Journal of Computational and Applied Mathematics 267 (2014) 117–130

Contents lists available at ScienceDirect

Journal of Computational and Applied Mathematics

journal homepage: www.elsevier.com/locate/cam

Solving global problem by considering multitude of local problems: Application to fluid flow in anisotropic porous media using the multipoint flux approximation

Amgad Salama [a,*], Shuyu Sun [a], Mary F. Wheeler [b]

[a] Computational Transport Phenomena Laboratory (CTPL), Division of Physical Sciences and Engineering (PSE), King Abdullah University of Science and Technology (KAUST), Thuwal, 23955-6900, Saudi Arabia

[b] Center for Subsurface Modeling, Institute for Computational Engineering and Sciences, The University of Texas at Austin, 201 E 24th Street, Austin, TX 78712, USA

HIGHLIGHTS

- A new methodology is introduced for solving the governing equations of flow and transport.
- Global problem is divided into a set of local problems which are easy to solve.
- The global system of equations are assembled automatically in the solver routine.
- This technique uses the equations as suggested by the physics without extra manipulations.
- The result is simple, easy to run, update, and maintain algorithms.

ARTICLE INFO

Article history:
Received 30 April 2013
Received in revised form 8 September 2013

Keywords:
Multi point flux approximation
Cell-centered finite differences
Experimenting pressure field technique
Anisotropic porous media

ABSTRACT

In this work we apply the experimenting pressure field approach to the numerical solution of the single phase flow problem in anisotropic porous media using the multipoint flux approximation. We apply this method to the problem of flow in saturated anisotropic porous media. In anisotropic media the component flux representation requires, generally multiple pressure values in neighboring cells (e.g., six pressure values of the neighboring cells is required in two-dimensional rectangular meshes). This apparently results in the need for a nine points stencil for the discretized pressure equation (27 points stencil in three-dimensional rectangular mesh). The coefficients associated with the discretized pressure equation are complex and require longer expressions which make their implementation prone to errors. In the experimenting pressure field technique, the matrix of coefficients is generated automatically within the solver. A set of predefined pressure fields is operated on the domain through which the velocity field is obtained. Apparently such velocity fields do not satisfy the mass conservation equations entailed by the source/sink term and boundary conditions from which the residual is calculated. In this method the experimenting pressure fields are designed such that the residual reduces to the coefficients of the pressure equation matrix.

* Corresponding author. Tel.: +966 2 8080423.
E-mail addresses: asas.tx@gmail.com, amgad.salama@kaust.edu.sa (A. Salama).

http://dx.doi.org/10.1016/j.cam.2014.01.016
0377-0427/

1. Introduction

The continuum hypothesis provides the framework in which transport phenomena in porous media may be described using a set of partial differential equations. In this framework, macroscopic variables are described as continuous functions of space and time, Salama and Van Geel [1,2]. Such governing laws are usually solved numerically using different techniques. This includes the finite differences method, the finite elements method, the boundary elements method, etc. Several difficulties usually encounter modeling of transport phenomena in porous media. This may be related to the wide spectrum of length scales associated with porous media applications (i.e., from small scale applications in confined spaces to large hydrogeological scales). Furthermore, porous media characterization is usually prone to uncertainties of parameters and properties that often require repeated simulation to optimize model prediction. In addition, phenomena occurring in porous media often incorporate complex mechanisms associated with the quantification of the possible interactions mechanisms (e.g., chemical, rheological, etc.) between the flowing fluid and the solid particles. Moreover, several porous media applications involve the transport of multi-phase systems with the associated complex interfacial phenomena. All these and more apparently require robust and efficient algorithms that could accomplish reliable computations in reasonable time. A robust numerical scheme is that which is simple in coding, faster in execution, can handle complex geometries, can work with anisotropic and heterogeneous properties of the medium, follows the physics of the phenomena under investigation, etc. While the first few aspects mentioned above are in some sense pertinent to the numerical techniques themselves, the last one is very much related to the algorithm methodology. Unfortunately, none of the above mentioned numerical techniques shows superiority in all aspects of interest to porous media applications. That is although finite differences are probably the simplest of all the numerical schemes approximating partial differential equations; they confront several difficulties, particularly, when the geometry of the simulation domain is complex. In this case it becomes exceedingly difficult for the mesh to conform to the boundary of the domain. On the other hand, when the properties of medium show directional dependence (i.e., anisotropy), traditional finite differences (i.e., the so called two-point flux approximation) cannot handle but diagonal tensor quantities. Finite elements, on the other hand, have the advantages that it works well for complex geometries and could also consider full tensor quantities. However, it is more involved and generally takes more time for coding and execution. The question is, if none of the above mentioned numerical techniques can, alone, collect all the favorable requirements mentioned above, is it possible to combine some of these methods and techniques so that a more reliable approach may be used? In other words, is it possible to broaden the scope of finite differences method (which is the simplest of all numerical techniques) to incorporate tensor quantities (for example) and if possible in what framework? To highlight this question it has been recognized that some finite element techniques can reduce, through some quadrature rules, to a finite difference scheme. As an example, mixed finite elements have shown to reduce to cell-centered finite differences through some quadrature rules. However, this was possible only for diagonal permeability tensors. For full permeability tensors, it has lately been recognized that within the framework of multi-point flux, mixed finite element method, and using certain quadrature rules, one may be able to find a finite difference representation of the governing equations which can handle full permeability tensor, Russell and Wheeler [3], Wang and Mathew [4], Arbogast et al. [5], Wheeler and Yotov [6,7], Wheeler et al. [8,9], Osman et al. [10], to list but a few. However, as will be discussed later, these expressions are quite long and would generally generate longer algebraic formulas upon discretization. For example, each flux component at the edge of an interior cell would require six pressure values in 2D rectangular mesh (or 18 pressure values in 3D rectangular meshes) from the surrounding cells and the divergence of the velocity field at any interior cell would require, respectively, 9 and 27 pressure data from the surrounding cells for the 2D and 3D rectangular meshes. This apparently would lead to algorithms which are difficult to maintain, develop, update, etc. In other words, one needs to use such long stencils in order to construct global matrix to obtain the pressure field over the domain of interest. This is apparently cumbersome and would generally require that one tracks all the contributing terms over each cell in order to construct the global matrix. If it is possible to handle each cell alone (i.e., locally) and automatically constructing the global matrix based on the solution over each cell, the solution of such problem will be significantly facilitated. Sun et al. [11], introduced a new methodology to model transport phenomena in porous media which follows the logical sequence of the governing equations. In this technique, no to minimal manipulation of the governing set of equations to obtain reduced equations are required. In this technique, the matrix of coefficients which is required to obtain the pressure field is constructed automatically within the solver routine. This allows one to discretize the governing equations as suggested by the physics which results in simpler and shorter algebraic expressions. This technique will show its superiority when applied to the multipoint flux approximation. That is while in isotropic porous media the matrix of coefficients which is used to obtain the unknown pressure field is generally five-diagonal matrix (in 2D rectangular meshes), in multipoint flux approximation of flow in anisotropic media, the matrix is generally nine-diagonal. Therefore constructing such matrix automatically is a noticeable success as will be apparent later. In addition, in this work, the boundary conditions are considered with the system of discretized equations. Although this methodology results in relatively larger system of equations, these extra equations only constitute a small portion of the system of equations particularly when the mesh is dense.

2. The governing equations

Consider a given porous medium domain as shown in Fig. 1. The governing flow equations for this system may be given as:

$$\mathbf{u} = -\boldsymbol{K}\nabla p \quad \text{in } \Omega \subset R^d,\ d = 2, 3 \tag{1}$$