ICIAM 2019 SEMA SIMAI Springer Series 3

Ralf Deiterding
Margarete Oliveira Domingues
Kai Schneider *Eds.*

Cartesian CFD Methods for Complex Applications

Springer

SEMA SIMAI Springer Series

ICIAM 2019 SEMA SIMAI Springer Series

Volume 3

This sub-series of the SEMA SIMAI Springer Series aims to publish some of the most relevant results presented at the ICIAM 2019 conference held in Valencia in July 2019.

The sub-series is managed by an independent Editorial Board, and will include peer-reviewed content only, including the Invited Speakers volume as well as books resulting from mini-symposia and collateral workshops.

The series is aimed at providing useful reference material to academic and researchers at an international level.

More information about this subseries at http://www.springer.com/series/16499

Ralf Deiterding • Margarete Oliveira Domingues •
Kai Schneider
Editors

Cartesian CFD Methods for Complex Applications

Springer

Editors
Ralf Deiterding
Department of Aeronautics & Astronautics
University of Southampton
Southampton, UK

Margarete Oliveira Domingues
Associate Laboratory of Computing and Applied Mathematics
National Institute for Space Research (INPE)
São José dos Campos
Brazil

Kai Schneider
Institut de Mathématiques de Marseille (I2M)
Aix-Marseille Université
Marseille, France

ISSN 2199-3041 ISSN 2199-305X (electronic)
SEMA SIMAI Springer Series
ISSN 2662-7183 ISSN 2662-7191 (electronic)
ICIAM 2019 SEMA SIMAI Springer Series
ISBN 978-3-030-61760-8 ISBN 978-3-030-61761-5 (eBook)
https://doi.org/10.1007/978-3-030-61761-5

Preface

Cartesian discretization approaches are ubiquitous in computational fluid dynamics. When applied to problems in geometrically complex domains or fluid–structure coupling problems, Cartesian schemes allow for automatic and scalable meshing; however, order-consistent immersed boundary conditions and efficient dynamic mesh adaptation take forefront roles. This volume contains selected contributions from the four-session thematic mini-symposium on "Cartesian CFD Methods for Complex Applications" at ICIAM 2019 held in Valencia in July. The papers highlight cutting-edge applications of Cartesian CFD methods and describe the employed algorithms and numerical schemes. An emphasis is laid on complex multi-physics applications such as magnetohydrodynamics or aerodynamics with fluid–structure interaction, solved with various discretizations, e.g. finite difference, finite volume, multi-resolution or lattice Boltzmann CFD schemes. Software design and parallelization challenges are also addressed briefly.

The volume is organized into two parts of three contributions each. Part one is focused on incompressible flows and has the following contributions: Bergmann et al. propose an adaptive finite-volume method with quad-tree discretization of the incompressible Navier–Stokes equations. Moving immersed bodies are modelled with volume penalization, and their interface is tracked using level sets. Test cases with flows around cylinders show the validity and precision of the approach. Fluid–structure interaction for flexible insect wings is studied in the paper by Truong et al. A mass spring model is used for the wing structure. The fluid solver is based on a Fourier pseudospectral discretization with volume penalization to take into account the complex and time-varying geometry. Applications consider flapping bumblebee flight in laminar and turbulent flow. The paper by Kadri and Perrier presents a numerical scheme for incompressible Navier–Stokes equations in three dimensions using divergence-free wavelets. Constructions for these basis functions are given for no-slip and free-slip boundary conditions and divergence-free wavelets in dimension higher than three are given. Numerical examples illustrate the scheme for lid-driven cavity problems.

The second part deals with compressible and weakly compressible flows and has likewise three contributions. Perron et al. propose an immersed boundary method

for compressible flows using structured Cartesian grids. A direct forcing approach based on the use of ghost cells is chosen. Two flow configurations are considered, a supersonic flow around a blunt body to demonstrate the capability of mesh adaptation to increase the accuracy and a large eddy simulation of the flow around a three-dimensional high-lift airfoil. Comparisons with experimental data and a reference body-fitted computation are as well presented. Moreira Lopes et al. discuss the performance and detail verification and validation of a wavelet-adaptive magnetohydrodynamic solver, realized within the MPI-parallel AMROC (Adaptive Mesh Refinement in Object-oriented C++) framework. A prototype simulation fuses this solver with actual satellite date for space weather forecasting. Finally, Gkoudesnes and Deiterding report on the incorporation of the lattice Boltzmann method into the AMROC environment. The algorithmic details and verification of large eddy simulation with the wall-adapting local eddy-viscosity model for dynamically adapting meshes and with ghost cell-based embedded boundary conditions are presented.

We thank all the speakers of the four sessions for making this mini-symposium a successful event, and we are grateful to the authors for their contributions. We are indebted to the numerous referees for their constructive and detailed reports. For all papers, we had three to four reviews, improving thus further the quality of this edited volume.

Southampton, UK — Ralf Deiterding
São José dos Campos, Brazil — Margarete Oliveira Domingues
Marseille, France — Kai Schneider
May 2020

Contents

AMR Enabled Quadtree Discretization of Incompressible Navier–Stokes Equations with Moving Boundaries 1
Michel Bergmann, Antoine Fondanèche, and Angelo Iollo

Fluid–Structure Interaction Using Volume Penalization and Mass-Spring Models with Application to Flapping Bumblebee Flight 19
Hung Truong, Thomas Engels, Dmitry Kolomenskiy, and Kai Schneider

No-Slip and Free-Slip Divergence-Free Wavelets for the Simulation of Incompressible Viscous Flows 37
Souleymane Kadri Harouna and Valérie Perrier

An Immersed Boundary Method on Cartesian Adaptive Grids for the Simulation of Compressible Flows 67
S. Péron, T. Renaud, C. Benoit, and I. Mary

Magnetohydrodynamics Adaptive Solvers in the AMROC Framework for Space Plasma Applications 93
Müller Moreira Lopes, Margarete Oliveira Domingues, Ralf Deiterding, and Odim Mendes

Verification of the WALE Large Eddy Simulation Model for Adaptive Lattice Boltzmann Methods Implemented in the AMROC Framework 123
Christos Gkoudesnes and Ralf Deiterding

About the Editors

Ralf Deiterding is currently Associate Professor in Fluid Dynamics at the University of Southampton. He graduated with a Master's in Technomathematics from the Technical University of Clausthal and has obtained a PhD in Applied Mathematics and CFD from the Technical University Cottbus. His research focuses on the development and application of innovative high-resolution and multi-scale simulation methods for CFD. He is the main author of the simulation frameworks AMROC and Virtual Test Facility.

Margarete Oliveira Domingues is senior researcher at the National Institute for Space Research (INPE), Brazil. She obtained her Master's in Meteorology at INPE, and PhD degree in Applied Mathematics from the University of Campinas (UNICAMP), Brazil. Her research activities are focused on nonlinear analysis, multi-scale techniques and wavelets for scientific computing and their application to space flows and data, including space magnetohydrodynamic developments and multi-dimensional signal processing.

Kai Schneider is Professor of Mechanics and Applied Mathematics at the Aix-Marseille University, Marseille, France. He obtained his Master's and PhD degree from the University of Kaiserslautern, Germany, and his habilitation from the University Strasbourg, France. His research activities are focused on multi-scale techniques and wavelets for scientific computing and their application to turbulent flows, including fluid–structure interaction, e.g. for insect flight, and magnetohydrodynamic turbulence.

AMR Enabled Quadtree Discretization of Incompressible Navier–Stokes Equations with Moving Boundaries

Michel Bergmann, Antoine Fondanèche, and Angelo Iollo

Abstract We present a versatile finite-volume method for the simulation of incompressible flows past moving bodies. The Navier–Stokes equations are discretized on AMR enabled quadtree grids, where the dynamic in time refinement is adapted to the evolution of the fluid–solid system. The immersed bodies are modeled through a second-order volume penalization method, and the interface is tracked using a level-set description. We highlight on two dimensional test cases that the uniform grids accuracy can be recovered using quadtree grids with less degrees of freedom.

1 Introduction

Efficient numerical tools to simulate fluid–solid interactions are in interest in a wide range of application fields, from engineering to medical applications. For instance, the simulation of a flow around a wind turbine blade [1] or in cardiac support devices [2] is an essential support to optimize the design of these new technologies.

To face this challenge, a large number of studies have been carried out to precisely describe these interactions, especially when dealing with complex geometries. These studies are based on two numerical approaches. The first approach is based on the Arbitrary Lagrangian–Eulerian methodology for which flows are calculated on a moving mesh in a time-varying area (see [3] for details). These methods are generally very accurate, based on sophisticated numerical schemes, but are difficult to implement, especially for the consideration of structures with large deformations. The generation of a body-fitted mesh is expensive, and the use of a dynamic mesh partitioner for parallel calculations is moreover necessary. The second approach is based on fictitious domain methods, such as immersed boundary

M. Bergmann · A. Fondanèche (✉) · A. Iollo
Equipe-Projet Memphis, INRIA Bordeaux-Sud Ouest, Talence, France

Université de Bordeaux, IMB, UMR 5251, Talence, France
e-mail: michel.bergmann@inria.fr; antoine.fondaneche@inria.fr; angelo.iollo@inria.fr

R. Deiterding et al. (eds.), *Cartesian CFD Methods for Complex Applications*,
SEMA SIMAI Springer Series 3, https://doi.org/10.1007/978-3-030-61761-5_1

methods [4] or augmented Lagrangian approaches [5], which represent a balance between precision and practicability of the simulation.

In this work, we use the Brinkman penalization method [6] that is an embedded interface method such as the immersed boundary method (IBM, introduced by Peskin [7, 8]) with discrete forcing. In the context of interface-capturing methods for simulating multiphase flows, such as volume-of-fluid [9], phase field [10], or level-set [11] descriptions, the whole system is strongly coupled as soon as both materials are subject to the same constitutive equation. Here we consider a level-set formulation with the sign distance function, where the fluid–solid interface is defined by the zero isoline. Cartesian methods for incompressible flows [12–14] need a very refined mesh to get accurate results because they need a good representation of the body geometry. With respect to these methods, we propose a quadtree-based method that provides an equivalent accuracy with a smaller number of grid points. By refining the mesh in regions of interest, such as in the vicinity of the interface or where the solution varies significantly and by coarsening where the solution varies slightly, the computational time is significantly decreased with a limited loss of accuracy.

2 The Penalized Navier–Stokes Model

The aim of this work is to study the interaction between an incompressible Newtonian fluid and some rigid moving bodies. A square domain $\Omega \subset \mathbb{R}^2$ is decomposed into two parts, namely $\Omega = \Omega_f \cup \Omega_s$, where Ω_f and Ω_s denote the fluid and solid domains, respectively. The fluid–solid interface is $\Gamma(t) = \partial\Omega_s(t)$. The sketch of the flow configuration is presented in Fig. 1. The governing equations are

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u}\cdot\nabla)\mathbf{u} = -\frac{1}{\rho}\nabla p + \nu\Delta\mathbf{u} \qquad \text{in } \Omega_f, \tag{1a}$$

$$\nabla\cdot\mathbf{u} = 0 \qquad \text{in } \Omega_f, \tag{1b}$$

$$\mathbf{u} = \mathbf{u}_s \qquad \text{on } \Gamma(t), \tag{1c}$$

$$\mathbf{u}(t=0,\cdot) = \mathbf{u}_0 \qquad \text{in } \Omega_f, \tag{1d}$$

where $\mathbf{u} = (u,\ v)^T$ is the velocity field, p is the pressure, ρ is the density, and ν is the kinematic viscosity of the fluid. Finally, $\mathbf{u}_0$ is the initial condition and $\mathbf{u}_s$ is the velocity of the body interface.

The volume penalization approach introduced in [6] is chosen. The main idea of this method is to consider the whole system as porous media, with a variable permeability ε. The solid structure is considered to have a very low permeability

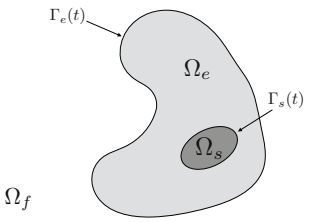

Fig. 1 Sketch of the flow setup

$\varepsilon \ll 1$. The Navier–Stokes equations (1a), (1b), and (1c) can thus be solved in a coupled way in Ω as

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla)\mathbf{u} = -\frac{1}{\rho}\nabla p + \nu \Delta \mathbf{u} + \frac{\chi_s}{\varepsilon}(\mathbf{u}_s - \mathbf{u}) \tag{2a}$$

$$\nabla \cdot \mathbf{u} = 0, \tag{2b}$$

where χ_s is the characteristic function of the solid, defined as

$$\chi_s(\mathbf{x}) = \begin{cases} 1 & \text{if } \mathbf{x} \in \overline{\Omega}_s \\ 0 & \text{if } \mathbf{x} \in \Omega_f. \end{cases}$$

The position of the solid Ω_s is tracked with a level-set function ϕ, chosen as the signed distance to the interface $\Gamma(t)$ with a negative sign inside the solid and a positive sign inside the fluid. The interface is then defined as the zero isoline of ϕ, namely, $\Gamma(t) = \{\mathbf{x} \in \mathbb{R}^2 : \phi(x) = 0\}$. As a consequence, the characteristic function χ_s may be defined using the level-set function as $\chi_s(\mathbf{x}) = 1 - H(\phi(\mathbf{x}))$, where H is the Heaviside function. Using this method, both solid and fluid equations are solved, with no distinction. The solution of the system (2) converges towards the solution of the decoupled system (1) (see [6]) as $\sqrt{\varepsilon}$ tends to zero [15]. In practice, the choice $\varepsilon = 10^{-10}$ is suitable for all our simulations.

3 Discretization of the Governing Equations

3.1 Time Integration

We denote by Δt the time step such that $t^{n+1} = t^n + \Delta t$ and $\varphi^n := \varphi(t^n)$ the discrete value of a function φ at the time t^n. For the sake of simplicity, the time

step is assumed to be fixed here, but the generalization to an adaptative time step is straightforward.

We consider the fractional time step method introduced by Chorin [16] and Temam [17]. First, a prediction step is performed to get a preliminary estimate $\mathbf{u}^*$ of the velocity starting from a guess for the pressure field q. This is done using a second-order Gear scheme as

$$\begin{aligned}\frac{3\mathbf{u}^* - 4\mathbf{u}^n + \mathbf{u}^{n-1}}{2\Delta t} + \left(2\nabla\cdot(\mathbf{u}\otimes\mathbf{u})^n - \nabla\cdot(\mathbf{u}\otimes\mathbf{u})^{n-1}\right) \\ = \frac{1}{\rho}\Big(- \nabla q + \mu\Delta\mathbf{u}^* + \frac{\chi_s^{n+1}}{\epsilon}(\mathbf{u}_s^{n+1} - \mathbf{u}^*)\Big).\end{aligned} \tag{3}$$

We use here the incremental version of the projection method proposed by Goda [18] for which $q := p^n$. A second-order accurate volume penalization method is employed as in [12], for which the velocity inside the body $\mathbf{u}_s$ is artificially imposed by image point correction (IPC). As long as the interface does not fit the grid points, this technique ensures that the velocity of the interface $\mathbf{u}|_{\partial\Omega_s}$ is enforced, and therefore, the velocity gradient in the first layer of fluid is consistent.

Since the predicted velocity field is not divergence free, a projection step in a solenoidal subspace is performed

$$\frac{\mathbf{u}^{n+1} - \mathbf{u}^*}{\Delta t} = -\frac{1}{\rho}\left(\nabla p^{n+1} - \nabla q\right) \quad \text{in } \Omega. \tag{4}$$

Since we want to recover a divergence-free velocity, i.e., $\nabla\cdot\mathbf{u}^{n+1} = 0$, by applying the divergence operator to Eq. (4), we get

$$\Delta p' = \nabla\cdot\mathbf{u}^* \quad \text{in } \Omega, \tag{5}$$

where we denote by $p' := \frac{\Delta t}{\rho}(p^{n+1} - q)$ the pressure increment. Homogenous Neumann boundary conditions are imposed in order to ensure that there is no perturbation at the boundaries for the normal velocity.

As soon as the increment of pressure p' is determined by solving the Poisson equation (5), the pressure p^{n+1} can be updated and the velocity field is corrected as follows:

$$p^{n+1} = q + \frac{\rho}{\Delta t}\, p', \tag{6a}$$

$$\mathbf{u}^{n+1} = \mathbf{u}^* - \nabla p'. \tag{6b}$$

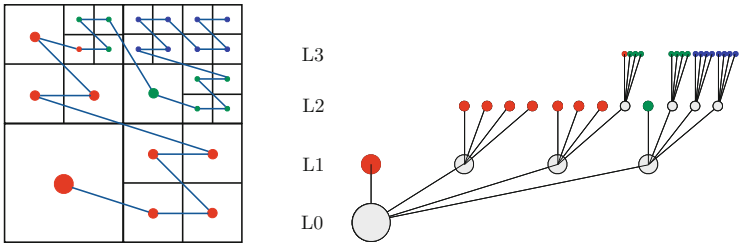

Fig. 2 Graded quadtree grid with the global Z-ordering. The different colors depicts the balancing between processors

3.2 Spatial Discretizations

The computational domain Ω is discretized with a quadtree grid. As depicted in Fig. 2, a quadtree grid is composed of square cells with different levels of refinement. Here, the hierarchical grid is graded, which means that the difference of level between a cell and all its adjacent cells (called *neighbors*) is at most one. Thanks to the library PABLO, as a part of Bitpit library,[1] we get use of an efficient tool for storing the data structure. Following the linear Z-ordering proposed by Morton in 1966 [19], we can get access to data coming from neighboring cells in an optimized way from computational cost and memory aspects. Moreover, Adaptative Mesh Refinement (AMR) is used to adapt the mesh dynamically to the flow configuration by refining in the areas of interest, such as wakes of bodies, vortices, or around the structures, which is even more interesting when the structures can move or be deformed. For the domain decomposition, the number of communications between processors is limited to only one layer of ghost cells. While this constraint guarantees a very high scalability of the parallelism, the discretizations of the operators are built with compact stencils limiting the order of numerical scheme.

In this section, we detail the finite-volume discretizations of the operators involved. To describe these discretizations, let φ be a scalar function and $\mathbf{v}$ a vector field. The square domain Ω is decomposed into a quadtree partition of N_{cells} square cells Ω_i of level L_i (being the *leaves* of the tree) such that $\Omega = \dot{\bigcup}_i \Omega_i$. By convention, the grid configuration is identified by its minimum and maximum levels of refinement L_{min} and L_{max}. In other words, for a $L_{min} - L_{max}$ grid, the characteristic length h_i of Ω_i is between $h_{min} = \min_k h_k$ and $h_{max} = \max_k h_k$. A two dimensional uniform L^ℓ grid is hence composed of $2^{2\ell}$ cells. We denote by $\mathbf{x}_i$ the center of the cell Ω_i, $|\Omega_i|$ its area, and $\varphi_i := \varphi(\mathbf{x}_i)$ the discrete value of a quantity φ evaluated at the cell center $\mathbf{x}_i$.

For a finite-volume method, the discrete operators are computed as face contributions called fluxes. Let f be the intersecting face of Ω_{out} and Ω_{in}. The length of

[1] https://optimad.github.io/bitpit.

f is denoted by $|f|$. As a convention, the normal vector $\mathbf{n}_f$ of f is pointing from Ω_{in} to Ω_{out}. The discrete values of φ in Ω_{in} and Ω_{out} are denoted by φ_{in} and φ_{out}, respectively.

3.2.1 Discretization of the Divergence Operator

The divergence operator is integrated over each cell Ω_i. Using the Stokes theorem, the volume integral is transformed into a surface integral as

$$\nabla \cdot \mathbf{v}\big|_{\Omega_i} = \frac{1}{|\Omega_i|} \int_{\Omega_i} \nabla \cdot \mathbf{v} \, d\mathbf{x} = \frac{1}{|\Omega_i|} \oint_{\partial\Omega_i} \mathbf{v} \cdot \mathbf{n} \, ds, \tag{7}$$

where $\mathbf{n}$ is the outward normal vector of the boundary $\partial\Omega_i$. By decomposing the whole boundary into separate faces, the discrete value of $\nabla \cdot \mathbf{v}$ on Ω_i can be computed as

$$(\nabla \cdot \mathbf{v})_i = \frac{1}{|\Omega_i|} \sum_{f \subset \partial\Omega_i} \mathbf{v}_{fc} \cdot \mathbf{n}_f \, |f|, \tag{8}$$

where subscript fc refers to the center position of the face f. Using the relation $\nabla\varphi = \nabla \cdot (\varphi I)$, the discrete cell-center gradient $(\nabla\varphi)_i$ is estimated similarly.

If the collocated cell-center velocity $\mathbf{u}^*$ is used to compute $\nabla \cdot \mathbf{u}^*$ in Eq. (5), some spurious grid-to-grid oscillations may appear due to odd–even decoupling between velocity and pressure. This is one of the main drawbacks for non-staggered grids. As a consequence, this decoupling causes large variations of pressure that are even more critical for quadtree grids, and this problem can lead to numerical instabilities at the level jumps. As shown by Ferziger and Peric [20], traditional collocated methods cannot guarantee the pressure smoothness and the mass conservation simultaneously. One way to overcome this problem has been proposed by Patankar [21] and consists in a fully staggered arrangement of the variables $(\mathbf{u}, p)$. For this kind of methods, the prediction step (3) and the Poisson equation (5) are solved at different locations, which leads to different spatial discretizations. In this sense, staggered arrangements become more challenging for Cartesian methods.

In order to stabilize the method, the collocated approach introduced by Rhie and Chow [22] for steady flows, and Zang et al. [23] for unsteady flows, is considered. A face-center velocity called U^*_{fc} is defined in Ω (see [13]) as

$$\tilde{\mathbf{u}} = \mathbf{u}^* + \frac{\Delta t}{\rho}(\nabla p^n)_{cc} \tag{9a}$$

$$\widetilde{U_{fc}} = \mathcal{F}(\tilde{\mathbf{u}}) \tag{9b}$$

$$U^*_{fc} = \widetilde{U_{fc}} - \frac{\Delta t}{\rho}(\nabla p^n)_{fc}, \tag{9c}$$