Finite Element Analysis on the Crack Growth and Stress Intensity Factor for the Contact Fatigue

P X Du\textsuperscript{1,3,4}, Y L Liang\textsuperscript{1,3,4,*}, C H Yin\textsuperscript{2,3,4}, L Q Zhong\textsuperscript{2,3,4}, and H J Fan\textsuperscript{1,3,4}

\textsuperscript{1}College of Materials and Metallurgy, Guizhou University, Guiyang, Guizhou, People's Republic of China
\textsuperscript{2}School of Mechanical Engineering, Guizhou University, Guiyang, Guizhou, People's Republic of China
\textsuperscript{3}Guizhou Key Laboratory for Mechanical Behavior and Micro Structure of Materials, Guizhou University, People's Republic of China
\textsuperscript{4}National & Local Joint Engineering Laboratory for High-performance Metal Structure Material and Advanced Manufacturing Technology, People's Republic of China.

E-mail: liangyilong@126.com

Abstract. In order to simulate contact fatigue crack propagation, the ANSYS APDL and Franc3D were used. The rolling contact model was established by the ANSYS APDL, then the distribution of the contact stress field under various loads was analyzed. The contact stress value demonstrated an error within ± 6%, compared to the obtained results from the Hertz theory. The contact stress field was imported into the Franc3D with a preset radical crack, the crack tip stress intensity factor distribution was calculated and the growth path of the three-dimensional contact fatigue cracks was analyzed. The comparisons with the experimental results demonstrated that the fatigue crack profile path from the peeling position of the subcutaneous layer was consistent with the simulated path, which was in a circular arc form. The main reason for this result was the fatigue crack change from the opening type I to the sliding type II. The a-N curves of the crack growth life under various load conditions were predicted based on the Franc3D, whereas the results displayed that the crack growth life decreased as the compressive stress increased.

1. Introduction
The contact fatigue failure phenomena occur frequently during the service of gears, rails, rolling bearings and other components and parts, mainly due to the surface fatigue damage of the workpieces which sustain a long-term cycle contact stress. The contact fatigue can cause serious damage. In 1998, the German high speed train derailment with serious casualties was caused by the wheel contact fatigue[1-2]. For a long time, the contact fatigue life extension of the workpieces has effectively been studied by a high number of scholars both locally and abroad. With the development of computer technology in recent years, the finite element method has been applied for the contact problems study, which can predict the contact fatigue life effectively and also minimize a high amount of manpower and financial resources. Wu[3] simulated the surface cracks under stretching and pure bending fatigue loads according to the Paris fatigue crack growth law, through the crack tip stress intensity factor(SIF) distribution fitted a cubic spline function curve, solving that the crack tip SIF distribution is difficulty measured. Shi[4]
established a thermal fatigue crack model of the brake disc by the Franc3D. The crack growth speed and the stress intensity factor change were studied with various crack ratios and multiple cracks shape, then the distribution regularity was obtained under each conditions. Wang et al.[5] simulated the crack initiation life of face gear under various working conditions and with various surface properties based on the MSC.Fatigue software. Then the crack growth life on the tooth root was simulated based on the Franc3D, finally discovering the bending life prediction of the face gear. When bending moment and stress ratio were constant, Janusz Lewandowski et al.[6] studied the crack growth behavior of a plate specimen with the two sides passivation notch and stripe weld respectively using the Franc3D. The fatigue crack growth life was calculated according to the fatigue mechanism and boundary element method. The results proved that the fatigue crack growth life was improved significantly subsequently to welding. In condition of increased temperature and sustained loading, Erik Storgärds et al.[7] expanded the crack growth results of the nickel-based Inconel 718 from the one dimensional to the three dimensional in the Franc3D. The crack growth life was calculated when the stress intensity factor on crack tip changed, which was consistent with the experimental results. A series of engineering problems have been solved by a high number of scholars worldwide in the Franc3D, however, only a low number of studies were focused on the contact fatigue crack growth behavior. The study of crack tip stress intensity factor change trend n means of the finite element analysis method is proven of guiding significance for the contact fatigue crack growth behavior research.

In this paper, the rolling contact model was established in both ANSYS APDL and Franc3D. After the initial crack was preset in the contact area, the change of stress intensity factor of crack under various contact stress fields were calculated. The Sweep method of volume mesh was utilized for the model mesh generation. The fix constraint was set for the bottom of the lower cylindrical, whereas the bottom of the upper cylindrical (See figure 1) was set as 1 kN, 1.5 kN, 2 kN, 2.5 kN and 3 kN (loads selected according to the contact fatigue experiment) of the normal compressive stress, respectively. The other boundary conditions were given, the point-surface contact pair was set, the upper cylindrical surface was the contact surface and the lower cylindrical surface was the target surface.

2. Experimental method

2.1. Finite element analysis
The test material was 42CrMo with the elasticity modulus of E=2.12×10^5 MPa, the density of ρ=7850 kg/m^3 and the Poisson ratio of λ=0.28. The solid 185 was utilized as the finite element and the control unit size was 1 mm. The Sweep method of volume mesh was utilized for the model mesh generation. The fix constraint was set for the bottom of the lower cylindrical, whereas the bottom of the upper cylindrical (See figure 1) was set as 1 kN, 1.5 kN, 2 kN, 2.5 kN and 3 kN (loads selected according to the contact fatigue experiment) of the normal compressive stress, respectively. The other boundary conditions were given, the point-surface contact pair was set, the upper cylindrical surface was the contact surface and the lower cylindrical surface was the target surface.

2.2. Crack preset method
The contact stress field obtained from ANSYS was imported into the Franc3D, two different types of cracks were placed in the radial and the axial direction of the sub model at the contact point. The crack diameter was 1 mm, as presented in figure 2.
2.3. Crack growth analysis
Firstly, select the appropriate crack preset direction. According to the Paris-equation, the radial crack growth with tensile stress was simulated by the Franc3D. The extension step length was set at 0.1 mm. The Paris-equation was written as:

\[
\frac{da}{dN} = C(\Delta K)^n
\]

where \(C\) and \(n\) are the basic coefficients for the crack growth properties description. The values of \(C\) and \(n\) in the 42CrMo crack growth were reported in reference[8], whereas the average values \((C=1.7636 \times 10^{-8},\ n=2.0703)\) were selected in this study.

2.4 Contact fatigue test
The 42CrMo material with the same coefficients as in the simulation model was utilized for thermal refining, oil quenching at 850°C and tempering at 580°C. The chemical composition are listed in table 1.

| Element | C   | Si | Mn | Cr   | Mo  | S    | P    | Fe  |
|---------|-----|----|----|------|-----|------|------|-----|
| Content | 0.41| 0.24| 0.68| 1.03 | 0.18| 0.011| 0.006| Bal.|

According to the YB/T 5345-2006 “Metal rolling contact fatigue test method” standard, five groups of contact fatigue specimens were processed in the same size with the simulation model. The contact fatigue test machine MJP-30 was utilized, a 10% slip was set, the given load was 200kg (corresponding simulation load was 2kN), the contact fatigue tests were executed and the failure sample was analyzed by the field emission scanning electron microscope.

3. Results
3.1. Contact stress field calculation
The calculated results of the ANSYS APDL are listed in Table 2, and the distribution of the contact stress field is presented in figure 2.

**Table 2. Contact stress under various loads**

| Load[kN] | 1   | 1.5 | 2   | 2.5 | 3   |
|----------|-----|-----|-----|-----|-----|
| Contact stress[MPa] | 733.42 | 838.19 | 942.96 | 1047.74 | 1152.51 |

With the assumption that in the simulation state, the instantaneous contact of the sample at any time was a pure theoretical contact, either clearance or wear did not exist at the contact center. The contact stress can be calculated according to the Hertz theory:

\[
\sigma_{\text{max}} = 0.418 \sqrt{\frac{FE}{l}} \left( \frac{R_1 + R_2}{R_1R_2} \right)
\]

where \( E_1, E_2 \) are the elastic modulus of the upper and lower cylinder model material, respectively, \( E_1 = E_2 = 2.12 \times 10^5 \text{MPa} \); \( R_1, R_2 \) are the radii of the upper and lower cylinder model, respectively, \( R_1 = R_2 = 30 \text{ mm} \); \( l \) is the contact width, \( l = 5 \text{ mm} \). The theoretical contact stress values are listed in Table 3, the errors between the simulation value and the theoretical value were within ±6%. The error generated in the actual simulation occurred because the deformation existed between two cylindrical contacts, however, the contact stress field under the deformation behavior was not considered in the theoretical calculation.

**Table 3. Theoretical value of contact stress and error**

| Load[kN] | 1   | 1.5 | 2   | 2.5 | 3   |
|----------|-----|-----|-----|-----|-----|
| Theoretical value[MPa] | 702.84 | 860.72 | 993.87 | 1111.18 | 1217.24 |
| Error[%] | -4.41 | 2.62 | 5.12 | 5.71 | 5.32 |

3.2. Crack preset direction
The calculation results of the KI of the crack tip in the radial and axial directions are presented in figure 3 (a) and (b), respectively. It could be observed that the type I stress intensity factor of the radial crack tip was positive, whereas the value was negative for the axial crack tip. The horizontal axis was represented by the crack front normalized coordinate X, whereas the corresponding crack surface points of X=0 and X=1 are presented in figure 3(c). According to the linear elastic theory, the KI stress intensity factor could be normally written as

\[
K_I = Y\sigma\sqrt{a}
\]

where \( \sigma \) is the nominal stress, \( Y \) is the shape coefficient related to the geometry boundary with the crack body, where in general case \( Y \geq 1 \). It was proved that the radial crack front nominal stress was tensile stress, which accelerated the crack growth; the axial crack front nominal stress was compressive stress and the crack closure phenomenon was generated. It proved that the radial direction is the appropriate direction.
3.3. Contact fatigue test results analysis

The failure mode of five groups of contact fatigue specimens peeled from the subcutaneous layer, is presented in figure 4(a). In general case, the origin of the peeling crack is on the surface. When the crack originated from the surface, the crack could be formed rapidly. Moreover, the oil wedge effect caused by the invasion of the lubricating oil formation also accelerated the crack growth, which would shatter fast finally[11].

The cross-section of the peeling is presented in figure 4(b), whereas it was measured that the deepest failure depth was approximately 195μm to the surface. No crack was observed in the subcutaneous layer, which proved that the failure crack initiated from the surface, because the surface was not strengthened, leading the crack to an easier growth at the surface. On the other hand, the failure crack occurred in the sub-surface easily by the surface strengthening [12].

Figure 5(a) demonstrates the crack morphology where the peeling initiated and the crack type was an intergranular fracture. The bottom morphology of the peeling is presented in figure 5(b), which was overall flat and a layered step pattern partially existed. This indicated that the crack at the peeling bottom still grew in a fatigue form, which was mainly developed by both the crack and the crack group.
Figure 5. Peeling failure morphology

3.4. Crack path analysis

Figure 6(a) demonstrates the crack growth by the Franc3D, where it could be observed that the crack expanded along the depth direction and the crack path was of a circular arc shape. Figure 6(b) presents the received SEM images from the cross-section of the peeling, where the section crack shape was in accordance with the Franc3D simulation result.

Figure 6. Crack path(a,Franc3D and b,contact fatigue test)

Figure 7(a) demonstrates the distribution of the KI stress intensity factor during crack growth. The change trend of KI at the position of the 1-15 steps crack growth path is presented in figure 7(b). It could be observed that in the 1-6 steps of the crack growth, the KI increased gradually, whereas in the 6-15 steps, the KI decreased entirely. figure 7(c) presents the distribution of the KII stress intensity factor during crack growth, the change trend of KII at the position of the 1-15 steps crack growth path is presented in figure 7(d). In the 1-6 steps of the crack growth, no apparent change could be observed for the KII, while the KII increased gradually with the expansion in the 6-15 steps. This was interpreted that the contact fatigue crack grew in the open type I from the beginning and continues growing subsequently to becoming the open type II crack, leading the crack growth direction changed and the crack growth section was finally of a circular arc shape.
3.5. Crack growth life numerical calculation

In order to predict the contact fatigue crack growth life under various load conditions, the crack growth life at the cyclic load \( R = -\infty \) was simulated in the Franc3D, and the results are presented in figure 8. It could be observed that following the crack growth of 1 mm, the latter entered the rapid expanding region, when the contact fatigue specimen was considered as invalid. The final crack length was within 1.5 mm. The crack growth cycle number with the same steps under various load conditions is presented in Table 4. The crack cycle number decreased as the stress pressure increased, which indicated that the stress pressure increase speeded up the contact fatigue crack growth [13].

| Load[kN] | 1   | 1.5 | 2   | 2.5 | 3   |
|----------|-----|-----|-----|-----|-----|
| Crack length/mm |     |     |     |     |     |
| cycles   |     |     |     |     |     |

**Figure 7.** Distribution of stress intensity factor KI(a and b) and KII(c and d)

**Figure 8.** Relationship between crack growth length and cycle number under various load conditions

**Table 4.** Crack growth cycle number with the same steps under various load conditions
4. Conclusions
The main conclusions of the present work are as follows:
(1) The rolling contact stress value under various stress pressure states was calculated by the finite element calculation; the error between the calculated results and the Hertz theory results was within ±6%.
(2) According to the calculation results of the stress intensity factor for various directions based on the Franc3D, the results showed that the stress intensity factor of the axial crack tip was negative. This proved the nominal stress was compressive stress which generated a closure phenomenon into the crack. While the stress intensity factor of the radial crack tip was positive. Namely the nominal stress was tensile which accelerated the crack growth.
(3) The results on the crack growth under various loads and the analysis of the contact fatigue failure demonstrated that the fatigue failure form was peeled off from the subcutaneous layer. The crack initiated from the surface and the fracture morphology was the intergranular fracture. Both numerical and experimental results of the crack growth paths were in a circular arc shape. The a-N curves of the crack growth life demonstrated that the entire crack length was within 1.5 mm. Also, the crack cycle life decreased as the stress pressure increased.

5. References
[1] Zhu J C 1998 J. Chinese Railways 8 48
[2] Dong X M 2000 J. Chinese Railways 11 46-49
[3] Wu Z X 2007 J. Chinese Journal of Applied Mechanics 1 42-46
[4] Shi X L, Li Q and Song Z X 2016 J. Journal of Mechanical Engineering 10 126-132
[5] Wang Y Z, Tian Z M and Hou L W 2014 J. Journal of Beijing University of Aeronautics and Astronautics 2 148-153
[6] Lewandowski J and Rozumek D 2016 J. Theoretical & Applied Fracture Mechanics
[7] Storgärd S, Simonsson K and Sjöström S 2015 J. Theoretical & Applied Fracture Mechanics 81 2-10
[8] The editorial board of mechanical engineering material performance data manual 1995 M. Mechanical engineering material performance data manual 232-236.
[9] Gao C Q and Liu J J 1989 M. Material of adhesive wear and fatigue wear
[10] Zhong QP, Zhou Y and Zhang Z 2014 M. Study of crack
[11] Hong H, Sun J Z and Ma C Y 2001 J. Journal of Shenyang University of Technology 23 463-466
[12] Ma L, Wang M Q and Shi J 2009 J. Chinese Journal of Materials Research 23(3) 251-256
[13] Song X 2009 D. Harbin University of Science and Technology.

Acknowledgment
The authors are especially grateful to the Scientific Cooperation Major Project of Guizhou Province, China (Grant No. [2014]6012).