Study on flow characteristics of rough undulating single fracture based on CFD

Shu Liang1,2*

1Faculty of Geosciences and Environmental Engineering, Southwest Jiaotong University, Chengdu, China
2China Southwest Geotechnical Investigation & Design Institute Co., Ltd, Chengdu, China
Corresponding author’s e-mail: ls_bps@163.com

Abstract. The cubic law is the basic law to describe the seepage in the cracks of parallel plates. However, it cannot be directly used to describe rough undulating fractures. By using the method of computational fluid dynamics, the law of water flow in rough undulating fractures with different roughness, fracture width, hydraulic gradient and fracture length is studied. The cubic law is modified and the mathematical formula of equivalent fracture width is established.

1. Introduction
A single fracture is the basic unit of a fracture network. The study of flow movement in a single fracture is the starting point and cornerstone of the study of seepage in a three-dimensional fracture network. It is well known that the flow in a single fracture is a cubic law based on the parallel smooth equal width model (Formula 1). However, natural cracks are not always parallel, smooth and equal in width, but rough and undulating, which do not satisfy the basic assumption of cubic law.[1-2]

\[ Q = \frac{g a^3}{12v} J \]  

(1)

Where \( Q \) is the flow rate per unit time, and \( g \) is the acceleration of gravity, and \( a \) is the gap width, and \( J \) is the kinematic viscosity coefficient of water, and \( I \) is the hydraulic gradient.

Based on the theory of fluid dynamics, this paper uses FLUENT software to calculate the flow law of single fracture under different roughness, different hydraulic gradient, different fracture width and different fracture length, so as to modify the cubic law and establish the calculation formula of equivalent fracture width, which provides parameters for the seepage calculation of three-dimensional fracture network rock slope model.[3]

2. Basic Theory of Computational Fluid Dynamics and Establishment of Single Fracture Flow Model
2.1 Basic Theory of Computational Fluid Dynamics
Computational fluid dynamics (CFD) is an interdisciplinary subject based on classical hydrodynamics and numerical methods. Its governing equations are flow conservation equations, including mass conservation equation, energy conservation equation and momentum conservation equation. Thermal field is not involved in this paper, and energy conservation equation is not considered. The mass conservation equation and momentum conservation equation can be seen in Formula 2–Formula 3.
\[
\frac{\partial p}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i) = S_m
\]  
(2)

\[
\frac{\partial}{\partial t} (\rho u_i) + \frac{\partial}{\partial x_j} (\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} (\rho \sigma_{ij}) + \rho g_i + F_i
\]  
(3)

\(\rho\) is fluid density, and \(u_i\) is fluid velocity tensor, and \(x_i\) is velocity direction, and \(S_m\) is source and sink term, and \(P\) is static pressure, and \(\sigma_{ij}\) is stress tensor, and \(c_j\) is stress direction.

(1) Mathematical model

Common fluid mathematical models include laminar model, standard \(k\)-\(\varepsilon\) model, RNG \(k\)-\(\varepsilon\) model and Spalart-Allmaras model.

Laminar model is the simplest flow mathematical model, and the flow pattern is laminar flow. The standard \(k\)-\(\varepsilon\) model obtains the solutions of \(K\) and \(\varepsilon\) by solving the turbulent kinetic energy \((k)\) equation and the turbulent dissipation rate \((\varepsilon)\) equation, then calculates the turbulent viscosity with the values of \(K\) and \(\varepsilon\), and finally obtains the solution of Reynolds stress through Boussinesq hypothesis. The RNG \(k\)-\(\varepsilon\) model is similar in form to the standard \(k\)-\(\varepsilon\) model, but is better than the standard \(k\)-\(\varepsilon\) model in computational function. Spalart-Allmaras model is one of the most successful models in one equation model. It was first used to calculate fluid with low Reynolds number, especially in the case of boundary layer viscous effect which needs to be calculated accurately.

(2) Treatment of boundary layer

Wall boundary can be used to simulate crack wall. In the wall region, especially in the viscous bottom layer, the fluid flow is almost laminar, and the turbulent stress hardly works, so it can not be solved directly by the turbulent model. The low Re \(k\)-\(\varepsilon\) model can be used to solve the viscous regions (viscous bottom layer and transition layer).

2.2 Establishment and test of model

Parallel fracture model is established to verify the correctness of the calculation results. The laminar flow constitutive model is adopted, the fracture length is 480 mm, the width is 1 mm, and the fracture wall is straight and smooth. The fluid is water flow, neglecting gravity, with a density of 0.998 g/cm\(^3\) and a dynamic viscous coefficient of 1.003\times10^{-3} Pa\cdot s. The mesh length of the model is 1 mm. In order to increase the accuracy of the boundary layer calculation, the mesh size near the fracture wall is refined. The mesh width of the first layer in contact with the fracture wall is 1 \times10^{-3} mm, and the width of the fracture increases gradually at a rate of 1.2 times to the center of the fracture. The maximum width is controlled below 1 \times10^{-4} mm. Fig. 1 is the schematic model.

The left side of the model is the inlet pressure boundary, the pressure is 10 Pa; the right side is the pressure outlet boundary, the pressure is 0 Pa, and the design hydraulic gradient is 0.002. The upper and lower edges of the model are fixed-wall non-slip boundaries.

The calculated flow rate of the model is consistent with that of the cubic law, which is 1.662 \times10^{-6} m\(^3\). The velocity distribution on the section is basically consistent with that derived from the cubic law.

![Figure 1 Velocity profile of cross section](image-url)

3. The establishment of seepage model of rough undulating single fracture

The basic model is 480 mm in length, 24 bulges, 4 mm in height, 10 mm in width, 100 Pa in inlet, 0 in outlet and 0.02 in hydraulic gradient. The meshing is shown in Fig. 2.
The calculated results of fissure water are shown in Fig. 3-5. The water pressure decreases gradually with the linearity of the process, and the flow velocity is larger in the narrow crack and smaller in the concave. Influenced by rough fracture wall, due to the bulge of fracture surface hinders fluid flow, the flow of fluid near the fracture wall is adjusted sharply, and the velocity and direction of flow change dramatically. In the depression of the fracture, there are eddies locally, which lead to the increase of friction resistance in the fluid, generate additional local energy consumption, and reduce the flow rate and flow rate of the fracture. At the corner of the fracture, the local streamline is tangent to the fracture wall, resulting in energy loss.

4. Influences of different factors on seepage in rough undulating single fissure

(1) Number of protrusions
   The flow rate of single fissure decreases with the increase of the number of bumps, and the curve is negative exponential (Fig.6 (a)). This is because the more the number of bumps, the more vortices are generated in the fissure, and the greater the loss of flow rate in the process of fissure flow. Linear fitting of the relationship between the flow rate of a single fracture and the number of uplifts can still retain a good fitting accuracy. In order to form a simple result, the linear fitting relationship is used to express the relationship between the number of single fractures and the number of uplifts.

(2) Height of protrusion
   The flow rate of a single fracture decreases linearly with the increase of the uplift height (Fig.6 (b)). The deeper the concave depth of the undulating fracture is, the larger the vortices are generated in the fracture, and the greater the loss of flow rate in the process of flow of the fracture water is. Linear fitting of the relationship between single fracture flow rate and uplift height can still retain a good fitting accuracy.

(3) Average gap width
   The flow rate in a single fracture is nearly linear with the average width of the fracture, rather than a cubic relationship as in the cubic law (Fig.6 (c)). This is because the flow in rough and undulating fissures has eddy current, which consumes a lot of energy, which is not consistent with the assumption of cubic law. However, it is not known whether the linear relationship used to fit the relationship between flow rate and average gap width is an approximate mathematical expression, and whether it has practical physical significance.

(4) Hydraulic gradient
The flow rate of single fissure increases with the increase of hydraulic gradient, because according to Darcy’s law, the larger the hydraulic gradient, the greater the velocity, the larger the flow rate (Fig.6 (d)). The relationship curve between flow rate and hydraulic gradient in a single fracture tends to be flat with the increase of hydraulic gradient. This is because the larger the hydraulic gradient is, the larger the Rayleigh number is, the more obvious the turbulence and eddy current are, the more energy is consumed, and the slower the flow rate increases with the increase of hydraulic gradient.

(5) Fracture length

The influence of fracture length on fracture flow rate is negligible (Fig.6 (e)). This is because the rough undulating fissure water is in the state of eddy current and turbulence, and it is difficult to form a stable transition section with parallel smooth and equal width fissures.

According to the principle of equal flow rate, the equivalent gap width under different conditions can be calculated according to the cubic formula (Formula 4), the equivalent gap width can satisfy the value of the cubic formula.

\[ a_e = \sqrt{\frac{12q_v}{gJ}} \]  \hspace{1cm} (4)

The mathematical expression of equivalent fracture width is established by fitting multiple linear equation:

\[ a_e=0.28a-0.12JRC-6.4J+2.3 \]  \hspace{1cm} (5)
Fitting degree $R^2 = 0.8919$, the maximum fitting error is 0.96 mm. When the average width and roughness of cracks are large, the fitting accuracy is poor, but it can basically meet the engineering accuracy requirements.

5. Conclusion
The computational fluid dynamics (CFD) method is used to study the flow patterns in rough undulating fractures with different roughness, fracture width, hydraulic gradient and fracture length. The main conclusions are as follows:

1) Roughness, fracture width and hydraulic gradient are the main factors affecting the seepage of rough undulating fractures. The larger the roughness is, the larger the whirlpool generated in the fracture is, the more energy loss of the fracture flow is, and the smaller the flow of the single fracture flow is. The larger the average fissure width is, the larger the flow rate of single fissure is. The larger the hydraulic gradient is, the greater the fracture velocity is and the larger the single fracture flow rate.

2) The mathematical expression of equivalent fracture width of rough undulating single fracture seepage is established by multiple linear regression method: $a_e=0.28a-0.12JRC-6.4\ln(J)+2.3$, which provides basic calculation parameters for seepage calculation of three-dimensional fracture network.

Acknowledgments
This paper has been sponsored by Project supported by the research and development project of CSCEC(CSCEC-2015-Z-37). Here authors express their appreciation for this support.

References
[1] Cornet F H , Li L , Hulin J P , et al. The hydromechanical behaviour of a fracture: An in situ experimental case study[J]. International Journal of Rock Mechanics and Mining Sciences, 2003, 40(7):1257-1270.

[2] Lee H S , Park Y J , Cho T F , et al. Influence of asperity degradation on the mechanical behavior of rough joints under cyclic shear loading[J]. International Journal of Rock Mechanics and Mining Sciences, 2001, 38(7):967-980.

[3] Ling D , Duan J , Tang H . Physical-constraints-preserving Lagrangian finite volume schemes for one- and two-dimensional special relativistic hydrodynamics[J]. 2019.