Available online at www.sciencedirect.com
SCIENCE DIRECT®
Journal of Computational Physics 208 (2005) 154–174
JOURNAL OF COMPUTATIONAL PHYSICS
www.elsevier.com/locate/jcp

High order Godunov mixed methods on tetrahedral meshes for density driven flow simulations in porous media

Annamaria Mazzia, Mario Putti *

Department of Mathematical Methods and Models for Scientific Applications, University of Padua, via Belzoni 7, 35131 Padova, Italy

Received 7 October 2004; received in revised form 25 January 2005; accepted 26 January 2005
Available online 14 April 2005

Abstract

Two-dimensional Godunov mixed methods have been shown to be effective for the numerical solution of density-dependent flow and transport problems in groundwater even when concentration gradients are high and the process is dominated by density effects. This class of discretization approaches solves the flow equation by means of the mixed finite element method, thus guaranteeing mass conserving velocity fields, and discretizes the transport equation by mixed finite element and finite volumes techniques combined together via appropriate time splitting. In this paper, we extend this approach to three dimensions employing tetrahedral meshes and introduce a spatially variable time stepping procedure that improves computational efficiency while preserving accuracy by adapting the time step size according to the local Courant–Friedrichs–Lewy (CFL) constraint. Careful attention is devoted to the choice of a truly three-dimensional limiter for the advection equation in the time-splitting technique, so that to preserve second order accuracy in space (in the sense that linear functions are exactly interpolated). The three-dimensional Elder problem and the salt-pool problem, recently introduced as a new benchmark for testing three-dimensional density models, provide assessments with respect to accuracy and reliability of this numerical approach.

Keywords: Coupled flow and transport equations; High order finite volume; Mixed hybrid finite element; Time splitting; Tetrahedral mesh

1. Introduction

The study of coupled flow and transport problems in porous media has received growing interest in the last few years within the specialized literature. In particular, the recent development of efficient numerical

* Corresponding author. Tel.: +390498275919; fax: +390498275995.
E-mail address: putti@dmsa.unipd.it (M. Putti).

0021-9991/$ - see front matter
doi:10.1016/j.jcp.2005.01.029

techniques and of fast and easily available computational tools has promoted a number of studies concerning the modeling aspects of the phenomenon [1–4]. The occurrence of local recirculation patterns due to the density differences that appear in the presence of dissolved salts in the groundwater renders the coupled flow and transport equations nonlinear and contributes to the difficulties encountered in the numerical simulations of this type of problems.

Recently developed well-controlled lab experiments [5–7] have contributed to the understanding of the physical phenomenon and have prompted the use of these solutions as benchmark test cases for numerical simulators. A recent review on these aspects [8] and a few significant applications [9,10] have shown that the correct accuracy and reliability of the numerical solutions may not be rapidly attainable. In particular, in the case of three-dimensional simulations, the numerical difficulties may easily become overwhelming. For this reason accurate and stable algorithms need to be devised.

When using standard finite element schemes, a common approach in the hydrological community, a number of drawbacks and inaccuracies arise, in particular in the calculation of velocities when unstructured grids are employed [11]. On the other hands, block centered finite differences or finite volumes can be shown to be equivalent to the mixed finite element approach on regular meshes [12]. In this paper, we describe the development of a three-dimensional numerical scheme based on the mixed hybrid finite element (MHFE) method for the discretization of the flow equation and a combination of MHFE with high resolution finite volumes (HRFV) via a time-splitting technique [13,14] for the discretization of the transport equation. This procedure has been shown to be an effective tool for the solution of the coupled flow and transport problem in two dimensions [15,9,16]. It has been applied to Elder's problem, showing its accuracy and reliability without suffering from numerical oscillations but still introducing only a minimum amount of numerical diffusion. The two-dimensional version of this approach was used in [9] to solve the salt lake problem, introduced by [17] as a test case for density-dependent groundwater flow and solute transport. Moreover, it has been applied for high-concentration brine transport [16] following the nonlinear dispersion law proposed by [18,19]. The reasons for the success of this approach are twofold. First the MHFE scheme applied to the flow equation yields a discrete velocity field with normal components that are continuous across interelement boundaries. This property guarantees that no mass balance errors due to numerical inaccuracies in the flow discretization are introduced in the solution of the transport equation. The second reason is related to the HRFV method used in the approximation of the convective fluxes. This technique is capable of capturing sharp fronts and to accurately follow their dynamics, a characteristic of fundamental importance for density driven flows.

The full transport equation is solved by means of a time-splitting approach that combines the MHFE discretizing the dispersion fluxes and HRFV for the convective fluxes. Since integration in time is explicit for HRFV, thus requiring time step size restrictions according to the CFL constraint, and implicit for MHFE, different time steps are allowed for advection and dispersion. It has been observed that some simulation runs require very long computing times, due to the extremely small advective time steps resulting from the presence of large velocities combined with small cell sizes in only small parts of the domain. This phenomenon suggests the use of space variable time step sizes for the time-splitting technique. A similar procedure was described and analyzed in [20] on two-dimensional rectangular meshes. In this paper, we describe how such a technique can be effectively implemented in a three-dimensional solver to achieve high computational performance.

When extending the time-splitting technique to three dimensions, particular attention has to be devoted to the choice of a truly three-dimensional limiter for the advection equation. Interpolation on uniform or rectangular meshes does not pose serious problems in the reconstruction phase. On the other hand, when working on unstructured tetrahedral meshes, it is difficult to maintain high accuracy for all cell configurations [21]. For this reason we perform a numerical comparison of the behavior of several reconstruction-limiter combinations using a simple linear advection equation. Verification of the achievement of superlinear global convergence (close to second order) is studied by solving simple multiple dimensional