Journal of Contaminant Hydrology 180 (2015) 34–55

Contents lists available at ScienceDirect

Journal of Contaminant Hydrology

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

Evaluation of a coupled model for numerical simulation of a multiphase flow system in a porous medium and a surface fluid

Yoshihiko Hibi [a,*], Akira Tomigashi [b]

[a] Faculty of Science and Technology, Meijo University, 1-501 Shiogamaguchi, Tenpaku-ku, Nagoya, Aichi 468-8502, Japan
[b] Technical Development Department, Engineering Management and Coordination Division, Yachiyo Engineering Co., LTD, 2-18-1112 Nishiochiai, Shinjuku-ku, Tokyo 161-8575, Japan

ARTICLE INFO

Article history:
Received 24 December 2014
Received in revised form 21 July 2015
Accepted 25 July 2015
Available online 29 July 2015

Keywords:
Multiphase flow
Darcy equation
Porous medium
Surface fluid
Coupling
Numerical simulation

ABSTRACT

Numerical simulations that couple flow in a surface fluid with that in a porous medium are useful for examining problems of pollution that involve interactions among atmosphere, water, and groundwater, including saltwater intrusion along coasts. Coupled numerical simulations of such problems must consider both vertical flow between the surface fluid and the porous medium and complicated boundary conditions at their interface. In this study, a numerical simulation method coupling Navier–Stokes equations for surface fluid flow and Darcy equations for flow in a porous medium was developed. Then, the basic ability of the coupled model to reproduce (1) the drawdown of a surface fluid observed in square-pillar experiments, using pillars filled with only fluid or with fluid and a porous medium and (2) the migration of saltwater (salt concentration 0.5%) in the porous medium using the pillar filled with fluid and a porous medium was evaluated. Simulations that assumed slippery walls reproduced well the results with drawdowns of 10–30 cm when the pillars were filled with packed sand, gas, and water. Moreover, in the simulation of saltwater infiltration by the method developed in this study, velocity was precisely reproduced because the experimental salt concentration in the porous medium after saltwater infiltration was similar to that obtained in the simulation. Furthermore, conditions across the boundary between the porous medium and the surface fluid were satisfied in these numerical simulations of square-pillar experiments in which vertical flow predominated. Similarly, the velocity obtained by the simulation for a system coupling flow in surface fluid with that in a porous medium when horizontal flow predominated satisfied the conditions across the boundary. Finally, it was confirmed that the present simulation method was able to simulate a practical-scale surface fluid and porous medium system. All of these numerical simulations, however, required a great deal of computational effort, because time was incremented in 0.05- to 0.10-s steps. Hereafter, the present simulation method needs to be improved so that the simulations can be conducted with less computational effort.

1. Introduction

Numerical simulations that couple flows in a porous medium and a surface fluid with a free water surface are useful for examining problems of pollution that involve interactions among atmospheres, surface water, and groundwater (Brock et al., 1982), including saltwater intrusion along coasts. The latter occurs along the boundary between groundwater and seawater, where groundwater flows seaward concurrently with the inverse (landward) flow of seawater (Belanger and Mikutel, 1985; Boufadel, 2000; Groen et al., 2000; Hibi et al., 2010; Li et al., 2008). In particular, when gravity currents in surface waters and groundwater near a coast lead to saltwater intrusion, the vertical flow of the surface water and the

* Corresponding author.
E-mail address: hibiy@meijo-u.ac.jp (Y. Hibi).

http://dx.doi.org/10.1016/j.jconhyd.2015.07.005
0169-7722/

complicated boundary conditions between the surface fluid and the porous medium should be modeled by a numerical simulation that couples surface fluid flow with flow in a porous medium. In a coupled solution, the mass and momentum of the fluid on the boundary must be conserved (Discacciati et al., 2002; Hill and Carr, 2013; Hill and Straughan, 2008, 2009).

Because Navier–Stokes equations are able to simulate rigorously the complex flow of a surface fluid, these equations are more suitable than other equations for application to multiphase flows such as two-phase gas–water flows. By simulating multiphase flows with Navier–Stokes equations, it is possible to precisely solve the migration of the free water surface and the predominantly vertical flow of the surface water with improved numerical simulation methods for modeling multiphase flows that have recently been developed (Chang et al., 1996; Hirt and Nichols, 1981; Sussman et al., 1994; Yabe et al., 2001).

However, because computing Navier–Stokes equations is difficult and costly, few numerical simulations coupling multiphase flow with Navier–Stokes equations and equations for flow in porous media have been done (Geng et al., 2014; Geng and Boufadel, 2015). Therefore, we developed a numerical simulation method in which Navier–Stokes equations for surface fluid flow are coupled with Darcy equations for flow in a saturated porous medium and explored the basic ability of the coupled numerical simulation by comparing the simulation results with the results of simple experiments, with the aim of assessing its applicability to more practical problems in the future.

The Navier–Stokes equations for surface fluid flow and the equations for flow in a saturated porous medium should be simultaneously and rigorously solved to satisfy the continuity of velocity normal to the boundary between the porous medium and the surface water and normal and tangential stress conditions at this boundary. Furthermore, it is necessary to confirm that the results of the developed numerical simulation satisfy these boundary conditions.

Accordingly, in this study we confirmed that the developed coupled simulation method was able to reproduce the migration of the free water surface in square-pillar experiments performed with an experimental device containing a surface fluid and a porous medium in which the free water surface was drawn down unidirectionally. Moreover, it was verified that the velocity in the porous medium could be rigorously reproduced by using the developed simulation method by comparing the migration of saltwater in a porous medium obtained by a square-pillar experiment with that of a simulation in which saltwater infiltrated into a saturated porous medium under a layer of surface water. It was also verified that the results obtained by the developed simulation method satisfied the conditions at the boundary between the porous medium and the surface water both in simulations of square-pillar experiments in which velocity was predominantly normal to this boundary and in simulations in which velocity was predominantly parallel to this boundary. Finally, we confirmed that a practical-scale surface fluid and porous medium system could be simulated by the developed simulation method. Therefore, we confirmed in this study that the developed simulation method could simulate flow and pressure in a system coupling flow in a surface fluid with that in a porous medium in which vertical flow was predominant in the surface water and with complicated boundary conditions between the surface water and the porous medium.

2. Theory of the numerical simulation method

Navier–Stokes equations were used to describe the surface fluid flow:

$$\nabla \cdot \mathbf{V} = 0, \tag{1}$$

$$\partial \mathbf{V}/\partial t + (\mathbf{V} \cdot \nabla)\mathbf{V} = -(1/\rho)\nabla p + (\mu/\rho)\nabla^2 \mathbf{V} - \mathbf{g}, \tag{2}$$

where t is elapsed time; $\mathbf{V}$ and p are the velocity and the pressure of the fluid, respectively; $\mathbf{g}$ is the gravity vector when the vertical coordinate z is positive upward; and μ and ρ are the viscosity and density of the fluid, respectively.

Eqs. (1) and (2) are often solved by using a segregated solver (e.g., a marker and cell method (Harlow and Welch, 1965), a simplified marker and cell method (Amsden and Harlow, 1970), a project method (Chorin, 1968), or a semi-Lagrange method (Smolarkiewicz and Pudykiewicz, 1992)). This study employed a semi-Lagrange method with cubic interpolated propagation (CIP) (Yabe et al., 1991), because discretization of Eqs. (1) and (2) with this method is simpler and more efficient than with the other methods. Furthermore, the semi-Lagrange method with CIP produces accurate results when applied to multiphase flow.

By applying the semi-Lagrange method with CIP and the velocity vector $\mathbf{V}^*$ at the position vector $\mathbf{R}_i - \mathbf{V}^t\Delta t$ in the velocity field at time t, where $\mathbf{R}_i$ is the position vector at point i, $\mathbf{V}^t$ is velocity at time t, and Δt is the time increment, Eqs. (1) and (2) can be transformed by the backward difference method as follows:

$$(\Delta t/\rho)\nabla \cdot \left(\nabla p^{t+\Delta t} + \rho \mathbf{g}\right) = \nabla \cdot \mathbf{V}^* \tag{3}$$

$$\mathbf{V}^{t+\Delta t} - (\mu/\rho)\Delta t \nabla^2 \mathbf{V}^{t+\Delta t} = \mathbf{V}^* - (\Delta t/\rho)\ \left(\nabla p^{t+\Delta t} + \rho \mathbf{g}\right). \tag{4}$$

First, $\mathbf{V}^*$ was obtained by interpolation at the position vector $\mathbf{R}_i - \mathbf{V}^t\Delta t$ in the velocity field at time t, and $p^{t+\Delta t}$ at time $t + \Delta t$ was derived by discretizing Eq. (3) using the Galerkin finite element method (FEM). The interpolation was performed by CIP with cubic curves imposing values and gradients at nodes. Second, $\mathbf{V}^{t+\Delta t}$ was calculated from $\mathbf{V}^*$ and $P^{t+\Delta t}$ at the nodes by using the algebraic expressions derived by discretizing Eq. (4) with the Galerkin FEM. Finally, $\mathbf{V}^*$ and $P^{t+\Delta t}$ at the nodes were explicitly calculated for every time increment Δt.

As there is a free water surface inside some elements of the FEM, μ and ρ should be expressed in terms of the void fraction of water S_w, which is the ratio of the water volume to the total fluid volume:

$$\rho = S_w\rho_w + (1 - S_w)\rho_g \tag{5}$$

$$\mu = S_w\mu_w + (1 - S_w)\mu_g, \tag{6}$$