

An Investigation of CutCell Meshing Strategies for Accurate Aerodynamic Performance Prediction

Ramadan Elhadi Ghmati and Badih Ali Jawad
Lawrence Technological Univ.

Evangelos Koutsavdis
ANSYS Inc.

ABSTRACT

With advances in computing power and Computational Fluid Dynamics (CFD) algorithms, the complexity of CutCell based simulation models has significantly increased. In this study three dimensional numerical simulations were created for steady incompressible flow around airfoil shape. The NACA-0012 airfoil was used for this study. Boundary layer thickness, mesh expansion ratio, and mesh density variation parameters were investigated. Drag and lift coefficients were compared to experimental data. Use of the CutCell method results in a good agreement between CFD results and experimental data.

CITATION: Ghmati, R., Jawad, B. and Koutsavdis, E., "An Investigation of CutCell Meshing Strategies for Accurate Aerodynamic Performance Prediction," *SAE Int. J. Passeng. Cars - Mech. Syst.* 5(1):2012, doi:10.4271/2012-01-0499.

1. INTRODUCTION

With advances in computing power and computational Fluid Dynamics (CFD) algorithms, the complexity of CutCell simulation models has significantly increased. The current study is focused on the effect of different CutCell meshing strategies on the accuracy of aerodynamic performance prediction. The Cartesian method can be described as a methodology in which CutCells are applied on the geometry resulting from a process involving rectangular/hexagonal cells on regular lattice cutting through the geometry[37].

The work of Purvis and Burkhalter in 1979 [1], Wedan and South in 1983 [2] marks the advent of the Cartesian CutCell method. Their work involved using the finite volume method for the full potential equations. In each cell, the process of cutting the geometry is based on a linear piecewise cut. Clarke et al. [3] extended this work to the 2D Euler equations. Their 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. However, the agreement was poor as the grid resolution was deemed too coarse. The Euler finite volume method was extended to 3D by Gaffney, Salas and Hassan [4]. Their extended approach continued to use the linear cuts and small volume cut cell agglomeration procedure.

Later, significant interest in the adaptive Cartesian Cut algorithms for problems associated with complex geometries is seen. In this regard, Aftosmis et al [5] 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. The mesh generation procedure was evaluated on a wide spectrum of complex geometries. Yang et al. [6, 7] suggested a methodology for computing compressible flows. The algorithm uses the Cartesian CutCell approach in conjunction with multidimensional 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. The authors qualified the method using a host of test cases that cover the cases of steady and unsteady compressible flows. Yang et al in [8], extended the work reported in [5] and [6]; specifically, the Cartesian CutCell method to the three dimensional case. Various types of CutCells are used to accurately represent the geometry. The extended method was tested using a collection of test cases that include steady and unsteady compressible flows that involve static and moving bodies in three dimensions.

In 2000, Tucker and Pan [9] applied the Cartesian CutCell 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. Decent agreement was observed relative to analytical, numerical and experimental data. Shortly afterwards, Causon et al. [10], reported a new approach in which the focus was on calculating shallow water flows in the presence of moving physical boundaries. Various types of CutCells represented the boundaries of solid regions which are cut out off a background Cartesian grid. The 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. The concept behind this approach is simple and it involves cutting solid bodies out of a background Cartesian mesh. Although the approach was intended for potential flow, the authors in [2, 3] and in [9] were able to successfully apply it to the shallow water equations in two and three dimensions. In [11], Murnam et al. proposed a supersonic missile system with a small number of synchronous canard control surfaces. An automated inviscid Cartesian method is used in designing the missile system. Simulations for total motion are 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. In [12] Wang et al., proposed a finite volume methodology to tackle electromagnetic wave dynamics in the time domain. The authors presented an argument in which they claimed that the adaptive Cartesian grid is more effective in terms of both accuracy and efficiency relative to the tetrahedral grid in cases where the associated geometry is complex. A host of test cases with existing analytical solutions were used to qualify the computational electromagnetic solver. Murnam et al [13] proposed a methodology to simulate impermeable boundaries within a fixed Cartesian mesh which is based on the Cartesian boundary scheme presented in [5]. The scheme lowers the frequency at which the geometry is intersected with the Cartesian volume within a full simulation. Special emphasis is paid to the complexities arising in higher dimensions. Ingram et al. [14] reported a Cartesian CutCell based method which is regarded as an alternative to the traditional boundary fitted grid methods. In this Cartesian approach, a background Cartesian grid is used to handle the majority of the flow domain. The method was applied to problems in which moving bodies and moving material interfaces were involved.

To the best of our knowledge, not much work has been done involving the full cell based methods for full Navier Stokes equations. In 1994 Tau [32] 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. [15] to solve the Navier-Stokes equations on a non-uniform staggered grid in three dimensions involving curved boundaries. The authors proposed a new cell-linking method in which small cell creation difficulties are alleviated while complexities

associated with other cell-merging schemes are avoided. The geometric information needed by the scheme is based on a quadric surface representation of the boundaries which are generated via techniques developed for this purpose. The methodology is 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. In [16], Dorge and Verstappen presented a method to solve the unsteady incompressible Navier-Stokes equations, in which the problem domain is of arbitrarily shaped boundaries. The symmetry of convection and diffusion is maintained in their new approach to handle the CutCell discretization. The method was examined using an incompressible unsteady flow around a circular cylinder in which Re is 100. Rosatti et al. [17], presented methods based on radial basis functions associated with a high degree of approximation. It has been reported that these methods are used to tackle the extension of semi-Lagrangian shallow water models to staggered Cartesian meshes in the presence of cut boundary cells. Open channel flow simulation was carried out to demonstrate the degree of accuracy and efficiency of the resulting semi-Lagrangian method.

Shortly afterward, Rosatti et al. [18] suggested extending the shallow water semi-implicit models on staggered Cartesian meshes to account for the existence of CutCells at the computational space boundaries. A host of simulations involving nonlinear open channel flows and baroclinic flows over an isolated obstacle were carried out to assess the accuracy of the environmental flow models. Chung [19], reported a Cartesian grid based methodology with CutCells intended for simulating two dimensional unsteady viscous incompressible flows associated with arbitrarily shaped rigid bodies. The proposed CutCell approach coupled with cell merging process provides a scheme which guaranteed conservation of global mass/momentum and simultaneously eliminates the need for small size control volume. The method was qualified using test examples that included moving body problems. Ji et al. [20] 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. Numerical solutions seem to indicate that the algorithm accuracy is second order which supports the authors claim.

Among the well-known problems in acoustics, is the sound generation from vibrating circular pistons. Popsecu et al. [21] attempted to simulate the sound waves generated by oscillating baffled pistons. In their method, both linear and nonlinear models are attempted to account for the interplay between the propagation of the sound waves and the geometric complexities. The wave equations in Cartesian coordinates along with cut cells are used along a compact finite volume scheme to implement spatial discretization. The authors also posed a whole range of problems for waves around various geometries. Sang and Li [22], presented a

methodology for handling the computations associated with the complex flow fields around three dimensional high lift configurations. The adaptive Cartesian grid method in conjunction with a cell center finite volume procedure was adopted in their methodology. The performance of the methodology was compared with experimental data. Hsu [23], examined the numerical performance pertaining to the explicit Cartesian methodologies in the context of compressible flow. The author addressed many methodology properties including simulation accuracy, stability, and issues related to grid convergence stemming from a moving impermeable boundary cutting across the background mesh. Pattenson et al. [24] and Pattenson [33], 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. Full approximation storage (FAS) multi-grid, which involves automatic generation of coarse meshes through volume agglomeration methodology is adopted to accelerate the solution process. The results revealed that the standard discretization and the other developed alternative methods seem to be equivalent as far as accuracy and computational cost are concerned.

Fidkowski and Darmofal [25, 35, 37] suggested a method that is based on mesh adaptation to handle higher-order discontinuous Galerkin discretizations associated with two-dimensional compressible Navier-Stokes equations. A key advantage of this methodology is the advent of a cut cell meshing technique in which no restrictions are imposed on the triangles to force them to conform to the boundary. In [26], Ji et al. suggested a Cartesian grid based methodology intended to handle the moving boundary problem in which the domain is complex. In this approach, a surface integral is used to evaluate gradient fluxes across all sections of the boundary curve intersecting a cut-cell. To qualify this algorithm, six test cases are used including both fixed and moving boundary problems. Validation of the extended approach is demonstrated using a test case of a three-dimensional free surface problem. In [27], Qian proposed a method based on a two fluid formulation of the Navier-Stokes equations and a Cartesian cut cell method. The method involves a new free surface capturing code to investigate the interaction between wave and floating body related problems. Barad et al. [28], suggested a numerical methodology to solve incompressible Navier-Stokes equations in the context of environmental fluid mechanics. The method is intended to examine environmental flows associated with multi-scale incompressible and variable density within possibly anisotropic complex domains. The accuracy of the method is demonstrated by presenting 2D and 3D numerical results.

Recently, Popsecu et al [29], suggested techniques for computational aero acoustics that possess the capability of generating low artificial dispersion and dissipation to sustain the level of amplitude and frequency characteristics associated with physical processes.

An optimized pre-factored compact finite volume scheme along with a Cartesian cut-cell procedure is developed by the authors to address the afore-mentioned issues. The scheme ensures the satisfaction of the conservation laws while attempting to minimize the dispersion and dissipation. Hartmann et al. in [30], presented a full description in which they covered how the multi-level method coupled with a Cartesian CutCell procedure is applied. Fujimoto et al. [31] suggested a method to enhance the efficiency of the body-fitted Cartesian grid procedure. The grid front was generated via Cartesian grid generation over the ghost surface. Consequently the handling of complicated geometry become more flexible. Li et al. [34], presented a methodology in which they extended their previous work related to a large eddy simulation (LES) model with the intention of comprehensive validation. Nemec and Aftosmis [36] reported a new approach in which they focused on shape sensitivities computations using discrete adjoint and flow sensitivity methods on Cartesian meshes. The Cartesian meshes considered involve general polyhedral cells at the wall boundaries. Shape sensitivities associated with problems governed by Euler equations were efficiently and robustly computed via directly linearizing geometric constructors of the CutCells. An exact solution model problem was used to verify the accuracy of the linearization scheme.

2. GOVERNING EQUATIONS

The major issues in high lift design are accurately modeling turbulence to represent flow with extensive separations and the difficulties associated with modeling the flow near the airfoil edge and trailing viscous wakes.

To date, there has been relatively little work focusing on the Cartesian CutCell method for the full Navier-Stokes equations. In the present work, the authors implemented a CutCell 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 collocate grid is used.

The following highlights the mathematical of continuity and Navier-Stokes formulae for Incompressible fluid:

$$\frac{\partial(\overline{u_j})}{\partial x_j} = 0$$

$$\frac{\partial(\overline{u_i u_j})}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \overline{P}}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\nu \left(\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i} \right) - \overline{u'_i u'_j} \right]$$

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

- 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 ϕ using information at three time levels, $n+1$, n and $n-1$ written as [15]:

$$\begin{aligned} A_p^{n+1} \phi_p^{n+1} = & \sum_{nb} (A_{nb}^{n+1} \phi_{nb}^{n+1}) + S_{imp} \phi_p^{n+1} + \\ & S_{exp} \phi_p^{n+1} + [\sum_{nb} (A_{nb}^n \phi_{nb}^n) - A_p^n \phi_p^n + S_{imp} \phi_p^n + \\ & S_{exp} \phi_p^n] + [\sum_{nb} (A_{nb}^{n-1} \phi_{nb}^{n-1}) - A_p^{n-1} \phi_p^{n-1} + S_{imp} \phi_p^{n-1} + S_{exp} \phi_p^{n-1}] \end{aligned}$$

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 , y , z 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 CutCell Method. CutCell Cartesian meshing is a general purpose meshing designed for ANSYS FLUENT. Make use of workbench to construct the airfoil shape and the CutCell mesh. For a very simple mesh, a mesh specifications 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, simulation construction 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 CutCell meshing is a patch independent volume meshing approach without the need for manual geometry cleanup or decomposition, thereby reducing the turnaround time required for meshing. The CutCell algorithm is suitable for a large range of applications, and due to the large fraction of hex cells in the mesh, often produces better results than tetrahedral methods. The general approach of the CutCell mesh process involves defining the size function values which are computed. The grid is then adaptively refined based on the local size function values. The cells intersected by the geometry are marked, only marked cells are considered for projection. The edges intersected by the geometry are identified, mesh edges to be recovered are determined and used to construct mesh faces. Once the mesh faces are identified, cells are decomposed to recover these faces see Figure (1). The boundary mesh is recovered and separated on the underlying geometry. The next step in the CutCell meshing workflow is to set global inflation controls. If you were to set global inflation controls after the CutCell mesh had been generated, a re-mesh of the CutCell mesh would be triggered. For this reason, it is important to set global inflation controls in advance of CutCell mesh generation [39].

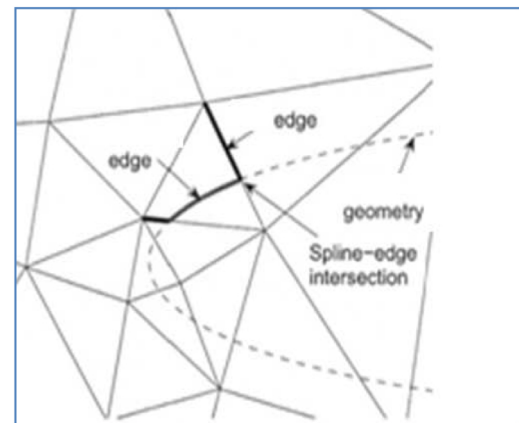


Figure (1). Intersection between a background mesh and an airfoil [37].

The CutCell method is useful for meshing fluid bodies in single body parts and multi-body parts. However, it cannot be used to mesh assemblies of parts. CutCell's may encounter problems capturing very acute internal and external face angles, such as the trailing edges of fins. 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 CutCell mesh of a computational domain obtained from the original background mesh by removing elements completely