Research on Structural Design of Friction Test Based on Finite Element Analysis Heory

Xiao Liu
Shengli College China University of Petroleum, 257061, Shandong, China
slytlu@163.com

Abstract. The thesis uses the ring upsetting method to study the determination of the friction coefficient in volume forming. Use finite element analysis to determine a set of friction coefficient calibration curves, and use the ring upset test to determine the percentage change rate of the ring inner diameter. The relationship curve between the ratio and the compression percentage of the ring height. By comparing the curves, the friction coefficient between the mold material and the forming material is obtained. The friction coefficient between the 6A02CZ material and the steel was determined when the 6A02CZ material was volume shaped in the steel mold. The friction coefficients of the 6A02CS material in the absence of lubricants and lubricants were 0.325 and 0.3, respectively. Studies have shown that the method of combining the finite element method and the ring upsetting test is more suitable for the finite element analysis in plastic volume forming.

1. Introduction
In the actual production of plastic forming, the friction coefficient between parts and equipment is often needed to guide the actual production. Because there are many factors that affect the coefficient of friction, and these factors are difficult to quantitatively determine the impact on the coefficient of friction. At the same time, in the plastic forming process, various influencing factors are constantly changing. Therefore, the friction coefficient also changes throughout the plastic forming process. Therefore, the friction coefficient measured by any method is only an approximate average value. Before the ring upsetting experiment, according to the original size of the ring and the possible size after deformation, the theoretical formula of the ring upsetting deformation (to find the position of the neutral layer) must be used to draw the ring height h and the inner diameter d The relationship curve between the friction factor m of the contact surface is the theoretical calibration curve.

At present, the finite element theory is basically mature, but there are still some problems in the application process, which affects the speed of popularization of the technology. For example, the selection of friction model and the determination of friction coefficient in the finite element analysis of plastic forming, the accurate description of the material model, etc. In this paper, the friction coefficient between 6A02CS and steel was determined by the ring upsetting method combined with finite element analysis [1].
2. Transient temperature field analysis of friction contacts of friction plate

2.1. Brake upper belt-upper belt friction plate finite element model

2.1.1. Material thermophysical parameters. The thermal conductivity of the brake upper belt is 48W/(m·K), the specific heat is 460J/(kg·K), and the thermal expansion coefficient is 11.5×10⁻⁶ m/K; the thermal conductivity of the upper friction plate is 100W/(m·K), specific heat is 9.8J/(kg·K), thermal expansion coefficient is 2×10⁻⁶ m/K [2].

2.1.2. Braking heat flux density. The kinetic energy of the band brake during operation is converted into a large amount of heat energy. This heat is distributed among the friction pair parts in the form of heat flow. Ignoring the effect of wear, the band brake heat flux density can be expressed as:

\[ q(t) = \mu \cdot p(t) \cdot v(t) \]  

In the formula, \( \mu \) is the friction coefficient; \( p(t) \) is the specific pressure on the friction surface (N/m²); \( v(t) \) is the relative moving speed of the brake belt (m/s). Among them, the ratio on the friction surface the pressure \( p(t) \) can be obtained from reference. This article assumes that its value has nothing to do with time, but only with spatial position.

In actual work, the relative moving speed of the brake belt is the instantaneous rotation speed of the anchor sprocket. During braking, it changes with time. When the ship drops anchor, in the process of the anchor chain gradually falling, part of the anchor the gravitational potential energy of the chain and anchor is converted into the kinetic energy of the anchor chain participating in the movement, the kinetic energy of the anchor sprocket, and the kinetic energy of the main axis of the anchor chain. According to the law of conservation of energy and the 72% of the machinery for efficiency loss, the instantaneous speed of the anchor sprocket before braking is 18.58 m/s.

2.1.3. Heat flow distribution coefficient. The heat flow distribution coefficient is:

\[ \gamma = \sqrt{\frac{k_b c_b \rho_b}{k_p c_p \rho_p}} \]  

\( G \rho =11.01. \) In the formula, \( \rho \) is the density, \( c \) is the specific heat, \( k \) is the thermal conductivity coefficient, the subscripts \( b \) and \( p \) denote the brake upper belt and the upper belt friction plate, that is \( \rho_b \) is the density of the brake band, and \( \rho_p \) is the density of the friction plate. Bring the thermal properties of the material into equation (2), we get: \( G=11.01. \)

2.1.4. Determination of convection heat transfer coefficient and friction surface heat conduction coefficient on the brake. The average heat exchange coefficient of the band brake and the sea breeze is obtained from the empirical formula:

\[ \alpha = 0.664 \times (\frac{\lambda}{l}) \times (\frac{\sigma l}{v})^{1/2} \times (P_{\infty})^{1/3} \approx 4.5 \]  

Among them, it is assumed that the sea breeze wind speed \( \sigma =10 \) m/s; the length of the belt brake \( l=7.5 \) m. The thermophysical parameters of the sea breeze are as follows: kinematic viscosity \( v=15.78 \times 10^{-6} \) m²/s; Prandtl constant \( P_{\infty}=0.71; \) thermal conductivity \( 26.1 \times 10^{3} \) W/(m·K). Reference, the contact heat transfer coefficient between the brake upper belt and the upper belt friction plate is \( 2 \times 10^{4} \) W/(m·K).
2.1.5. Brake upper belt-upper friction plate finite element model. Apply the heat flux density that changes with space and time to the inner surface of the upper friction plate, define the two contact bodies of the upper friction plate and the brake upper belt, and specify the contact heat transfer properties: ambient temperature, convection heat release coefficient with the ambient medium. The convective heat release coefficient between the contact body and the contact body during contact heat transfer. At the same time, the initial temperature of the upper friction plate and the upper brake belt is defined as the ambient temperature of 20°C.

2.2. Transient thermal analysis results

In order to observe the transient temperature distribution, some nodes were selected on the inner and outer surfaces of the brake upper belt and the upper belt friction lining. The selected nodes on the inner surface of the upper belt friction lining are shown in Figure 1, and the time-varying curve is shown in Figure 2 [3].

![Figure 1. Selected nodes on the inner surface of the friction plate.](image)

![Figure 2. The temperature variation curve of selected nodes on the inner surface of the upper friction plate with time.](image)

It can be seen from Figure 2: In the initial stage of braking (about 0s ~ 0.1s), due to the high speed of the anchor sprocket, the heat flux density input as a thermal boundary condition is also very large. And under emergency braking conditions, due to The braking time is very short, and the frictional heat is too late to contact the heat conduction to the brake, so the temperature of each selected point rises quickly; in the middle of braking (about 0.1s ~ 0.4s), as the speed of the anchor sprocket decreases, The input heat flux density also continues to decrease. At the same time, due to the contact heat conduction between the upper friction plate and the brake upper belt and the convection heat exchange between the upper friction plate and the air, the temperature at each selected point begins to slowly
decrease. The main reason is that the upper friction plate is organic asbestos-free, is a good conductor of heat, and has a high thermal conductivity. The effect of contact heat conduction and convection heat transfer on the moving belt is greater than the effect of the heat flux density input as a thermal boundary condition, and the temperature drop on the inner surface of the upper friction plate accelerates [4].

3. Establishment and solution of the finite element model of the ring upset friction plate

Design Modeler is abbreviated as DM, which is the modeling platform of ANSYS Workbench. DM can be fully parameterized for solid modeling, and has the functions of model creation, CAD model repair, CAD model simplification, and conceptual model creation. In order to facilitate the follow-up thermal analysis, this article is modeled in the DM that comes with ANSYS. The model is shown in Figure 3. Import the 3D model of the ring upset friction plate into the ANSYS transient thermal analysis module for solution. The specific solution steps are as follows:

![Finite element model of circular upset friction plate](image)

**Figure 3.** Finite element model of circular upset friction plate.

1. Set the friction plate density, heat capacity and other material properties; 2. Mesh the friction plate; 3. Add boundary conditions and initial conditions, including temperature, heat flow, thermal conductivity, etc.; 4. Insert the solution term, and set the solution time step to solve.

3.1. Friction torque and pressure at the joint surface

During the upsetting process of the ring, due to the existence of friction on the contact surface of the friction plate, it is subjected to a rotational torque perpendicular to the axial direction. In the finite element analysis, according to the calculation result of the friction force, each joint surface the frictional force at each node is loaded at the corresponding position to simulate the rotational torque experienced by the friction plate. According to the dynamic analysis result of the ring upset joint characteristic, the corresponding pressure load is applied on the joint surface of the friction plate finite element model. Simulate the pressure of the friction plate when the ring is upset.

3.2. Solid modeling

Take the outer friction plate as an example. The material of the friction plate is No. 15 steel. Create a geometric model, select materials, set the mesh element size, and generate mesh elements for finite element analysis. The Pro / MECHANICA finite element module uses adaptive P-method technology to automatically mesh. In the P-method technique, the displacement equations of each finite element are high-order polynomials, and the displacement equations of each finite element are linear, quadratic,
or cubic. Therefore, the P-method technique can be used to accurately fit the geometric shape and large stress gradient, and in the actual calculation process, the element first performs a preliminary calculation at a lower order, and then where the stress gradient is larger and the calculation accuracy is required Higher places automatically increase the order of the stress equation of the element, thereby ensuring the accuracy and efficiency of the calculation [5].

The P-method is used to divide the finite element mesh, and some optional features in the model must be simplified. In the outer friction plate model, the features such as rounded corners and chamfers between the plane and the surface, spline and surface are removed, and these features are simplified into one point. In this way, on the one hand, it facilitates meshing, on the other hand, it reduces the total number of generated cells, which reduces the workload of finite element calculation. After the meshing is completed, a total of 1767 units are generated. The finite element mesh is shown in Figure 4.

![Finite element mesh](image)

**Figure 4.** Finite element analysis model.

### 4. Experimental research

4.1. **The effect of $\mu$ at the time of occlusion on the final forming**

First, when $\mu$ is small, the inner diameter change trend is increasing. The amount of increase in inner diameter when considering occlusion is smaller than the amount of increase in inner diameter when not considering occlusion, which is equivalent to hindering the increase in inner diameter. Second, when $\mu$ is larger, the change trend of the inner diameter is reduced. The reduction in inner diameter when considering bite is larger than the reduction in inner diameter when not considering bite, which is equivalent to promoting the reduction in inner diameter [6].

In general, the larger $\mu'$ in the bite stage (because the actual $\mu$ in the bite is larger than the average $\mu$ in the forging process. Therefore, the effect of $\mu' = \mu \times 1.5$ in this experiment) is to reduce the inner diameter the trend of. In this experiment, the process of occlusion was 1 unit time and the depression process was 59 unit times; a total of 60 unit times and the depression amount was 6 units. Although the $\mu$ action time in the occlusal phase is very short, it has a great influence on the entire plastic deformation results. After the initial occlusal phase, this effect will continue and expand. Therefore, using the finite element model to draw the friction calibration curve needs to refer to the lubrication mechanism in actual production to modify the model, as shown in Figure 5.
Figure 5. The effect of μ at the time of occlusion on the final forming.

From the study in Figure 5, it is found that when μ=0.05, the inner diameter increases first and then decreases, indicating that the neutral layer is near the initial inner diameter in this case, and the inner diameter basically does not change after large deformation. When μ>0.05, the inner diameter of the ring shrinks obviously. After μ reaches 0.3, it can be observed that the gap between the curves is not large, indicating that when μ=0.3, 0.4, the forming is close to the limit, and the deformation is difficult. When μ<0.05, the distance between the curves is large, and the specimen has a large amount of deformation, showing good plasticity.

4.2. Thermal finite element analysis

The thermal deformation of the friction plate after heating is mainly along the axial and radial directions, and its value gradually increases from the middle position to both sides, and its maximum value is on the outer edge of the friction plate. After being heated, the two ends are deformed in the axial direction away from the friction surface, but because the friction plate is fixed on the driven disk by the diaphragm spring, a wave-shaped deformation is generated. This deformation exacerbates wear. In order to reduce the amount of deformation, the shape of the friction plate is designed to be slightly concave when designing, so that when the thermal load acts, it can offset the axial deformation at both ends, so that the friction plate is better combined with the pressure plate and the flywheel [7].

5. Conclusion

The finite element method was used to analyze the transient temperature field of friction plates with different oil groove widths, and the curve of maximum temperature with time and radial temperature distribution were obtained. Due to the influence of the oil groove, the radial temperature of the friction plate shows a stepwise distribution. The temperature difference between the bottom of the oil groove and the friction surface is large during the bonding period. The friction coefficient of 6A02CS material was measured without lubricant and lubricating oil, and the calibration curve was checked through the ring upsetting test to determine that the friction coefficient of 6A02CS material in both cases was 0.325 and 0.3.

References

[1] Richard A. Albert, & John W. Rudnicki. Finite element simulations of tennessee marble under plane strain laboratory testing: effects of sample–platen friction on shear band onset. Mechanics of materials, Vol. 33 (2016) No. 1, p. 47-60.
[2] Wang, Y., Chen, X., Young, R., & Kinloch, I. Finite element analysis of effect of inter-yarn friction on ballistic impact response of woven fabrics. Composite Structures, Vol. 135 (2016) No. 1, p.8-16.
[3] Jain, R., Pal, S. K., & Singh, S. B. Finite element simulation of temperature and strain
distribution during friction stir welding of aa2024 aluminum alloy. Journal of the Institution of Engineers, Vol. 98 (2017) No. 1, p. 37-43.

[4] Nam, J., Choi, H., & Kang, J. Finite element analysis for friction noise of simplified hip joint and its experimental validation. Journal of Mechanical Science and Technology, Vol. 30 (2016) No. 8, p.3453-3460.

[5] Zhao, X., & Li, Z. A solution of transient rolling contact with velocity dependent friction by the explicit finite element method. Engineering Computations, Vol. 33 (2016) No. 4, p.1033-1050.

[6] Djoko, J. K., & Mbehou, M. Finite element analysis of the stationary power-law stokes equations driven by friction boundary conditions. Journal of numerical mathematics, Vol. 23 (2015) No. 1, p. 21-40.

[7] Lai, V. V. Chielo, O., Brunel, Jean-François, & Dufrénoy, Philippe. Full finite element models and reduction strategies for the simulation of friction-induced vibrations of rolling contact systems. Journal of Sound & Vibration, Vol. 444 (2019) No. 5, p. 197-215.