C.-H. Chen et al. (Eds.)

doi:10.3233/ATDE240754

Study on the Simulation of Plunger Adaptive Motion and Leakage Velocity Calculation Model in Plunger Gas Lift Wells

Lu JIANG[a,b1], Liping JIANG[a,b], Jianyi LIU[c], Heng XUE[a,b], Jingyi ZHANG [a,b], Yong XIAO[a,b], Songyi GUO[a,b]

[a]*Zhenhua Oil Co., Ltd, Beijing 100032, China*
[b]*Chengdu Northern Petroleum Exploration and Development Technology Co. Ltd., Chengdu 610000, Sichuan, China*
[c]*State Key Laboratory of Oil and Gas Reservoir Geology and Exploitation, Southwest Petroleum University, Chengdu 610500, CN*

Abstract. Due to the depleted exploitation of shale gas reservoirs, liquid loading in horizontal wells during the later stages of exploitation is a common problem. Among these, the plunger lift has advantages such as a simple manufacturing process, low operating cost, and good performance, making it widely used in various gas fields. In view of the characteristics of plunger motion in the wellbore, researchers typically employ simulation and laboratory experiments to study both the plunger motion and gas-liquid flow mechanism. Existing simulations and experiments are generally conducted by setting either the plunger movement speed to be constant or the differential pressure between the wellhead and bottomhole to be constant. However, the wellhead pressure changes during the life cycle of gas production in a gas well, transitioning from the initial pressure state to a gradually decreasing pressure after the well is opened. Therefore, it is difficult for existing results to accurately reflect the dynamic coupling relationship between the plunger motion state and the differential pressure between the wellhead and bottomhole during the plunger lifting process. This paper establishes an adaptive simulation model of fluid-structure interaction (FSI) for plunger motion in gas well production, which accounts for decreasing wellhead pressure. And a model is established based on simulation results to calculate the liquid leakage upper of the plunger.

Keywords: Plunger gas lift; Liquid leakage; Gas channeling; Fluid mechanics; COMSOL simulation

1. Introduction

Insufficient formation energy in a gas well to effectively drain the formation water can result in liquid loading[1][2]. The process of drainage gas recovery is commonly employed at production sites to address liquid loading issues and enhance gas well

[1] Corresponding Author, Lu JIANG, Zhenhua Oil Co., Ltd, Beijing 100032, China Chengdu Northern Petroleum Exploration and Development Technology Co. Ltd., Chengdu 610000, Sichuan, China; E-mail: jianglu@zhenhuaoil.com.

productivity[3][4]. Currently, the commonly used drainage gas recovery methods include gas lift, plunger lift, foam drainage, velocity strings, screw pump, and electric submersible pump. Among these, plunger lift stands out as an efficient technology for drainage and gas lifting. It employs plunger tools placed in the wellbore to create a mechanical seal between the lifting gas and liquid, minimizing liquid slippage and maximizing the gas well's energy utilization for efficient liquid drainage[5][6]. Plunger lift, known for its wide applicability, high economic benefits, simple manufacturing process, and stable operation[7][8], has been implemented in over 7,000 wells across various oil and gas fields. These include Changqing Oil field[9], Southwest Oil & Gas field[10], Xinjiang Oil field[11], and Yanchang Oil field[12] in China. It is increasingly becoming a vital technology for stable production in low permeability and tight gas reservoirs[13][14][15].

Laboratory experiments can simulate the process of plunger lift, allowing for an in-depth study of the gas-water flow mechanism during plunger motion[16][17]. Currently, a significant number of simulations focusing on plunger lift experiments are concentrated on measuring liquid leakage and observing gas channeling. The prevalent experimental testing methods for plunger lifts include: leaving the plunger at rest, pulling the plunger at a certain speed using a traction rope, and lifting the plunger under the impact of constant pressure. However, these methods fall short in accurately describing the adaptive motion of the plunger, driven by the pressure difference from the initial opening to steady production.

Currently, simulations are primarily conducted using only the CFD (Computational Fluid Dynamics) module. In the first method for establishing the simulation model of plunger motion in the wellbore, the wellbore area at the plunger's position is left empty, and a grid is generated in the remaining area for simulation calculations. This approach enhances computational efficiency. The second method focuses on studying the liquid leakage volume above the plunger. Here, the plunger is designated as a moving domain, and dynamic grid technology is utilized to simulate its motion in the wellbore. The movement speed of the plunger is preset, without considering the interaction between the plunger and the fluid. Most current simulations are geared towards mechanism analysis, and there is a lack of research on the adaptive simulation of plunger motion, transitioning from rest to movement due to wellhead pressure reduction.

In terms of calculating liquid leakage, the leakage volume of a smooth surface cylindrical plunger can be determined by formulating a fluid flow equation at the plunger-wellbore gap. Currently, scholars primarily categorize the flow at this gap into two types: shear flow and pressure difference flow. They typically represent the leakage volume as a linear superposition of the amounts caused by these two types of flow. However, the research presented in this paper indicates that the leakage volume is predominantly due to the velocity difference between the fluid and the plunger at the gap, which deviates from the conventional superposition method used for calculating liquid leakage volume. To address this, the paper establishes a new calculation model for the velocity distribution of the gap fluid at varying plunger speeds. It then calculates the flow rate through integration, converts this rate into superficial velocity, and finally computes the liquid leakage volume resulting from the velocity difference.

Addressing the current shortcomings in experimental research and simulation of plunger gas lift, this paper utilizes COMSOL multiphysics field coupling software to establish an adaptive motion FSI (Fluid-Structure Interaction) simulation model for the plunger. This model enables analysis of the gas-water two-phase flow mechanism in the wellbore during the process. Additionally, it facilitates the calculation of liquid leakage under various conditions and aids in establishing a model for calculating plunger leakage.

2. Adaptive FSI simulation model for plunger lifting

In this paper, the COMSOL simulation software's CFD module, coupled with the solid mechanics module, is employed for numerical simulation. Mimicking commonly used plunger structures in the field, the plunger seat position is manipulated using a placer. Additionally, various sizes of oil nozzles are installed at the wellhead to simulate gas well production scenarios. During the simulation, the plunger lifts the liquid slug driven by the pressure difference. As the simulation progresses, mirroring actual production, the wellhead pressure gradually decreases. This results in a corresponding decrease in the pressure exerted on both the plunger and liquid slug.

2.1. Physical model

For the FSI simulation of plunger lift, a typical cylindrical plunger is chosen as the subject. the relevant parameters of the plunger are detailed in Table 1.

Table 1. Parameters statistical table of cylindrical plunger

Type	Mass, kg	Length, mm	Diameter, mm	number of teeth
Cylindrical plunger	5.75	458	60	10

2.2. Basic parameters of the model

Given that the vertical depth of most gas wells on site exceeds 1,000 m, creating a model to scale would demand substantial computing power and necessitate prolonged calculation times. Additionally, the significant disparity between the length and width of the wellbore model poses a challenge in achieving convergence when using actual dimensions. To circumvent these issues, a shorter wellbore model is established based on the similarity criterion, still exceeding what is achievable in laboratory conditions, to simulate the plunger's lifting motion after the gas well is opened. The simulated well is vertical, with pipe diameters set to actual on-site dimensions such as 50.6 mm, 62 mm, 76 mm, etc. The pipe length in the model is 100 m, substantially longer than the maximum 15 m typically feasible in laboratory experiments.

Based on the similarity criterion and incorporating the relevant parameters from the current laboratory plunger lift experimental device[18] a three-dimensional model for simulating the plunger lift has been established. In this model, the liquid slug at the top part of the plunger is set to a height of 10 m, while the initial height of the plunger at the bottom of the wellbore is 1.140 m.

The grid used for the simulation calculations adopts a triangular pattern. Due to the plunger having numerous corners, a corner refinement method is employed to enhance the grid division, with a refinement scale factor of 0.25. Additionally, four extra boundary layers are incorporated during the grid division. Before initiating the simulation, various grid numbers are utilized for grid independence verification to avoid errors that may arise from excessive coarsening of the grid. When the number of degrees of freedom exceeds 14,593, the calculated leakage velocity remains essentially unchanged.

A dynamic simulation model for plunger lift is developed based on the actual production conditions of the well. The model's outlet is an open boundary, with its size set to 6mm to simulate the wellhead oil nozzle. For the sake of computational efficiency and to facilitate the convergence of calculation results, the casing part is omitted in the numerical simulation. Instead, a constant pressure boundary of 20MPa is established at the model inlet, representing the formation pressure and providing the necessary lifting energy. The initial pressure in the wellbore is also set to 20MPa to simulate conditions in a high-pressure gas well. Below the plunger, a fixed domain is created to represent a stationary retainer. The model includes water at the top part of the plunger and gas at the bottom, with a distinct gas-water interface present in the initial state.

When the simulation begins, due to the wellbore's internal pressure being higher than the outlet pressure, the gas in the wellbore flows out driven by this pressure difference. As the wellbore output increases, the pressure difference across the plunger—from top to bottom—also increases, prompting the plunger to gradually start moving. During this motion, the liquid slug in the top part of the plunger progressively leaks to the bottom part.

3. Model for liquid leakage velocity

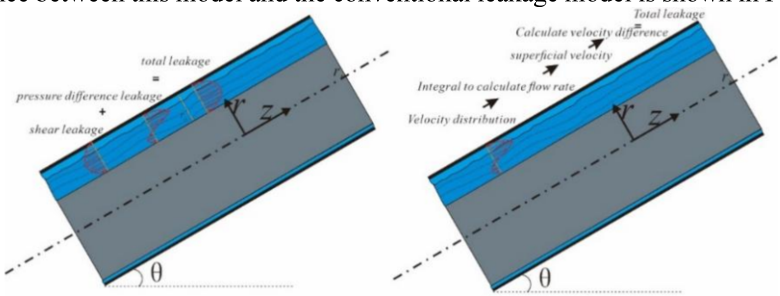
During the liquid lifting process, the plunger moves in relation to the inner wall of the tubing, maintaining a certain gap. This gap and the resulting fluid flow within it significantly influence the plunger's lifting efficiency. An excessively large clearance can lead to increased leakage speed of the lifting fluid, thereby reducing the efficiency of the lift. Conversely, too small a clearance can impede the plunger's descent, extending the duration of each plunger lifting cycle and thus affecting the overall efficiency.

Current research on the liquid leakage velocity of plungers primarily concentrates on the leakage associated with rod pumps, and significant advancements have been made in this area. It's important to note that the process of plunger leakage differs from that of pump leakage. Under the influence of a pressure difference, the plunger moves upwards, driving the liquid in the gap to also move upward. The leakage of the liquid slug at the top part of the plunger is mainly attributed to the pressure difference between the top and bottom of the plunger. Additionally, the leakage velocity is defined relative to the plunger's motion and is determined by the velocity difference between the plunger and the fluid in the gap.

Conventional models typically view the total leakage of the plunger as a combination of pressure difference leakage and shear leakage. However, based on the analysis presented earlier, it is evident that the velocity difference (between the fluid in the gap and the plunger) more accurately represents leakage. In light of this, this paper established a new calculation model that accounts for the liquid flow velocity in the gap between the plunger and the tubing, taking into consideration both the pressure difference and motion shear. Moreover, it integrates the relative motion velocity into the leakage calculation. This approach differs from conventional models in that it does not merely superimpose pressure difference and shear effects. Instead, it offers an understanding of the leakage mechanism that is more aligned with the actual behavior of the plunger.

3.1. Model established

In the gap between the plunger and the tubing, there is a specific distribution of liquid velocity, which is influenced by the motion velocity of the plunger. Utilizing the incompressible fluid Navier-Stokes equation and considering the impacts of gravity, pressure difference, and the plunger's shear effect on the fluid, a fluid motion velocity model has been established. This model enables the calculation of the flow rate in the gap between the plunger and the pipe wall, accomplished through integration. The difference between this model and the conventional leakage model is shown in Figure 1.



(a) The conventional leakage model (b) The velocity difference model

Figure 1. Differences between the model in this paper and the conventional model

When considering the movement of the plunger, it has a lifting effect on the clearance fluid, which drives the fluid to move upwards. Any position at the gap follows the Navier Stokes equation:

$$-g\sin\theta-\frac{1}{\rho_{\mathrm{L}}}\frac{\partial p}{\partial z}+\frac{\mu_{\mathrm{L}}}{\rho_{\mathrm{L}}}\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial v_{\mathrm{z}}}{\partial r}\right)=0 \quad (1)$$

Where, g is the acceleration of gravity, m/s2; μL is the liquid viscosity, Pa·s; P is the pressure, Pa; vz is axial velocity, m/s; z is the axial coordinate axis; r is the radial coordinate axis.

The outer boundary condition and the inner boundary as follows:

$$v_z\big|_{r=r_{\mathrm{t}}}=0\ ,\ v_z\big|_{r=r_{\mathrm{t}}-\delta}=v_{\mathrm{p}} \quad (2)$$

Where, r_{t} is the radius of pipe, m;

For any small section of pipe Δz the pressure difference is constant, and the general solution of the above equation is:

$$v_z=c_1\ln r+c_2+\frac{\rho_L}{4\mu_L}r^2\left(g\sin\theta+\frac{1}{\rho_L}\frac{\partial p}{\partial z}\right) \quad (3)$$

Substitute the boundary conditions and solve as:

$$c_1=\frac{v_{\mathrm{p}}-c_3\left(\left(r_{\mathrm{t}}-\delta\right)^2-r_{\mathrm{t}}^2\right)}{\left(\ln\frac{r_{\mathrm{t}}-\delta}{r_{\mathrm{t}}}\right)},c_2=-r_{\mathrm{t}}^2c_3-\frac{v_{\mathrm{p}}-c_3\left(\left(r_{\mathrm{t}}-\delta\right)^2-r_{\mathrm{t}}^2\right)}{\left(\ln\frac{r_{\mathrm{t}}-\delta}{r_{\mathrm{t}}}\right)}\ln r_{\mathrm{t}},c_3=\frac{\rho_L}{4\mu_L}\left(g\sin\theta+\frac{1}{\rho_L}\frac{\partial p}{\partial z}\right) \quad (4)$$

Given the coordinates of any point in the gap, after calculating the distance δ based on the positional relationship, the liquid velocity at that position can be obtained by substituting into Eq. (8).

The calculated velocity vz is the flow velocity of the liquid under the pressure difference, shear, and liquid sliding. The two-dimensional integration of this velocity is used to calculate the flow rate at the tangent plane (Figure 2).

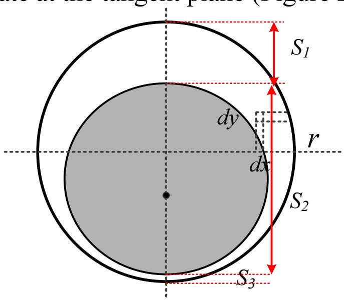


Figure 2. Integral diagram at tangent plane

Divide the liquid film into three regions, and the total liquid flow rate on the cross-section can be expressed as integrating the liquid velocity of the three regions. Similarly, the following equation can be solved through numerical integration:

$$q_{\mathrm{sp}} = 2\left(\int Q_{S_1} + \int Q_{S_2} + \int Q_{S_3}\right) = 2\left(\int_{-r_\mathrm{t}}^{-r_\mathrm{p}-e}\int_0^{\sqrt{r_\mathrm{t}^2-y^2}} v_z dxdy + \int_{-r_\mathrm{p}-e}^{r_\mathrm{p}-e}\int_{\sqrt{r_\mathrm{p}^2-(y+e)^2}}^{\sqrt{r_\mathrm{t}^2-y^2}} v_z dxdy + \int_{r_\mathrm{p}-e}^{r_\mathrm{t}}\int_0^{\sqrt{r_\mathrm{t}^2-y^2}} v_z dxdy\right) \quad (5)$$

So, the average velocity at the gap and the speed difference between the plunger and the liquid can be expressed as:

$$v_\mathrm{L} = \frac{q_{\mathrm{sp}}}{\pi\left(r_\mathrm{t}^2 - r_\mathrm{p}^2\right)}, v_{\mathrm{re}} = v_\mathrm{p} - v_\mathrm{L} \quad (6)$$

The liquid -leakage rate during the time period t is:

$$vm_\mathrm{L} = \frac{\rho_\mathrm{L}\pi\left(r_\mathrm{t}^2 - r_\mathrm{p}^2\right)v_{\mathrm{re}}}{t} \quad (7)$$

3.2. Correction of leakage model

Subsequently, a refined calculation model for the leakage of a rod-shaped plunger has been derived, which is based on simulation results. Given that current experimental testing for plunger leakage is typically conducted in pipe sections no longer than 10 meters, the plunger is still in its acceleration phase during these tests. As a result, the pressure difference at both ends of the plunger does not equate to its gravitational pressure difference and cannot be accurately measured. This discrepancy makes it challenging to use existing experimental data to correct the model presented in this paper. Consequently, this section focuses on refining the calculation model primarily through the data derived from the simulation of plunger motion.

The turbulent sealing coefficient related to velocity difference and clearance is defined as follows:

$$v_{\mathrm{leak}} = \lambda v_{\mathrm{leak\_m}}$$
$$\lambda \sim \left(v_{\mathrm{gp}}, d\right) \quad (8)$$

Where, vleak is the leakage velocity by simulating, kg/s; vleak_m is the leakage velocity calculated by model, kg/s; vgp is the average velocity difference between the plunger and the liquid, m/s; d is the gap between the plunger and the tubing, m.

The relationship between velocity difference and leakage velocity when the gap is 3mm is shown in Figure 3. Excluding several previous points, there is a clear linear relationship between leakage velocity and velocity difference. The comparison between the corrected leakage velocity using this linear expression and the simulated leakage velocity is shown in Figure 4. Except for a few unstable points in the early stage, the corrected leakage velocity is very close to the simulated velocity.

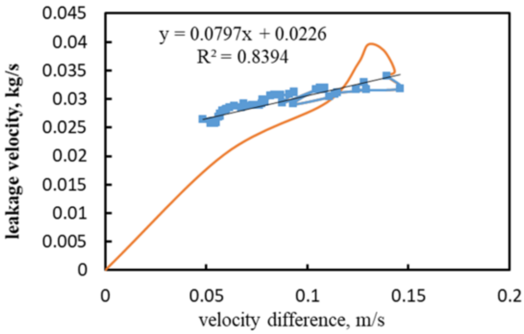


Figure 3. Relationship between velocity difference and leakage velocity with a gap of 3mm

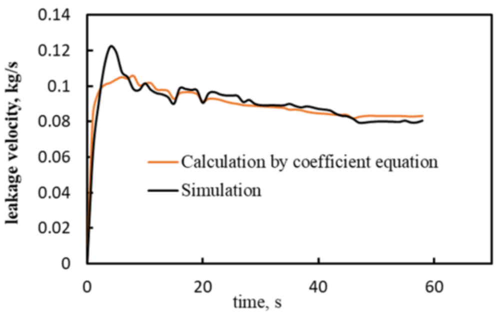


Figure 4. Corrected Leakage velocity

Similarly, the linear relationship between the turbulent sealing coefficient and the velocity difference is calculated for 1mm, 3mm, 5mm, and 6.5mm, as shown in Table 2.

Table 2. Relationship between Turbulent Sealing Coefficient and Velocity Difference for Different Gap Sizes

Gap width	equation	R^2
1	y=0.3565x+0.0816	0.8150
3	y=0.0741x+0.0227	0.8210
5	y=0.0583x+0.0105	0.8685
6.5	y=0.0281x+0.0101	0.8452

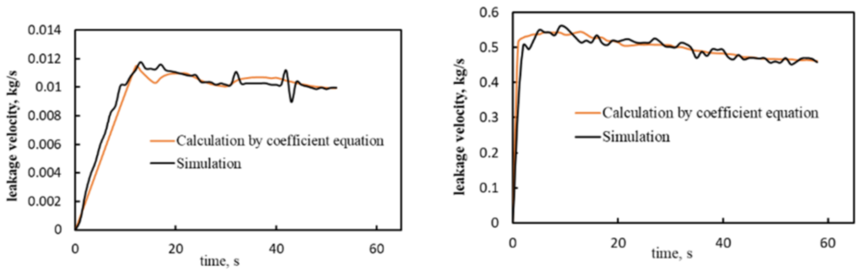
Thus, the relationship between coefficient a, b, and gap size is:

$$a = 0.3476\delta^{-1.271}$$
$$b = 0.0806\delta^{-1.172} \tag{9}$$

The calculation model for corrected leakage velocity is obtained as follows:

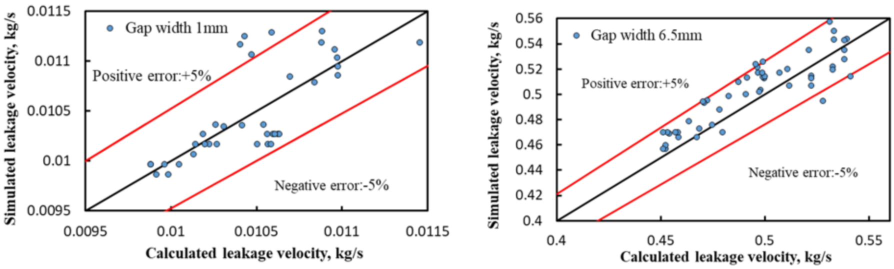
$$vm_{\text{leak}} = \left(av_{\text{gp}} + b\right) vm_{\text{L}} \tag{10}$$

The summary of the comparison chart between the calculated leakage velocity of the model and the simulated leakage velocity is shown in **Fig. 5**, and the error limit comparison curve is shown in the **Fig. 6**. The modified model has a high degree of agreement with the simulated data.



(a) Gap width 1mm **(b) Gap width 6.5mm**

Figure 5. Comparison between the calculated and simulated results of the modified model



(a) Gap width 1mm, average relative error 2.692% (b) Gap width 6.5mm, average relative error 3.529%

Figure 6. Error Limit Curve

4. Conclusions

By establishing a fluid-structure interaction simulation calculation model for the plunger lift, the adaptive motion process of the plunger lift is simulated. The comparison of simulation results with the leakage calculations for a smooth rod plunger reveals a significant discrepancy: the leakage velocity calculated without accounting for the turbulent sealing effect is substantially higher than what is observed in the simulations. By analyzing the fluid forces within the plunger gap, a new method for calculating the liquid phase leakage velocity has been developed. The results obtained from this method demonstrate a high level of concordance with the simulation results, indicating the effectiveness of the model established in this paper.

References

[1]Zhang P, Cheng X, Liu R, et al. Abnormal liquid loading in gas wells of the Samandepe Gasfield in Turkmenistan and countermeasures[J]. Natural Gas Industry B, 2015, 2(4): 341-346.
[2]Tugan M F. Deliquification techniques for conventional and unconventional gas wells: Review, field cases and lessons learned for mitigation of liquid loading[J]. Journal of Natural Gas Science and Engineering, 2020, 83: 103568.
[3]P.J. Waltrich, G. Falcone, J. Barbosa. Liquid transport during gas flow transients applied to liquid loading in long vertical pipes. Exp. Therm. Fluid Sci., 68 (2015), pp. 652-662.
[4]Waltrich P J, Posada C, Martinez J, et al. Experimental investigation on the prediction of liquid loading initiation in gas wells using a long vertical tube[J]. Journal of Natural Gas Science and Engineering, 2015, 26: 1515-1529.
[5]R. Schmitz, G. Steele. Combining plunger lift and compression to lift liquids in low rate gas wells Paper Presented at the Petroleum Society's 7th Canadian International Petroleum Conference (57th Annual Technical Meeting), Calgary, 13-15 June (2006)
[6]Darden D P. Plunger Lift Application in Wells With Set Packers or Permanent Tubing[J]. Journal of Petroleum Technology, 1994, 46(01): 6-6.
[7]Hassouna M. Plunger lift applications: challenges and economics[C]//North Africa Technical Conference and Exhibition. OnePetro, 2013.
[8]Schwall G H. Case Histories: Plunger Lift Boosts Production in Deep Appalachian Gas Wells[C]//SPE Production Operations Symposium. OnePetro, 1989.
[9]YANG Quanwei, ZHOU Shaodan, MA Lianwei, et al. Application and Effect Evaluation of Plunger Gas Lift Technology in Shenmu Gas Field[J]. Drilling and Production Technology, 2022, 45(1): 139-143.
[10]LI Bochun . XIANG Jianhua , XIAO Fan , et al . Large-scale Application and Effect Analysis of Plunger gas Lift Technology in Changning Shale Gas Reservoir [J] . Drilling and Production Technology , 2023 , 46 (2) : 65-70.
[11]Duan Yuming , Wang Faxing, Lei Tengjiao, Li Yan, Dong Fengshuo. Design of Continuous Plunger Gas Lift Process and Its Application [J] . Drilling and Production Technology , 2020,43(01):35-37+9.
[12]Yan Yingfei, Tang Yong huai, Yao Jun. Application of the plunger air lift draining gas recovery technology in Yanchang gas field [J]. Petrochemical Industry Technology, 2020, 27(07): 80+86.
[13]Nascimento C M M, Becze A, Virues C J J, et al. Using dynamic simulations to optimize the start-up of horizontal wells and evaluate plunger lift capability: Horn River Shale Gas Trajectory-Based Case Study[C]//SPE Western Regional Meeting. SPE, 2015: SPE-174053-MS.
[14]Nurkas Z. Case Study: Plunger Lift Application Doubled Oil Production[C]//SPE Annual Caspian Technical Conference. OnePetro, 2020.
[15]Kravits M, Frear R, Bordwell D. Analysis of plunger lift applications in the Marcellus shale[C]//SPE annual technical conference and exhibition. OnePetro, 2011.
[16]Zhao K, Tian W, Li X, et al. A physical model for liquid leakage flow rate during plunger lifting process in gas wells[J]. Journal of Natural Gas Science and Engineering, 2018, 49: 32-40.
[17]Zhao K, Bai B. Transient liquid leakage during plunger lifting process in gas wells[J]. Journal of Natural Gas Science and Engineering, 2018, 59: 250-261.
[18]Zhao K, Bai B. Instantaneous liquid and gas leakage flow during plunger lifting process in vertical wells[C]//11th North American Conference on Multiphase Production Technology. OnePetro, 2018.