Computers and Geotechnics 82 (2017) 237–248

Contents lists available at ScienceDirect

Computers and Geotechnics

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

Research Paper

An auto-adaptive moving mesh method for the numerical simulation of piping erosion

Yue Liang [a,c], Tian-Chyi Jim Yeh [b,c,*], Junjie Wang [a], Mingwei Liu [a], Yuanyuan Zha [d], Yonghong Hao [b]

[a] National Engineering Research Center for Inland Waterway Regulation, Chongqing Jiaotong University, Chongqing, China
[b] Key Laboratory for Water Environment and Resources, Tianjin Normal University, Tianjin, China
[c] Department of Hydrology and Water Resources, University of Arizona, Tucson, AZ, USA
[d] State Key Laboratory of Water Resources and Hydropower Engineering Science, Wuhan University, Wuhan, China

ARTICLE INFO

Article history:
Received 13 July 2016
Received in revised form 26 September 2016
Accepted 13 October 2016

Keywords:
Piping
Flownet mesh
Streamline tracing
Finite volume method
Auto-adaptivity

ABSTRACT

A numerical approach for simulating piping erosion is proposed based on an auto-adaptive moving mesh. This approach adapts a flownet mesh (FM) to discretize the domain. It reduces the simulation of two-dimensional transport of moving particles to one-dimensional problem by taking advantages of flow tube concept. Furthermore, it employs a stepwise procedure to decouple the solid-liquid interaction equations for modeling piping erosion. Specifically, the FM is updated and evolved automatically in accordance to simulated erosion since the flownet is refined in high-velocity regions, where the piping erosion progresses. Therefore, the FM is auto-adaptive to the piping erosion paths.

1. Introduction

Backward piping erosion is one of the most common causes of failures in levees, earth dams, cofferdams, and other hydraulic earth structures [1–4]. The piping erosion is initiated by the seepage force exerted on fine particles in cohesiveless (or weakly cohesive) soils. The transport and migration of the fine particles out of the soil matrix result in development of seepage pipes, leading to failures of structures if the piping continues and expands. The impacts of the piping erosion have been recognized for more than 100 years. Numerous researchers have conducted multiscale experiments to investigate piping behaviors, including studies of the erosion criteria and the erosion rate [5–7], and the controlling factors, such as the stress state [8–12] and the soil composition [13,14]. Nevertheless, our understanding of the piping mechanisms is still limited due to their complexity and difficulty in detecting piping in the field [3].

Besides experimental studies, mathematical models have been widely used to describe the piping erosion process. Early models concentrated on erosion criteria, including critical hydraulic gradient, critical hydraulic head difference, and erosion length [15–17]. These models are generally solved with analytical methods based on simplified assumptions. Recently, mathematical models, which consider the solid-liquid coupling and other controlling factors, have been proposed. These more realistic and complex models are generally solved by using the finite element method (FEM) [18–21], the finite volume method (FVM) [22], or the finite difference method (FDM) [1]. In addition, models with new numerical methods are introduced to consider the solid-liquid coupling process, such as the two-flow model [23], the seepage-tube flow coupling model [24], the DEM-based method [2,25–27], and the energy-based method [28–30]. These numerical models generally are computationally inefficient for simulating the complex coupling processes of piping erosion.

Flow and unknown small-scale heterogeneity in the soil triggers the development of piping, and then changes in hydraulic and mechanical characteristics of the soil owing to particle loss follow [3]. These changes take place along unknown preferential flow paths [18], dictated by heterogeneity, and the other portion of the soil remains intact. For this reason, the computational mesh, utilized in the numerical simulation of the piping progression, is often set to be fine enough to capture all possible developments of the piping paths. As a consequence, those numerical solutions using uniform fine meshes are computationally expensive since

* Corresponding author at: Department of Hydrology and Water Resources, The University of Arizona, 1133 E. James E. Rogers Way, 122 Harshbarger Bldg 11, Tucson, AZ 85721, USA.
E-mail address: yeh@hwr.arizona.edu (T.-C.J Yeh).

http://dx.doi.org/10.1016/j.compgeo.2016.10.011
0266-352X/

the regions without erosion are much larger than those with erosion [22], especially in multi-dimensional problems, in which the number of mesh elements required can be prohibitively large.

The adaptive moving mesh (AMM) [31] appears to be a possible solution to the aforementioned problems for simulating the piping erosion problem. The AMM places fine meshes in the active regions where the state variables exhibit strong spatial and temporal variabilities, and uses coarse meshes in the dormant regions. This algorithm has been widely used in solving fluid dynamics, hydraulics, combustion, and heat transfer problems [32–34]. The AMM however requires a prescribed mesh density function to facilitate the mesh refinement.

In this paper, we propose a flownet-based strategy to generate auto-adaptive meshes for numerical simulations of piping erosion. Flownet is a graphical representation of a two-dimensional seepage field. It has been an important tool for analyzing two-dimensional irrotational flow problems, and frequently used in civil engineering studies, hydrogeology or soil mechanics as a first-cut analysis for problems of flow under hydraulic structures like dams or sheet pile walls. The construction of the flownet is based on the equipotential lines and the streamlines, which discretizes the seepage field into a series of discrete elements. A mesh based on the flownet is called the flownet mesh (FM). It is notable that the edges of the elements in the flownet mesh are neighboring streamlines and equipotential lines. As a result, the FM provides significant conveniences for the numerical simulation of the piping erosion. The first advantage is the auto-adaptivity of the FM. The elements are automatically refined at the high seepage velocity regions where the gradient is steep and streamlines converge. These high seepage velocity regions are exactly the potential areas for the development of piping erosion. Besides, FM approach reduces mathematical complexity for modeling the movement of eroded particles. Specifically, the flownet defines stream tubes, and within each stream tube, fluids macroscopically move along the stream tube and do not interfere with those in adjacent stream tubes. For this reason, the convective mass transfer along a stream tube can be regarded as a one-dimensional process. The multi-dimensional movement of the eroding particles can therefore be reduced to a one-dimensional problem.

2. FM-based numerical algorithm

Generally, a moving mesh method for numerical simulation has three major components: the strategy to move the mesh, the method to discretize the physical PDE, and the approach to solve the coupled system of physical and mesh equations [31]. In this section, the three components are separately discussed in detail.

2.1. FM generating

Under steady-state and irrotational flow conditions in aquifers with homogeneous and isotropic hydraulic conductivity, flownet can be constructed by solving two harmonic potential and stream functions analytically or numerically [35]. While analytical method is not available for flow in heterogeneous and anisotropic aquifers, numerous of numerical methods have been developed to derive streamlines for simulation of seepage field in heterogeneous and anisotropic petroleum and groundwater reservoirs [36–38]. In this study, a popular method for generating streamlines in an irregular mesh is adopted. This method evolves from the algorithm developed by Prévost, which is a semi-analytical approach. It uses the algorithm proposed by Pollock [39] to create streamlines in regular meshes firstly, and then employs the bilinear transformation to estimate the locations of the streamlines in the corresponding irregular meshes [36].

In a regular element, the Pollock's method assumes that each directional velocity component varies linearly within a unit square element. As shown in Fig. 1(a), the four edges of a unit element is marked as A, B, C, and D, respectively. The edge velocities u_A, u_B, v_C, and v_D are perpendicular to the edges and do not change with time. The x-velocity component u and the y-velocity component v in the element are simplified as a lowest order Raviart-Thomas velocity field, and are expressed as

$$u(x,y) = u(x) = (u_B - u_A)x + u_A \tag{1a}$$

$$v(x,y) = v(y) = (v_D - v_C)y + v_C \tag{1b}$$

Now, suppose a particle exits an element from a known point $N(1, y_0)$ on the edge B. All the other three edges then could be the possible entry for the particle. The travel time for the particle moving from any possible entry from the three edges to the exit can be calculated by integrating Eqs. (1a) and (1b). Assuming the travel times from edges A, C, and D are T_A, T_C, and T_D, respectively, the minimal value of these three times is the time-of-flight (TOF) that the particle travels from the entry to the exit. That is,

$$T_o = \min(T_A, T_C, T_D) \tag{2}$$

where T_o is the time-of-flight. The edge corresponding to the TOF is then considered as the entry area. The analytical solution of the travelling trajectory, namely the streamline, of the particle in the element is

$$x(t) = \begin{cases} \frac{(G_x x_e + u_A)e^{G_x t} - u_A}{G_x}, & G_x \neq 0, \quad 0 \leqslant t \leqslant T_0 \\ u_A t + x_e, & G_x = 0, \quad 0 \leqslant t \leqslant T_0 \end{cases} \tag{3a}$$

$$y(t) = \begin{cases} \frac{(G_y y_e + v_C)e^{G_y t} - v_C}{G_y}, & G_y \neq 0, \quad 0 \leqslant t \leqslant T_0 \\ v_C t + y_e, & G_y = 0, \quad 0 \leqslant t \leqslant T_0 \end{cases} \tag{3b}$$

where x_e, and y_e are the coordinates of the entry points. u_A and v_C are the velocities defined in Eq. (1). Afterward, the streamline tracing can be carried out for the neighboring element that shares the particle entry edge. The entry point (x_e, y_e) determined above is then the known exit point for the tracing in the adjacent element. This backward tracing from the exit boundary to the entry boundary of the solution domain yields a complete streamline. Other streamlines can be generated similarly with this approach by changing the initial tracing location on the exit boundary.

Now, we consider the case where the element in irregular in shape. The geometry of a typical irregular element is shown in Fig. 1(b). Four corners of the element are N_1, N_2, N_3, and N_4, respectively. The flux over each edge is f_A, f_B, f_C, and f_D, respectively. With the bilinear transformation, the relationships between the coordinates in a physical space [Fig. 1(b)] and the coordinates in a reference space [Fig. 1(c)] is given by

$$\begin{pmatrix} x \\ y \end{pmatrix} = (1-\xi)(1-\eta)\begin{pmatrix} x_1 \\ y_1 \end{pmatrix} + \xi(1-\eta)\begin{pmatrix} x_2 \\ y_2 \end{pmatrix} + \xi\eta\begin{pmatrix} x_3 \\ y_3 \end{pmatrix} + (1-\xi)\eta\begin{pmatrix} x_4 \\ y_4 \end{pmatrix} \tag{4}$$

where ξ, and η are the local reference space coordinates varying between [0,1]; (x_1,y_1), (x_2,y_2), (x_3,y_3), and (x_4,y_4), are the coordinates of the points N_1, N_2, N_3, and N_4, respectively. The transformation of the fluxes is given by

$$u_A = \frac{f_A}{J}; \quad u_B = \frac{f_B}{J}; \quad v_C = \frac{f_C}{J}; \quad v_D = \frac{f_D}{J} \tag{5}$$

where J is the determinant of the Jacobian of the transformation, which is

$$J = \frac{1}{2}(x_1y_2 - y_1x_2 + x_2y_3 - y_2x_3 + y_1x_4 - x_1y_4 + x_3y_4 - y_3x_4) \tag{6}$$