Finite element simulation and multifactor failure analysis of an important stepped shaft fracture events

Jun Qiu¹, Yaoting Tong, Weiping Ouyang, Xing Chai and Youle Zhao
Shanghai Institute of Special Equipment Inspection and Technical Research, Jinshajiang Road No.915, Shanghai, China

¹E-mail: qiuj@ssei.cn

Abstract. Aiming at a fracture event of an important stepped shaft parts, ABAQUS finite element method was used to simulate and analyze. The results show that the stress concentration is serious at the step where shaft fracture occurs. The maximum stress occurs at the R2 transition corner. The maximum stress value under the most unfavorable condition is 319.9MPa, which exceeds the yield strength limit of the material. Scanning Electron Microscope and X-ray Energy Spectrum analysis shows that fracture of the shaft is a fatigue fracture initiated by multiple sources. Design analysis states clearly that the initiation is mainly related to the small turning radius of the step, which causes the stress concentration factor increase. At the same time, the shoulder height is higher which cause the sudden change of the section size, and will aggravate the stress concentration effect. Furthermore, machining quality analysis indicates that the rough surface with micro-defects due to out of control of metalworking may be the origin of fatigue crack initiation. Heat Treatment process quality analysis indicates that improper control of the quenching and tempering process has a negative impact on the initiation of the crack. Working conditions analysis shows that the actual load state, which exceeds the L2 intermediate level of the design standard load spectrum, accelerating the fatigue damage of mechanical parts.

1. Introduction

Finite element method (FEM) is a common method for design and stress analysis of stepped axles, especially for stress concentration analysis [1-3]. In the application of finite element method, mesh generation is a key step. Researchers need to select the appropriate mesh size according to the purpose of analysis [4-5]. In this paper, the finite element method is applied to the fracture analysis of an important stepped shaft, and other failure analysis methods are synthesized to thoroughly find out the cause of the fracture of the stepped shaft.

A 450t shipbuilding gantry crane in a company, which has been in operation for 3 years, is used in large lifting operation. Due to the breakage of the drum shaft of the lifting mechanism, the lifting objects fall to the ground and the crane is damaged (Figure 1).

The fracture region is located at the step of the stepped shaft. By observing and analyzing extra features of the damaged area, the investigators believed that the macro morphology of the section is fatigue fracture and the final fracture zone is small (about φ40mm, Figure 2).
2. Geometry and loading analysis of shaft

The geometric sketch of the shaft is shown in Figure 3. The fracture occurs at the turning angles of the steps of $\phi 180\text{mm}$ and $\phi 220\text{mm}$.

According to the crane working conditions, the force analysis sketch of the drum structure is drawn as shown in Figure 4, and the bearing support reaction of the drum shaft is solved. The results are shown in Table 1.

| $F_{AX}$ | $F_{AY}$ | $F_{BX}$ | $F_{BY}$ |
|----------|----------|----------|----------|
| 242.1kN  | 682.2kN  | 14.7kN   | 141.3kN  |
3. Finite element simulation

3.1. Model Establishment

The geometric model of the whole drum structure is established in CATIA (as shown at Figure 5). As the fracture of the mechanism occurs on one side of the shaft, it’s unnecessary to applying the finite element analysis for the whole model. The drum part can be replaced by a through shaft (D=240mm) in the finite element software ABAQUS, so that the two ends of the mechanism can be connected together, thus the model can be easily calculated, and the bending moment and the force at the fracture do not change. The simplified model is shown in Figure 6.

3.2. Material properties and constraints

The base material of the shaft is No.35 steel, whose Young's modulus is 212,000MPa, Poisson's ratio is 0.31, yield stress is 315MPa, and ultimate stress is 530MPa. The parameters are input into the material library of the finite element software, and the material properties are assigned to the components. Because the hoisting mechanism uses a self-aligning bearing, the shaft does not bear additional bending moment, so the two fixed ends of the shaft are constrained by XYZ displacement in three directions, one end is constrained by YZ displacement in two directions, and the shaft is not constrained by rotation. The constraint diagram is shown in Figure 7.
3.3. Loading

Through the analysis of the most disadvantageous working conditions of the drum shaft, it can be concluded that the force acting on the drum single rope is X-direction \( p = 210.8 \text{kN} \), and the tangential force, radial force and ring weight are projected to the XZ plane respectively, \( F_{X1} = 281.2 \text{kN} \), \( F_{Z1} = 256.8 \text{kN} \), and the total \( F_{X2} = 120.6 \text{kN} \) of the drum and the wire rope on it. Because of the singularity of stress in local area under concentrated load, the calculation results are deviated. Therefore, the concentrated load is converted into volume force in the model, eliminating unnecessary errors caused by loading. The loading diagram is shown in Figure 8.

3.4. Finite Element Mesh

According to the size of the model and the requirement of calculation accuracy, entity unit and the mesh refinement was employed to the model, and the mesh is divided and refined at the corner transition of R2 (the mesh size is 1.5mm). The mesh is divided and refined as shown in Figure 9 (as the mesh size of different regions in the model varies greatly, tetra shaped elements was chosen).

3.5. Results

After checking correctly, the model is submitted to ABAQUS software for post-processing calculation, and the calculation results are shown in Figure 10.
From Figure 10, it can be seen that the stress concentration occurs at the fracture site of the shaft under rated load. The stress distribution phase diagram of its left end is analyzed. Maximum stress occurs at the transition point of R2 fillet. The maximum stress value is 319.9MPa, which exceeds the yield strength limit of the material Rel=315MPa. In order to stereoscopically reflect the stress state of this section, the section is cut off along R2 rounded cross section, XY plane and YZ plane, and the stress distribution diagram is also shown in Figure 11.

Figure 10. Stress distribution of the shaft and its left end.

Figure 11. Stress distribution diagrams of cross section, xy section and yz section at r2 round corner at axis fracture.
4. Failure analysis

4.1. Design analysis of shaft
The design structure diagram shows that the radius of transition corner \( R = 2 \text{mm} \) at the step shaft \( \phi 180 \text{mm} / \phi 220 \text{mm} \) is less than the national standard value, and it is quite different from the industry standard value. If the radius of transition corner is too small, the stress concentration factor will increase. The shoulder height is higher which cause the sudden change of the section size, and will aggravate the stress concentration effect.

4.2. Quality analysis of heat treatment process
According to the technical requirements of the shaft (35 steel), the hardness after quenching and tempering should reach 170HB-200HB. Brinell hardness (Figure 12) was measured from surface to interior on longitudinal of the shaft at \( \phi 220 \text{mm} \) fracture section. Physical and chemical test results show that the actual near surface hardness (220HB) exceeds the technical requirements, and the effective quenching layer depth is shallow, which has a certain impact on fatigue strength.

![Figure 12. Hardness gradient on longitudinal section of shaft at \( \phi 220 \text{mm} \) fracture zone.](image)

4.3. Quality analysis of metal processing
There are obvious turning marks on the surface of the shaft near the fracture section. Compared with the roughness sample, the surface quality of the shaft is between 3.2 \( \mu \text{m} \) and 6.3 \( \mu \text{m} \), and the roughness of the surface does not meet the design requirement of Ra1.6 \( \mu \text{m} \) (Figure 13). Surface state coefficient \( \beta \) has some influence on fatigue fracture [6-7], and surface micro-defects may be the origin of fatigue crack initiation.

![Figure 13. Comparisons of surface roughness in turning.](image)

4.4. Scanning electron microscope and X-ray energy spectrum analysis
The fatigue fracture of shafting can be clearly confirmed by scanning electron microscopy (SEM) analysis of the fracture surface of shafting. On the outer surface of the shaft, there are rough bamboo-
shaped metal processing marks and transverse stretch-like stripes with nodal spacing distribution; at higher magnification, the morphology shows that the processing cracks open and turn, and extend to the edge of the section, which is related to the beginning of the section; at the edge of the section, there are parallel and dense fatigue stripes (Figure 14). Parallel fatigue growth fringes and intergranular secondary cracks can be seen in the sub-edge region, showing brittle morphology.

![Figure 14. Machining traces of rough surface of axis and sub-layer fatigue glow.](image)

4.5. Working conditions analysis
Statistical data of hoisting mechanism operation show that the actual load state exceeds the L2-intermediate level of the design standard load spectrum, and it is easy to accelerate the fatigue damage of mechanical parts. At the same time, the fatigue fracture surface accounts for about 85% of the area of the cross-section, and the potential safety hazard cannot be found in daily use.

5. Conclusions
(1) According to ABAQUS finite element simulation calculation, the maximum stress of local stress concentration exceeds the yield strength limit of material under the most unfavorable conditions at the transition corner of the shaft's $\phi$180 mm/ $\phi$220 mm step.

(2) In the design of shaft, the selection of transition corner radius at fracture is too small and the selection of shoulder height is too large, which makes the stress concentration factor too high, and has adverse effects on the fatigue life of shaft.

(3) The surface roughness and surface hardness of the fracture zone of the shaft do not meet the technical requirements, which will greatly reduce the toughness and fatigue resistance of the corner zone. The use beyond the design level has a negative impact on the initiation of cracking.

(4) A large proportion of the fatigue growth zone on the cross section indicates that there is a period from crack initiation to final fracture, but the fatigue crack cannot be found in the daily maintenance of the crane during this period, which indicates that there is a blind area in the daily maintenance of the crane.

(5) The fracture of the drum shaft is a fatigue fracture initiated by multiple sources. The initiation of the drum shaft is mainly related to the small transition fillet in the step area and the high stress concentration effect. It is easy to initiate fatigue cracking under impact load, which eventually leads to the overall failure, the drum losing its restraint, and high-speed rotation under the gravity drag of the suspender until the suspender falls to the ground and damaged.

Acknowledgments
This paper is support by The National Key Research and Development Program of China (No.2017YFC0805705).

References
[1] Zhang Jin, Chen Xiangyang and Wang Long 2017 *Journal of Anhui university of science and technology* (natural science) 37(3) 56
[2] Jiang Bao, Zhang Qiang and Cui Wei 2018 *Machine design and manufacturing engineering*
47(7) 11

[3] Liang Keshun 2017 *Machinery* 55(10) 67
[4] Wang Mingqiang, Zhu Yongmen and Lu Wenxin 2004 *Machinery design & manufacture* 1 22
[5] Wang Yu, Lu Ling and Li Wentao 2014 *Hoisting and conveying machinery* 3 52
[6] Cai Yan 2018 *Internal combustion & parts* 21 128
[7] Sun Junlan and Jiang Dazhi 2002 *Journal of yancheng institute of technology (natural science)* 15(4) 21