Numerical simulation of allocator annular flow field

Yi Zhang¹, Jiexiang Wang¹,*, Li Gao² and Guoyu Chu¹

¹School of Petroleum Engineering, China University of Petroleum (East China), Qingdao, China
²College of Biological and Chemical Engineering, Qingdao Technical College, Qingdao, China

*Corresponding author e-mail: upc1609@163.com

Abstract. The allocator is the key tool for the oilfield to realize the multi-layered polymer flooding. The most common allocator is the annular pressure-reducing groove structure. The selection of reasonable structural characteristics parameters has always been a difficult problem in the design of pressure reducing structure. The structural optimization design involves many parameters, and the numerical simulation method can effectively reduce the dependence on the experiment and the number of experiments. In this paper, the parametric modelling of the annular pressure-reducing groove structure is carried out, and the numerical simulation model of the power law fluid flowing in the annular channel is established. The turbulence calculation model is used to numerically calculate the flow field. The velocity, pressure, turbulent kinetic energy and strain rate contours are obtained and analysed. The numerical calculation results of the annulus flow field with four different structural characteristic parameters are compared and the optimal structural parameters are obtained, which can provide valuable reference for the design and improvement of the annular pressure-reducing groove allocator.

1. Introduction

With the continuous promotion and application of polymer flooding enhanced oil recovery technology, it is particularly important to study the multi-layered injection technology suitable for polymer flooding. The key to solving this problem is the development of allocator. Satisfying the pressure control and reducing the apparent viscosity loss of the polymer solution during the compounding process are two key technologies in the multi-layered polymer flooding [1-3]. The designed pressure is generally achieved by the annular pressure-reducing groove throttling method. It is difficult to ensure that both the high pressure drop and the low apparent viscosity loss are simultaneously achieved. How to choose the structural characteristic parameters of the annular pressure-reducing groove to achieve the balance between the resistance and the apparent viscosity loss is a difficult problem in the design of the pressure-reducing groove [4, 5].

At present, judging the performance of the pressure-reducing groove allocator depends on experimental research methods. With the development of computer simulation technology, the numerical simulation method to optimize the design can greatly reduce the time and cost [6, 7]. In this paper the influence of the characteristic parameters of the annular pressure-reducing groove on its pressure drop and apparent viscosity loss is studied. The optimal groove structure parameters are
obtained and it provides theoretical basis for optimal design of polymer flooding and annular groove allocator.

2. Numerical Simulation Calculation Method

2.1. Mathematical Model

The fluid flow should follow the law of conservation of physics. These laws mainly include the law of conservation of mass, the law of conservation of momentum, and the law of conservation of energy [9, 10]. The basic equations of computational fluid dynamics are the descriptions of these conservation laws. When the flowing conditions are constant, the flow field in the control unit is a constant and isothermal flow [11]. In the cylindrical coordinates, the corresponding governing equations mainly include continuity equation, momentum equation, energy equation, non-Newtonian fluid constitutive equation and initial and boundary conditions of the basic equation.

2.2. Meshing

For the study of the flow channel, in order to reduce the amount of numerical calculation, the actual three-dimensional problem is transformed into a two-dimensional axisymmetric problem to calculate. The unstructured hybrid mesh [12] is used because of the large variation in the flow path scale. Figure 1 is a two-dimensional meshing diagram of the flow channel. The two-dimensional grid is mostly a quadrilateral mesh and a triangular mesh of transitions. The total number of grids is between 50,000 and 80,000.

![Meshing diagram of the flow channel.](image)

2.3. Boundary and Flow Medium

The flow is turbulent; the two-dimensional simulated exit is the pressure boundary, the gauge pressure is 0; the solid wall adopts the non-slip boundary condition, and the wall function method is adopted near the wall surface. The two-dimensional model of the runner structure adopts the velocity inlet boundary condition. The inlet velocity values of different examples are 0.48m/s, 0.96m/s, 2.87m/s, 3.83m/s, 5.74m/s, and 7.18m/s, respectively (the corresponding injection flowrates are 10m³/d, 20m³/d, 60m³/d, 80m³/d, 120m³/d and 150m³/d respectively). The flow medium in the allocator is a polymer solution, which belongs to a non-Newtonian fluid.

2.4. Structural Characteristics of Allocator

The alternating peaks and troughs of the annular depressurization groove cause the velocity and pressure of the fluid in the flow channel to change periodically, and a strong disturbance occurs during the change. This periodic variation and disturbance cause a large pressure drop when the polymer solution flows through the depressurization groove [8]. The structure of the annular pressure-reducing groove is shown in Figure 1. The main structural parameters are the inner diameter $D_i$ of the outer tube, the outer diameter $D_o$ of the inner tube, the diameter $d$ of the semicircle groove, the spacing of the groove $a$ and the length $L$ of the allocator.
Figure 2. Schematic diagram of allocator.

In the field application, the length of the allocator is limited. In this paper, the profile control is used for the purpose of operation. The length of the allocator is 400mm and four different structural parameters is studied. The structural parameters are as follows:

- Scheme 1: groove spacing is 10mm and semicircle diameter is 6mm.
- Scheme 2: groove spacing is 12mm and semicircle diameter is 6mm.
- Scheme 3: groove spacing is 14mm and semicircle diameter is 8mm.
- Scheme 4: groove spacing is 16mm and semicircle diameter is 10mm.

3. Simulation Results Analysis and Structural Characteristic Parameter Optimization

Taking the characteristic parameters of scheme 2 as an example, the flow field contours are obtained by simulation at a flow rate of 60m³/d. Figure 3 is contours of static pressure in allocator. It can be seen from Figure 3 that the pressure is gradually reduced from the inlet to the outlet, and each time a slot is passed, the pressure drops accordingly.

Figure 3. Contours of static pressure.
Figure 4. Contours of velocity.

Figure 4 is a flow field velocity contour map, which reflects the velocity distribution and variation in the flow channel. Observing the velocity distribution map in flow channel, it can be found that the maximum velocity appears at the smallest portion of the flow channel section, and when the flow channel area becomes large, eddy current is generated, and the velocity gradient at the sudden change of the flow channel area is large. At the transition between the groove and the straight channel, the velocity contour suddenly becomes dense, indicating that the flow changes greatly here. The flow state and the flow velocity change, and the shearing effect on the solution is strong.

The turbulent energy reflects to the fierce state of the flow. When the turbulent energy is large, the fluid flow changes complexly and violently, resulting in a large pressure drop and a large loss of viscosity. Figure 5 shows the contour of turbulent kinetic energy in flow channel. Observing the turbulent energy distribution map, it can be found that generally, where the flow area is abrupt, a large turbulent kinetic energy is generated, and the pressure is consumed. In the shape of the smoothly changing flow path, the turbulent energy value is small.

Figure 5. Contours of turbulent kinetic energy
The strain rate of the solution reflects the shearing condition of the solution, and the loss of viscosity is mostly from mechanical shearing. When the strain rate of the solution is studied, the shearing condition of the solution and the viscosity loss can be analyzed. Figure 6 shows the solution contours of strain rate in the flow channel. Observing the cloud image, it can be found that the strain rate of the solution is large at the wall of the straight channel, indicating that the shearing action on the solution mainly occurs at the wall of the straight channel. At these locations, a large velocity gradient leads to a higher shear. When the shear rate is greater than a certain value, the long-chain structure of the macromolecule in the polymer solution will be sheared, and the viscosity of the solution will be correspondingly reduced. Therefore, in order to evaluate the retention rate of the throttling structure, the strain rate of the solution is the criteria of analysis, and only a lower solution strain rate can have a higher solution viscosity retention rate.

The optimization of structural feature size mainly compares the throttling pressure consumption, average solution strain rate to find the optimal structural characteristic parameters among 4 schemes. For the study of the annular groove, the groove spacing and the diameter of the semi-annular groove are the characteristic values when the length of the allocator is fixed. According to the simulation data, the relationship between the pressure drop, the average solution strain rate and the flow rate under different characteristic parameters of the annular groove structure is obtained, as shown in Figure 7 and Figure 8.

**Figure 6.** Contours of strain rate

**Figure 7.** Pressure drop versus flowrate.

**Figure 8.** Strain rate versus flowrate.
It can be found from Figure 7 that under the same flow rate, the throttling pressure drop of each scheme is not much different. When the flow rate is 120 m³/d, the throttling pressure drop is greater than 0.35 MPa. However, the average solution strain rates vary greatly, especially under high flowrate injection conditions, which the strain rate values of each structure differ by more than 1000 S⁻¹. Comparing the throttling pressure loss and the solution strain rate value, the fourth scheme structure has the smallest strain rate, which means that it has the least effect on the mechanical shear degradation of the polymer, and it produces a sufficiently large pressure drop. Therefore, scheme 4 is chosen as the optimal structural solution for the annular groove.

4. Conclusion

The annular groove structure has a high throttling pressure drop and a high solution viscosity retention rate within a limited length. Taking the throttling pressure drop and the average solution strain rate as the evaluation criteria, the optimal flow channel structure is obtained. It can produce a large enough throttling pressure loss and minimize the shearing to the solution which meets the requirements for maximum viscosity retention.

References

[1] X. Chen, H. Duan, Principle and application of polymer single-tube multi-layer injection technology, J. Daqing Petro. Geolo. Devel. 23 (2004) 53-55.
[2] H. Cui, X. Song, and X. Pei, Polyacrymide aqueous solution flows in concentric annulus with equidistant annular grooves in the inner tube, J. Northeast Petro. Univer. 30 (2006) 38-40.
[3] S. Zhang, Research on viscosity analysis method of polymer flooding and injection process, J. Petro. Geolo. & Eng. 2 (2010) 118-120.
[4] C. Geng, Q. Liu, Study on polymer single tube layered injection process, J. Daqing Petro. Geolo. Devel. 25 (2006) 34-36.
[5] J. Li, Y. Wang, Research and application of layered injection tube string, J. Oil Field Machin. 33 (2004) 100-102.
[6] Meng, H. Cui, and C. Wang, Numerical simulation of flow field in corrugated annulus of layered allocator, J. Petro. Machin. 2 (2006) 15-17.
[7] R. Keslerová, K. Kozel, Numerical modelling of incompressible flows for Newtonian and non-Newtonian fluids, J. Math. & Comput. in Simu. 80 (2010) 1783-1794.
[8] Z. Su, D. Lu, Research on polymer flooding process technology, J. Petro. Plan. & Design 3 (2000) 30-32.
[9] M. Su, S. Huang, Computational Fluid Mechanics, third ed., Tsinghua University Press, Beijing, 1997.
[10] F. Wang, Computational Fluid Dynamics Analysis - Principle and Application of CFD Software, Tsinghua University Press, Beijing, 2004.
[11] T. Dyakowski, A. Williams, Modeling turbulent flow within a small diameter hydrocyclone, J. Chem. Eng. 48 (1993) 1143-1152.
[12] R.J. Smith, L.J. Johnston, Automatic grid generation and flow solution for complex geometries, J. AIAA, 34 (1996) 1120-1124.