East Asian Journal on Applied Mathematics
doi: 10.4208/eajam.2022-335.300123

Vol. **13**, No. 3, pp. 671-716
August 2023

An Implicit Staggered Hybrid Finite Volume/Finite Element Solver for the Incompressible Navier-Stokes Equations

Alessia Lucca[1], Saray Busto[2] and Michael Dumbser[3,*]

[1]*Department of Mathematics, University of Trento, Via Sommarive 14, 38123 Trento, Italy.*
[2]*Departamento de Matemática Aplicada I, Universidade de Vigo, Campus As Lagoas Marcosende s/n, 36310 Vigo, Spain.*
[3]*Laboratory of Applied Mathematics, DICAM, University of Trento, Via Mesiano 77, 38123 Trento, Italy.*

Received 30 November 2022; Accepted (in revised version) 30 January 2023.

Dedicated to Professor Tao Tang on the occasion of his 60th birthday.

Abstract. We present a novel fully implicit hybrid finite volume/finite element method for incompressible flows. Following previous works on semi-implicit hybrid FV/FE schemes, the incompressible Navier-Stokes equations are split into a pressure and a transport-diffusion subsystem. The first of them can be seen as a Poisson type problem and is thus solved efficiently using classical continuous Lagrange finite elements. On the other hand, finite volume methods are employed to solve the convective subsystem, in combination with Crouzeix-Raviart finite elements for the discretization of the viscous stress tensor. For some applications, the related CFL condition, even if depending only in the bulk velocity, may yield a severe time restriction in case explicit schemes are used. To overcome this issue an implicit approach is proposed. The system obtained from the implicit discretization of the transport-diffusion operator is solved using an inexact Newton-Krylov method, based either on the BiCStab or the GMRES algorithm. To improve the convergence properties of the linear solver a symmetric Gauss-Seidel (SGS) preconditioner is employed, together with a simple but efficient approach for the reordering of the grid elements that is compatible with MPI parallelization. Besides, considering the Ducros flux for the nonlinear convective terms we can prove that the discrete advection scheme is kinetic energy stable. The methodology is carefully assessed through a set of classical benchmarks for fluid mechanics. A last test shows the potential applicability of the method in the context of blood flow simulation in realistic vessel geometries.

AMS subject classifications: 65M08, 65M60, 35Q30, 76D05

Key words: Hybrid finite volume/finite element method, finite volume scheme, continuous finite element method, incompressible Navier-Stokes equations for blood flow applications, staggered implicit schemes.

*Corresponding author. *Email addresses:* `alessia.lucca@unitn.it` (A. Lucca), `saray.busto@uvigo.es` (S. Busto), `michael.dumbser@unitn.it` (M. Dumbser)

1. Introduction

The development of numerical methods for the simulation of incompressible flows is a wide field of research since they allow the solution of many industrial, environmental and biological problems. One of those applications is the simulation of blood flow in the human cardiovascular system. Nowadays the study and treatment of diverse pathologies may require the use of invasive techniques, which might be a risk for the life of the patient. A common example of this situation is the presence of stenosis in a vessel. An abnormal narrowing in an artery may lead to many syndromes, lowering the life quality of the patient and even causing its death. To analyse in detail the impact of the stenosis without using invasive techniques an alternative may be the use of simulation tools, see e.g. [51, 55, 69, 105]. The physical geometry can be obtained via medical images which allow the definition of a computational domain to be used in numerical simulations [99]. Therefore, having a methodology able to efficiently solve this kind of flows would constitute an important step forward in personalized medicine that might help the medics in their decisions they need to take concerning the optimal treatment of each patient. For a non-exhaustive overview of some computational methods that have already been successfully applied to the simulation of the human cardiovascular system, the reader is referred to [37, 52–54, 56, 75, 78, 79, 87–89, 97, 98] and references therein.

From the numerical point of view, some of the most widespread methodologies to simulate incompressible flows fall in the framework of pressure based semi-implicit solvers, see e.g. [7, 29, 35, 58, 60, 83, 84]. They rely on the computation of the pressure by deriving a Poisson-type equation from the mass and momentum conservation equations while an approximation of velocity is obtained from a transport-diffusion subsystem that is then updated with the contribution of the new pressure. Let us remark that performing an adequate splitting of the equations allows the decoupling of the bulk flow velocity from and the fast sound waves [110]. Then an unconditionally stable implicit algorithm can be used to solve the elliptic, linear and symmetric positive definite pressure subsystem, while the non-symmetric and nonlinear transport-diffusion subsystem, if solved explicitly, will be characterized by a CFL number that depends only on the bulk velocity. Within this framework, many families of methods have been developed depending on the approach selected to solve those subsystems including continuous finite elements (FE) [11,20,61,104,119], discontinuous Galerkin schemes (DG) [45,50,100,102], or finite volume methods (FV) [60,90,108], among others. Moreover, extensions of pressure-based methods showing the potential of this approach to solve also all Mach number flows have been proposed in the last decades see, for instance [1, 18, 19, 40, 44, 70, 82, 103] and references therein.

Despite the wide variety of methods available inside each one of the aforementioned families, their combination within one algorithm is less common. In the framework of hybrid FV/FE schemes, a new family of methods combining FV and FE methods has been built for different mathematical models, including Newtonian and non Newtonian incompressible flows [14, 22, 24], weakly compressible flows [12, 23], all Mach number flows [25, 92] and the shallow water equations [21]. The main idea behind this methodology is to employ a semi-implicit scheme on staggered unstructured meshes, where the pressure subsystem