ISSN: 2472-3703 DOI: 10.6911/WSRJ.202309_9(9).0006

Analysis of Fracture Width in Hydraulic Fracturing Based on ANSYS

Yu Fu, Fuwei Li

Petroleum Engineering School, Southwest Petroleum University, Chengdu, China

Abstract

The width of hydraulic fractures determines their conductivity, and when the width is too small, proppant-laden fluid cannot flow through, resulting in a sharp increase in construction pressure beyond the operational limit, leading to the failure of hydraulic fracturing operations. Additionally, fracture width also affects the post-fracturing production of the well. In this study, an ANSYS finite element analysis software was utilized, along with the finite element method of fracture mechanics, to establish a finite element model for predicting fracture width. The elastic-plastic fracture mechanics finite element method was employed to determine the width of the fracture and the stress intensity factor at the crack tip, while also capturing the stress variation surrounding the fracture. By simulating different wellhead pressures, the study reveals the relationship between the stress intensity factor at the crack tip, fracture width, and wellhead pressure. It is observed that as the wellhead pressure increases, the stress intensity factor and fracture width also increase. Furthermore, by simulating different elastic moduli of the formation, the study demonstrates that the hydraulic fracturing width gradually decreases with an increase in elastic modulus. Through analyzing the factors influencing hydraulic fracture width, this research provides valuable insights for the design and construction of hydraulic fracturing operations.

Keywords

Hydraulic fracturing, ANSYS finite element, Elastic modulus, Fracture width, Stress intensity.

1. INTRODUCTION

Hydraulic fracturing is currently the most effective method for extracting hydrocarbons from low-permeability reservoirs, and accurately understanding and evaluating the effectiveness of fracturing operations plays a crucial role in improving the ultimate oil and gas recovery. Currently, there are various methods for simulating hydraulic fracturing, including finite element methods, discrete element methods, boundary element methods, etc. Among them, the finite element method is considered an effective approach for simulating hydraulic fracturing, providing both computational accuracy and relatively ideal calculation speed. Numerical simulation studies using the finite element method to analyze the dynamic process of hydraulic fracturing have important guiding implications for on-site fracturing operations. Construction parameters for hydraulic fracturing, such as the volume of fracturing fluid, number and orientation of perforations, all have significant impacts on the width of reservoir fractures. Therefore, accurately simulating the influence of construction parameters on reservoirs is of great value for on-site oilfield operations. In this study, an in-depth analysis of fracture width in hydraulic fracturing is conducted to provide valuable insights for on-site fracturing operations in oilfields.

ISSN: 2472-3703 DOI: 10.6911/WSRJ.202309_9(9).0006

2. BASIC THEORY AND CRITERIA FOR FRACTURE PROPAGATION

2.1. Criteria for Fracture Initiation and Propagation

The process of hydraulic fracture propagation is generally accompanied by shear slip effects, resulting in a complex fracture pattern. ANSYS employs the cohesive zone model to study the initiation and propagation criteria for this type of fracture. The main content includes the functional response relationship between interfacial tensile stress and relative displacement and the relationship between interfacial energy during the fracture process. Regarding the determination of initial fracture, stress-strain relationships are considered, and ANSYS provides several standard approaches. In this study, the widely used secondary stress failure criterion is selected, which states that initial fracture occurs when the sum of the squares of stress ratios in three directions reaches 1, i.e.

2.2. Stress Intensity Theory at the Crack Tip

The crack tip refers to the region near the tip of the crack, which is also referred to as the crack front or apex in some literature. The stress field and displacement field at the crack tip are fundamental for studying rock fracture. Therefore, investigating the stress field and displacement field at the crack tip is crucial as it controls the fracture process in that vicinity. Assuming the crack body is linear elastic, the stress field and displacement field for different types of cracks can be obtained using elastic theory. This leads to the introduction of the concept of stress intensity factors.

In the vicinity of the crack tip, within a sufficiently small region called the KⅠ-dominant zone or control zone, for each loading mode, regardless of the geometric shape and load conditions, as long as they have the same value of KⅠ, the distribution of stress, displacement, and strain is exactly the same, and their magnitudes are directly proportional to KⅠ. Therefore, KⅠ is a measure of the intensity of the stress field at the crack tip in linear elastic materials, and it is called the stress intensity factor. It is a significant parameter in linear elastic fracture mechanics and is essential for analyzing or solving the stress field and displacement field near the crack tip in geological formations, which involves calculating the stress intensity factor.

2.3. Physical Model

2.3.1 Basic Assumptions

a) The rock is assumed to be homogeneous, linear elastic, and isotropic.

b) The fracture height is assumed to be constant.

c) The flow of fracturing fluid in the cross-sectional direction of the fracture is neglected, The fluid flows one-dimensionally along the length of the fracture.

d) The fracture undergoes two-dimensional failure in terms of crack length and width.

e) The fracture surface is assumed to be planar.

3. FINITE ELEMENT ANALYSIS

3.1. Element Properties

In previous studies, most finite element models of cracks were established using modeling methods for symmetric half cracks. In this study, the PLANE183 eight-node quadrilateral solid element with plane strain is used. The finite element model is established by creating coincident keypoints near the crack tip.

ISSN: 2472-3703 DOI: 10.6911/WSRJ.202309_9(9).0006

3.2. Mesh Generation

The element size near Key Point 1 is set to 20mm, and the element size near Key Point 6 is also set to 20mm. The remaining element sizes are controlled at 500mm. The A1 plane is meshed using the Meshing—Mesh—Areas—Free command.

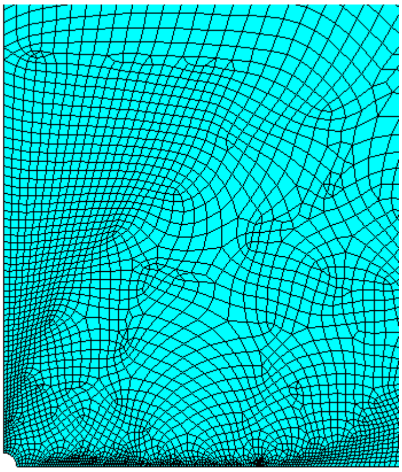

Figure 1. Grid Section

3.3. Application of Boundary Conditions

The wellbore fracturing pressure, Pj, is applied to the AB and AF edges, i.e., L1 and L6. Based on the distribution of the stress field around the wellbore, the minimum in-situ stress is applied to the L4 edge, and the maximum in-situ stress is applied to the L3 edge. According to the theory of elastic mechanics, the displacement at the L3 edge, far from the stress concentration zone, has a negligible effect. Therefore, the x and y-direction displacements at the L3 edge are constrained to zero using the Define Loads—Apply—Structural—Displacement—Symmetry B.C.—OnLine command. The symmetric displacement constraints are applied to the L2 and L5 edges using Define Loads—Apply—Structural—Displacement—Symmetry B.C.—OnLine.

3.4. Solution Settings

To visualize the distribution of Von Mises stress, the contour plot for nodal solutions can be generated by selecting Plot Results—Contour plot—Nodal solu—Stress—Von Mises stress. An example of the Von Mises stress contour plot is shown in the figure below:

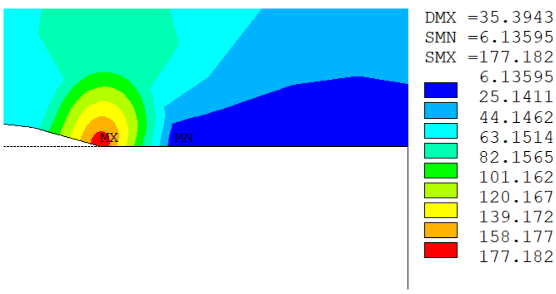

Figure 2. Stress-strain diagram

To establish a local coordinate system at the crack tip, enter the following command in the command window: LOCAL,11,0,r+length. Refer to Figure 3-9 for the visualization of the local coordinate system. Next, create a stress intensity factor path as shown in Figure. Use a symmetric model to calculate KI as KI1. Obtain the stress intensity factor KI1 by entering the

ISSN: 2472-3703 DOI: 10.6911/WSRJ.202309_9(9).0006

command *get,KI1,Kcalc,,K,1 in the command window. Check the variable data by navigating to Parameters—Scalar Parameters.

To establish a crack width path, enter the command path,Dy,2,,100 in the command window. Create paths n4 and n5. Set up Path Operations to obtain the crack width on the paths. Finally, use Map onto Path and select UY to display the crack width curve.

3.5. Simulation of Different Wellbore Pressures

Based on the established hydraulic fracturing finite element model, simulate the variations in the stress intensity factor and crack width curve for wellbore pressures of 40 MPa, 50 MPa, and 60 MPa using ANSYS software.

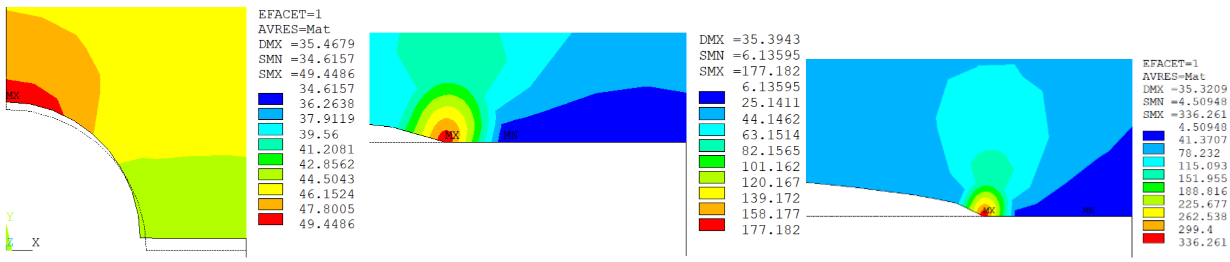

Figure 3. Stress-strain diagram under different wellbore pressures

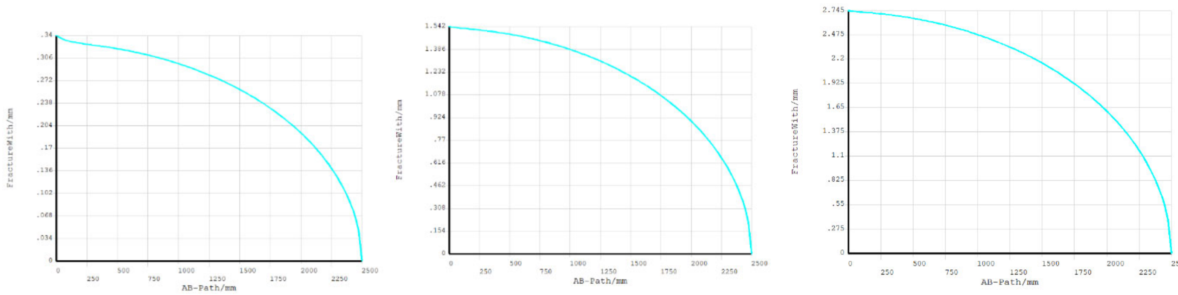

Figure 4. Relationship between crack width and path for wellbore pressures of 40 MPa, 50 MPa, and 60 Mpa

Simulation of different wellbore pressures reveals that as the wellbore pressure increases, the Von-Mises stress also increases, and the location of the maximum Mises stress gradually moves closer to the crack tip from the crack width direction. Additionally, the stress intensity factor at the crack tip increases with increasing wellbore pressure. From Figure 3.18, it can be observed that the relationship between crack width and path is nonlinear, and the crack width continuously increases with increasing wellbore pressure.

4. CONCLUSION

(1) As the wellbore pressure increases, the Von-Mises stress increases, and the location of the maximum Mises stress gradually moves closer to the crack tip from the crack width direction. The stress intensity factor at the crack tip also increases with increasing wellbore pressure. The relationship between crack width and path is nonlinear, and the crack width continuously increases with increasing wellbore pressure.

(2) Analyzing the hydraulic fracturing finite element and its influencing factors provides valuable guidance for optimizing fracturing design.

ISSN: 2472-3703 DOI: 10.6911/WSRJ.202309_9(9).0006

REFERENCES

[1] Ren Q W, Dong Y W, Yu T T.Numerical modeling of concrete hydraulic fracturing with extended finite element method[J].Science in China Series E; Technological Sciences, 2019, 52(3); 559-565.

[2] Shi F, Wang X L, Liu C, et al.A coupled extended finite element approach for modeling hydraulic fracturing in consideration of proppant[J].Journal of Natural Gas Science & Engineering, 2021, 33; 885-897.

[3] Wang X L, Shi F, Liu H, et al.Numerical simulation of hydraulic fracturing in orthotropic formation based on the extended finite element method[J].Journal of Natural Gas Science and Engineering, 2016, 33; 56-69.

[4] Shi F, Wang X, Liu C, et al.An XFEM-based method with reduction technique for modeling hydraulic fracture propagation in formations containing frictional natural fractures[J].Engineering Fracture Mechanics, 2017, 173; 64-90.

[5] Cleary M P, Kavvadas M, Lam K Y. Development of A Fully Three-Dimensional Simulator for Analysis and Design of Hydraulic Fracturing[C]//SPE/DOE Low Permeability Gas Reservoirs Symposium. Society of Petroleum Engineers, 1983.

[6] Chen Z, Bunger A P, Zhang X, et al. Cohesive zone finite element-base.