Chemical Engineering Science 65 (2010) 3363–3371

Contents lists available at ScienceDirect

Chemical Engineering Science

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

SPH simulation of oil displacement in cavity-fracture structures

Guangzheng Zhou [a,b], Zhihai Chen [c], Wei Ge [a,*], Jinghai Li [a]

[a] State Key Laboratory of Multiphase Complex Systems, Institute of Process Engineering, Chinese Academy of Sciences, Beijing 100080, China
[b] Graduate University of Chinese Academy of Sciences, Beijing 100049, China
[c] Petroleum Exploration and Production Institute of Sinopec, Beijing 100083, China

ARTICLE INFO

Article history:
Received 5 June 2009
Received in revised form
11 February 2010
Accepted 16 February 2010
Available online 20 February 2010

Keywords:
Oil recovery
Simulation
Smoothed particle hydrodynamics
Immiscible fluids
Surface tension
Wetting behavior
Displacement
Gravity drainage

ABSTRACT

This paper presents a study of the immiscible displacement of oil by water in cavity-fracture structures using smoothed particle hydrodynamics (SPH). The surface tension and wetting behavior are incorporated into the equation of motion, and pseudo periodic boundary conditions are applied. As for ''middle-fractured'' structure, it is found that the ultimate oil recovery is almost determined by the height of fracture regardless of its orientation, and the result compares well with corresponding experiments. Besides, the water-wet wall is favorable to higher oil recovery. A systematic exploration is carried out on ''upper-fractured'' structure for the feasibility of gravity drainage, where a critical width of fracture is found, beyond which the oil in the cavity can be driven out. The possible development of SPH in this background for large-scale simulation is prospected finally.

1. Introduction

Multiphase immiscible flows are frequently encountered both in nature and industries, such as oil recovery, groundwater contamination, and packed bed chemical reactors. With increasing demand for oil products in the background of limited oil reserves, the role of secondary oil recovery (water flooding) is getting more and more important. In general, parametric investigation on the production conditions such as fracture orientation, wettability, and inflow velocity is helpful to increase the oil recovery, and for this purpose, expensive and time-consuming physical experiments can be effectively reduced by numerical modeling. However, as for naturally fractured karst reservoir, oil is mostly stored in huge cavities, and flow mainly takes place in fractures and cavities (Lin, 2002; Chen et al., 2005). Apart from the large dimensional ratio of cavity to fracture, the fractures, cavities, and matrixes are intensively heterogeneous and anisotropic (Castaing et al., 2002; Lunati and Jenny, 2006), thus posing great difficulty for the prediction of oil recovery with traditional numerical technology based on the flow theory of porous media (Sahimi, 1995). Since a better understanding about the fundamental physics which govern flow and transport processes is of high practical significance, a direct numerical simulation of immiscible flow in typical cavity-fracture structures is carried out in this study accordingly.

Macro-scale particle methods such as smoothed particle hydrodynamics (SPH) (Gingold and Monaghan, 1977; Lucy, 1977), macro-scale pseudo-particle modeling (Ge and Li, 2001, 2003), and some other variants (Koshizuka et al., 1995; Koshizuka and Oka, 1996; Ma et al., 2006), are a collection of fully meshfree Lagrangian techniques of computational fluid dynamics, in which the numerical solution is achieved through the movement and interactions of numerous particles. Though originally proposed in the context of astrophysical applications, SPH is quite suitable for flows involving geometrically complex boundaries and dynamical interfaces, where coalescence and breakup of surface can be readily handled without complicated procedures of mesh generation and management in mesh-based methods. In addition, SPH can be readily extended to involve extra physical and chemical effects. Through decades of rapid development, SPH has been applied to a wide range of areas, such as free-surface incompressible flows (Monaghan, 1994), low Reynolds number incompressible flows (Morris et al., 1997), high energy impacts and explosions (Liu et al., 2003), and even large strain solid mechanics (Libersky et al., 1993; Benz and Asphaug, 1995). As to the application of SPH to fluid flow in porous medium, it is limited to single phase yet (Zhu et al., 1999; Jiang et al., 2007). For a comprehensive description of the SPH method, please refer to (Liu and Liu, 2003; Monaghan, 2005).

Surface tension plays an important role in the simulation of immiscible fluids, and it has been implemented in SPH in different ways. Continuum surface force model (Morris, 2000; Liu and Liu, 2005; Hu and Adams, 2006) relies on an explicit estimation of

* Corresponding author. Tel.: +86 10 82616050; fax: +86 10 62558065.
E-mail address: wge@home.ipe.ac.cn (W. Ge).

0009-2509/$ - see front matter © 2010 Elsevier Ltd. All rights reserved.
doi:10.1016/j.ces.2010.02.029

interface curvature, and some other methods mimic the microscopic physics giving rise to surface tension (Nugent and Posch, 2000; Tartakovsky and Meakin, 2005a). Particularly, the model proposed by Zhou et al. (2008) has proven to be simple, computationally efficient, and reasonably accurate. With this model, the displacement of oil by injected water is simulated using SPH for the typical "middle-fractured" and "upper-fractured" structures. The effects of some important factors on flow behaviors and ultimate oil recovery are investigated.

2. Smoothed particle hydrodynamics for immiscible fluids

2.1. SPH fundamentals

In SPH, the continuous fluid is represented by a large set of particles. Each particle i located at $\mathbf{r}_i$ is associated with physical parameters such as mass m_i, density ρ_i, velocity $\mathbf{v}_i$, and pressure P_i. The value of any generic function $f(\mathbf{r})$ can be approximated by a number of neighboring particles (which may also be regarded as interpolation points) using a weighting function W

$$f(\mathbf{r})=\sum_j \frac{m_j}{\rho_j} f(\mathbf{r}_j)W(\mathbf{r}-\mathbf{r}_j,h), \tag{1}$$

and its gradient $\nabla f(\mathbf{r})$ becomes

$$\nabla f(\mathbf{r})=\sum_j \frac{m_j}{\rho_j} f(\mathbf{r}_j)\nabla W(\mathbf{r}-\mathbf{r}_j,h), \tag{2}$$

where h is the smoothing length that determines the support of the weighting function.

The weighting function is assumed to be an even function of finite range, which is normalized to unity when integrated over space. As illustrated in Fig. 1, only the solid particles within the circular influence domain (radius=κh) of the smoothing function for particle i actually contribute to the summation process. A variety of forms, including spline functions of different order, have been used for the weighting functions. Due to its high precision and stability (Morris et al., 1997), the quintic spline of the two-dimensional version:

$$W(s)=\frac{7}{478\pi}\begin{cases}(3-s)^5-6(2-s)^5+15(1-s)^5, & 0\le s<1;\\ (3-s)^5-6(2-s)^5, & 1\le s<2;\\ (3-s)^5, & 2\le s<3;\\ 0, & s\ge 3,\end{cases} \tag{3}$$

where $s=|\mathbf{r}-\mathbf{r}'|/h$, is also adopted in our work.

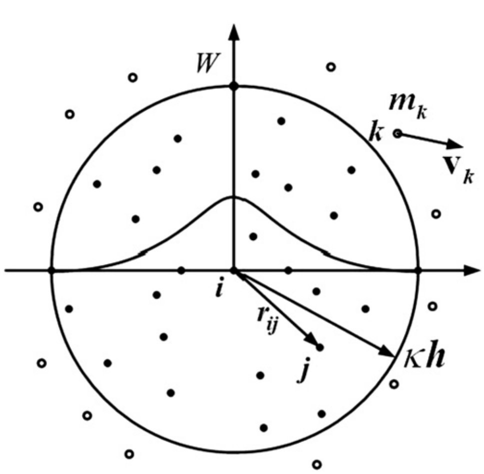

Fig. 1. Sphere of influence for particle i in a two-dimensional (2D) space.

The evaluation of the density at a particle for a given distribution of particle masses is readily obtained by applying Eq. (1) to the density to give

$$\rho_i=\sum_j m_j W(r_{ij},h), \tag{4}$$

where r_{ij} denotes the distance between particles i and j. It should be mentioned that another method based on continuity equation is also frequently used to calculate density (Monaghan, 1994), which is particularly advantageous for free surface flow. Since Eq. (4) conserves the total mass exactly, it is used for the simulations described herein.

Although most implementations of SPH employ an artificial viscosity that was first introduced to permit the modeling of strong shocks (Monaghan, 2005), the physical viscosity of real fluid has been realized for low Reynolds number incompressible flow (Takeda et al., 1994; Morris et al., 1997). Employing a commonly used form of the pressure gradient and the expression for viscous diffusion derived by Morris et al. (1997) which ensures the conservation of linear momentum exactly, the resulting SPH formulation of the Navier–Stokes equation for nearly incompressible flow is

$$\frac{d\mathbf{v}_i}{dt}=-\sum_j m_j\left(\frac{P_i}{\rho_i^2}+\frac{P_j}{\rho_j^2}\right)\nabla_i W_{ij}+\sum_j\frac{m_j(\mu_i+\mu_j)(\mathbf{v}_i-\mathbf{v}_j)}{\rho_i\rho_j}\left(\frac{1}{r_{ij}}\frac{\partial W_{ij}}{\partial r_i}\right)+\mathbf{g}, \tag{5}$$

where μ_i and μ_j are the dynamic viscosity of the fluid with respect to particles i and j, respectively.

However, it is found that the straightforward extension of Eq. (5) to multiphase flows creates an artificial surface tension due to the jump in density at the interface between the two fluids (Hoover, 1998; Colagrossi and Landrini, 2003; Tartakovsky and Meakin, 2005b). To eliminate this negative effect, the particle number density n_i as proposed by Tartakovsky and Meakin (2005b), is weighted instead to evaluate the local density, that is,

$$n_i=\sum_j W(r_{ij},h). \tag{6}$$

Actually, the notion of particle number density is the same to the idea of specific volume suggested by Hu and Adams (2006).

Adding force $\mathbf{F}_i$ for surface tension on the fluid–fluid interface or wetting force on the wall to Eq. (5), the new version of momentum equation based on the particle number density is

$$\frac{d\mathbf{v}_i}{dt}=-\frac{1}{m_i}\sum_j\left(\frac{P_i}{n_i^2}+\frac{P_j}{n_j^2}\right)\nabla_i W_{ij}+\frac{1}{m_i}\sum_j\frac{(\mu_i+\mu_j)(\mathbf{v}_i-\mathbf{v}_j)}{n_i n_j}\left(\frac{1}{r_{ij}}\frac{\partial W_{ij}}{\partial r_i}\right)+\mathbf{g}+\frac{\mathbf{F}_i}{m_i}, \tag{7}$$

where different treatments of surface tension are reflected in the detailed forms taken by $\mathbf{F}_i$.

SPH cannot model a truly incompressible fluid. Instead, an artificial compressibility technique is used to model the incompressible flow as a slightly compressible flow via an artificial equation of state as follows:

$$P=c^2 n=Kn, \tag{8}$$

where c is the numerical sound speed, and K is usually called stiffness parameter. The numerical sound speed is a key factor that deserves careful consideration (Morris et al., 1997). Last but not least, it is well known that the explicit integration is conditionally stable, and some criteria of selecting time step is followed according to (Morris et al., 1997).