Numer. Math. Theor. Meth. Appl.
doi: 10.4208/nmtma.OA-2019-0143

Vol. **13**, No. 4, pp. 881-907
November 2020

Numerical Solution of Partial Differential Equations in Arbitrary Shaped Domains Using Cartesian Cut-Stencil Finite Difference Method. Part I: Concepts and Fundamentals

M. Esmaeilzadeh[1], R.M. Barron[1,2,*] and R. Balachandar [1,3]

[1] *Department of Mechanical, Automotive & Materials Engineering, University of Windsor, Windsor, Ontario, Canada, N9B 3P4*
[2] *Department of Mathematics & Statistics, University of Windsor, Windsor, Ontario, Canada, N9B 3P4*
[3] *Department of Civil & Environmental Engineering, University of Windsor, Windsor, Ontario, Canada, N9B 3P4*

Received 9 September 2019; Accepted (in revised version) 23 February 2020

Abstract. A new finite difference (FD) method, referred to as "Cartesian cut-stencil FD", is introduced to obtain the numerical solution of partial differential equations on any arbitrary irregular shaped domain. The 2^{nd}-order accurate two-dimensional Cartesian cut-stencil FD method utilizes a 5-point stencil and relies on the construction of a unique mapping of each physical stencil, rather than a cell, in any arbitrary domain to a generic uniform computational stencil. The treatment of boundary conditions and quantification of the solution accuracy using the local truncation error are discussed. Numerical solutions of the steady convection-diffusion equation on sample complex domains have been obtained and the results have been compared to exact solutions for manufactured partial differential equations (PDEs) and other numerical solutions.

AMS subject classifications: Primary Classification: 65N06, Secondary Classification: 35Q35

Key words: Cartesian cut-stencils, finite difference method, irregular domains, convection-diffusion equation, local truncation error.

1. Introduction

Many problems in engineering and science can be modeled as initial or boundary value problems using second-order partial differential equations (PDEs), as represented for example by the unsteady Navier-Stokes equations in fluid mechanics,

*Corresponding author. *Email address:* `az3@uwindsor.ca` (R.M. Barron)

Cauchy-Navier's equations in elasticity, the Laplace equation in heat transfer and Maxwell's equations in electromagnetism. Generally speaking, there are three well-established mesh-based space discretization methods for the numerical solution of such PDEs, namely the finite difference (FD) [20], finite volume (FV) [42, 60] and finite element (FE) [13, 59] methods.

Finite difference methods were first introduced by Richardson [45] in his study of masonry dams, and later in numerical weather prediction. Although many researchers have successfully applied the finite difference method to a wide range of problems, particularly in fluid mechanics, and it continues to be implemented in codes used for academic research, it has not gained much popularity in commercial PDE solver codes, including computational fluid dynamics (CFD) software. In order to use FD for very complex geometries, the solution domain must be decomposed into a set of sub-domains that are transformable to rectangular blocks in a computational domain [27, 56, 63]. The process of generating a good quality structured grid on a highly complex domain for a FD code can be a time-consuming and costly component of an overall simulation, and requires a high level of expertise [37, 57]. However, particularly for industrial applications, it is inevitable that the PDEs have to be solved in highly complex domains. This is the main reason that, for commercial purposes, researchers and code developers have largely abandoned the FD approach.

Finite volume (FV) methods originated in the 1970's and gained popularity after the seminal work of Patankar and Spalding [43]. There is now a vast literature on development, enhancements and applications of the finite volume methodology. FV methods enjoy flexibility in meshing, being able to handle cells of arbitrary shape and not suffering from the structured mesh restriction of the FD method. Because of this flexibility, FV methods can treat flows in highly complex domains like those encountered in many industrial applications. This is the primary reason that most of the commercial computational fluid dynamics (CFD) codes today are based on the FV formulation.

The finite element (FE) method was originally developed to solve problems in elasticity and structural mechanics. Key components of the FE method can be found in the early works of Hrenikoff [30] and Courant [15], but formalization of the method is due to Turner *et al.* [59] and Argyris and Kelsey [5]. The phrase "finite element" was coined by Clough [13] in 1960. FEM has now become an alternative to the FV formulation in fluid mechanics and multiphysics simulations with the ability to handle arbitrarily complex domains.

Among these methods, FD is the simplest to understand and therefore is still often used to explain some key numerical concepts such as order of accuracy, stability and convergence [20, 29].

It is well-known that properly designing the mesh is a critical factor in obtaining an accurate numerical solution of a PDE. Compared to the use of structured body-fitted curvilinear grids and unstructured tetrahedral or polyhedral meshes, a Cartesian grid system is simple and easy to construct [2, 54]. Furthermore, solutions on Cartesian grids tend to converge better than those of body-fitted methods, and Cartesian grid methods generally require less storage. Even during the heyday of the development