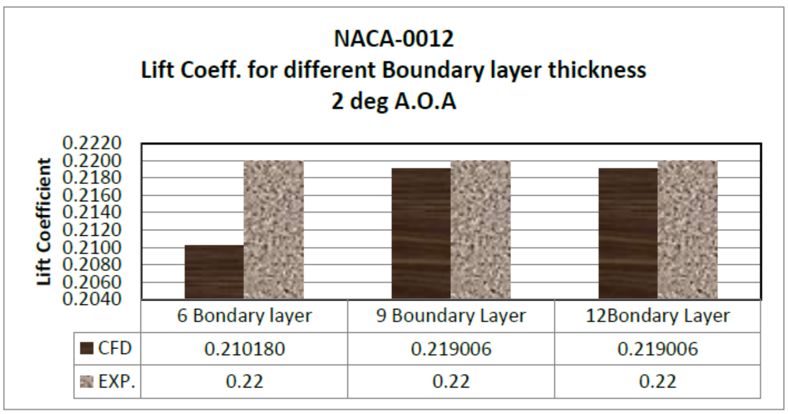

Figure 4: Lift coefficients for different Boundary layer thickness.

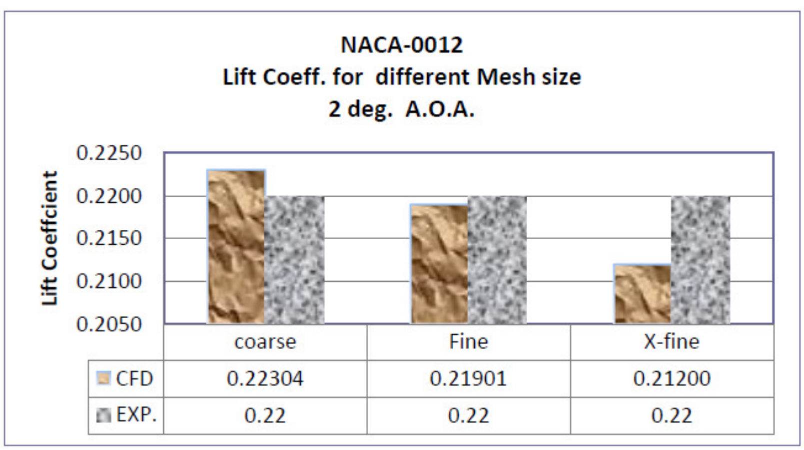

Figure 5: Lift coefficients for different mesh size.

4. CONCLUSIONS

The primary factor to consider in aerodynamic design is the precision of the turbulent model used to simulate intricate turbulent flows. In addition to computer and numerical simulation approaches, the field of turbulence modeling has made significant advancements in the last decade to effectively address the complexities of studying modern aerodynamic systems. The validity of the Cut Cell numerical approach was established by applying it to three-dimensional viscous steady incompressible flow using the NACA-0012 airfoil. The current findings for NACA-0012 were calculated using two accessible models in ANSYS FLUENT: sst k- ω and transition k-kl omega models. The research demonstrated that the Cut Cell mesh approach had the capability to produce a mesh of excellent quality, effectively capturing the intricate features of the viscous boundary layer. Subsequent research will prioritize the use of the Cut Cell Method for advanced turbulence models and mesh refinement in order to forecast the lift coefficients of high lift devices with flaps.

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 Cartesiangrids, 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 cellmethod 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, AIAAA-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, 2nd edition.
31. J. H. Ferziger and M. Peric, Computational Fluid Dynamics, 3rd 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-ansyssoftware.
34. D. K. Walters and D. Cokljat, A three-equation eddy-viscosity model for reynoldsaveraged navier-stokes simulations of transitional flows. J. of Fluids Eng. 130(8) 2008.