Journal of Petroleum Science and Engineering 96–97 (2012) 58–67

Contents lists available at SciVerse ScienceDirect

Journal of Petroleum Science and Engineering

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

Application of space–time conservation element and solution element method in streamline simulation

Majid Siavashi [a,*,1], Peyman Pourafshary [b], Mehrdad Raisee [a]

[a] School of Mechanical Engineering, University of Tehran, Tehran, Iran
[b] Institute of Petroleum Engineering, University of Tehran, Tehran, Iran

ARTICLE INFO

Article history:
Received 15 January 2011
Accepted 6 August 2012
Available online 19 August 2012

Keywords:
streamline simulation
reservoir simulation
Conservation Element and Solution Element (CE/SE)
time of flight (TOF)
Finite volume method (FVM)
Total variation diminishing (TVD)

ABSTRACT

Streamline simulation has been developed as an alternative to conventional Eulerian methods for the simulation of multiphase fluid flow in oil and gas reservoirs. In streamline simulation the saturation equations are solved over streamlines as a function of time-of-flight (TOF).We present a parallel implementation of this method. Furthermore, to increase the accuracy and speed of simulation, a new method called space–time conservation element and solution element (CE/SE) is implemented to solve the saturation equations along streamlines. CE/SE has many non-traditional features, including a unified treatment of space and time and stable numerical behavior with no need to introduce total variation diminishing schemes. As flux is a time–space property, in CE/SE, both time and space are discretized and treated on the same footing. In CE/SE, parameters and their derivatives are considered as independent variables and are computed simultaneously at each time-step that leads to local and global flux conservation. In addition by introduction of solution element and conservation element, cell fluxes can be calculated without extrapolation. To show the strength of method, the Buckley–Leverett equation is solved using CE/SE and compared to the finite volume method (FVM). Then, the method is employed in a streamline simulator to simulate water injection in a heterogeneous oil reservoir. CE/SE can capture the correct solution better than the FVM especially near sharp fronts. Though CE/SE is a second order method, its accuracy is higher than the third order Leonard's scheme and its order of convergence is higher than current methods in the literature. In addition, the simulation time of CE/SE is about 10 percent lower than the other second and third order methods we test.

1. Introduction

The simulation of fluid flow in oil and gas reservoirs using streamline techniques is now widely employed for reservoir modeling studies. Excellent speed and efficiency, computation of new valuable engineering data, flow visualization (Thiele, 2005), complex and full field modeling ability on fine-scale geological models (Batycky et al., 1997) and easy parallel processing potential (Batycky et al., 2010) are some of its advantages over conventional grid-based techniques.

Streamline simulation has many applications in petroleum engineering. This method is used for history matching (Cheng et al., 2005', 2007), upscaling problems, flood surveillance, flood management (Thiele et al., 2010b), fractured reservoir simulation (Al-Huthali and Datta-Gupta, 2004; Di Donato et al., 2003; Huang et al., 2004), hot water flooding (Zhu et al., 2010) and polymer flooding (Thiele et al., 2010a). For detailed information about streamline simulation theory and its application, the reader is referred to (Batycky et al., 1997; Datta-Gupta and King, 2007).

In a conventional finite-difference or finite-volume method, the governing equations are solved in 3D on a fixed (Eulerian) grid. In contrast, in streamline simulation, by introducing the time-of-flight concept, the saturation equation is decoupled into a set of one-dimensional equations along streamlines and multiple one-dimensional equations are solved along streamlines. As a result, in streamline simulation a dual-grid method is applied to solve the fluid flow equations: an Eulerian grid to solve the pressure equation and trace the streamlines which is called the "pressure grid", and the 1D "streamline grid" to solve the saturation equations over them. This methodology is successful because the time step size to update the pressure field can often be much larger than the time increment for the solution of the saturation equation along streamlines. Hence, by solving the pressure equation once, the saturation equation can be solved along streamlines for a long time. Furthermore, it is easy to extend the method to 3D using a semi-analytical method to trace streamlines and compute the time-of-flight (Pollock, 1988).

* Correspondence to: School of mechanical engineering, University of Tehran, Faculty of Engineering (Campus #2), North Kargar St., Jala-Al-Ahmad Expressway, Tehran, Iran.
E-mail addresses: msiavashi@ut.ac.ir, m.siavashi@imperial.ac.uk (M. Siavashi).
[1] Current Temporary Address: Department of Earth Science and Engineering, Imperial College London, SW7 2AZ.

0920-4105/$ - see front matter © 2012 Elsevier B.V. All rights reserved.
http://dx.doi.org/10.1016/j.petrol.2012.08.005

The transport equation is a hyperbolic equation and often its solution features discontinuities. This means that the method of solution must be capable of capturing shocks and high gradient regions. First order finite difference or finite volume methods are easy to implement, but it is necessary to use a fine grid in order to capture sharp gradients. In addition, explicit methods are limited by the *CFL*[2] condition and application of the fine grid leads to small time steps and increases the solution time. High order methods can capture sharp gradients without needing fine grids. In these methods, local flux/slope is estimated based on neighboring node values. As the gradients are high around shocks, the flux/slope is predicted erroneously in these regions leading to spurious oscillations (Datta-Gupta and King, 2007). Such spurious oscillations can be eliminated by flux/slope limiting methods[3] (Van Leer, 1997).

Chang (1995), Chang and To (1991), Chang et al. (1994, 1999, 2000) introduced a new method for solving conservation laws over Eulerian grids that is called "Space–Time Conservation Element and Solution Element" (CE/SE). CE/SE is designed to avoid the limitations and weaknesses of the traditional methods. Nevertheless, its foundation is sufficiently mathematically simple that one can build from it a coherent, robust, efficient, and accurate numerical framework. Two basic features that set the new method apart from the established methods are at the core of its development (Chang et al., 1999). The first idea is that, in order to capture physics more efficiently and realistically, the modeling focus should be placed on the original integral form of the physical conservation laws, rather than the differential form. The latter form follows from the integral form under the additional assumption that the physical solution is smooth, an assumption that is difficult to realize numerically in a region of rapid change, such as shocks. The second idea is that, with proper modeling of the integral and differential forms themselves, the resulting numerical solution should automatically be consistent with the properties derived from the integral and differential forms, e.g., the jump conditions across a shock and the properties of characteristics. Therefore, a simple and more robust method can be developed by avoiding the explicit use of the above derived properties.

CE/SE has some new features and advantages over common numerical methods. In traditional methods, the mesh is generated only for the space domain and averaged values in time are used to estimate flux. In contrast, as flux is fundamentally a time–space property, CE/SE uses a time–space grid and both time and space are treated in a unified manner. By this treatment, flux can be estimated as a function of time and space that is consistent with the physics of the problem. In the CE/SE method, all independent variables and their derivatives are considered as unknowns in each grid point and are computed simultaneously at time–space nodes. This feature is very important, because both individual parameters and their derivatives are computed such that flux can be conserved locally and globally all over the time–space domain. Consequently, flux/slope limiting methods are not needed anymore in order to prevent spurious oscillations caused by erroneous estimated flux. CE/SE introduces two types of elements: solution element (SE) and conservation element (CE). Integration is performed over CE boundaries and parameters are defined on SE. SEs construct boundaries of CEs, thus solution variables are available on boundaries of CEs for integration, in contrast to FVM that uses approximate extrapolated/interpolated face values for integration. These features enable CE/SE to capture shocks without using Riemann solvers and to predict discontinuities accurately.

Overall CE/SE is beneficial for the solution of conservation equations, especially in problems with discontinuous profiles. Numerous highly accurate solutions of conservation laws such as advection and diffusion (Wang et al., 1995), Navier–Stokes (Chang, 1995; Zhang et al., 2002), magneto-hydrodynamics (Qamar and Mudasser, 2010; Qamar and Warnecke, 2006; Zhang et al., 2006), heat transfer (Yang et al., 2009) and fluid flow in porous media (Yang et al., 2009) have been obtained using the CE/SE method.

Despite these excellent advantages, the CE/SE has not been developed like other well-known numerical methods. It is due to one major disadvantage, the complexity of the topology of the 3D time–space grids. Hence, despite the superior capabilities and features of the CE/SE method, it is easier to implement other numerical methods to solve the governing equations. In this work we avoid these problems, since transport equations are solved only in 1D.

Streamline simulation is used to simulate two-phase (water–oil) flow in a heterogeneous reservoir. The Visual Studio Parallel Pattern Library (PPL) is applied to parallelize streamline simulation simply and efficiently. By coupling CE/SE with streamlines, the major disadvantage of the CE/SE method can be eliminated and its advantages can be used to improve the accuracy and efficiency of the simulation. Accordingly, in order to increase the accuracy and speed of reservoir simulation, CE/SE is proposed to solve the saturation equation over streamlines. In following, first the accuracy and performance of the method are analyzed for the solution of the hyperbolic transport equations over a specific streamline. Then the effect of the method implementation on accuracy and performance of multi-dimensional streamline simulation is investigated.

Streamline simulation is explained briefly in the proceeding section. In Section 3, solution methods of saturation equation are discussed. In Part 3.1, the finite volume method is explained, in Part 3.2 CE/SE is described and in Part 3.3 appropriate boundary conditions for the CE/SE method are developed. Section 4 focuses on the results of the CE/SE method and presents a comprehensive comparison of CE/SE and FVMs.

2. Streamline simulation

Streamline simulators are based on the implicit pressure, explicit saturation (IMPES) approach to solve the governing conservation equations. The general conservation equation is represented by the pressure and saturation equations. This technique relies on six key principles: 1—solution of pressure equation and calculation of total velocity 2—tracing of streamlines in a velocity field (Pollock, 1988) 3—writing the mass conservation equations in terms of time-of-flight (TOF) 4—numerical solution of conservation equations along streamlines 5—periodic updating of the streamlines and 6—operator splitting to account for transverse flux effects (Bratvedt et al., 1996). More details of this method can be found in related references (Batycky et al., 1997; Datta-Gupta and King, 2007). The method is explained here in brief.

Initially, the pressure equation (Eq. (1)) is solved under the steady state condition.

$$\nabla \times k(\lambda_t \nabla P + \lambda_g \nabla D) = 0 \tag{1}$$

where k is permeability, D is depth, λ_t is total mobility and λ_g is total gravity mobility.

The total velocity is then computed using the Darcy's law (Eq. (2)) and thus streamlines are traced, by implementing the

[2] Courant Friedrich Levy.
[3] Sometimes called TVD methods.