Finite element analysis of crankshaft for high-speed punch based on ANSYS

Xinzhou Zhang [a], Lan Chen [b,] *, Shuyuan Gan and Yuan Wang

School of Mechanical Engineering, Jiangsu University, Zhenjiang, 212013, China

[a]zhangxinzhou1984@126.com, [b]chenlan5321@126.com (Corresponding author)

Abstract

The crankshaft is a key part of main transmission system for high-speed punch, which has fateful consequence to the performance and reliability of the punch. After analyzing the structure of crankshaft, the finite element analysis model of crankshaft is built and the boundary condition is confirmed in ANSYS. The static analysis of crankshaft under different strokes are carried out. Under different punch strokes, the change of the strength and stiffness of crankshaft in two limit positions, are analyzed. The finite element analysis results provide a basis for the design and optimization of crankshaft for high-speed punch.

Keywords

FEM, crankshaft, high-speed punch, ANSYS.

1. Introduction

The high-speed punch is the fundamental equipment of metal forming process. Because of the characteristics of high efficiency, high quality, low cost and wide range, high-speed punch is employed in the industry of automobile manufacturing, aerospace and chemical metallurgical [1-5].

Fang [6] made comparative analysis between the one-way load and the two-way load of crankshaft, analyzed the stress with continuous beam method of the one-piece crankshaft, and proposed a new type of assembled crankshaft. Yu [7] established an entity model of a car engine and made modal analysis and static analysis of crankshaft. The result provides a reference for design and improvement of the crankshaft. Pu [8] carried out dynamics simulation of crankshaft, obtained the distribution of stress and strain of the crankshaft, and verified the dynamic characteristic. Wang [9] carried out stress analysis of crankshaft based on ANSYS and found the fracture reason of crankshaft. Wang [10] obtained the topology pseudo density curve, the objective function and the constraint relation of the crankshaft, and the concept design model based on the topological structure analysis of the crankshaft. Niu [11] gained the natural frequencies of the crankshaft by static and modal analysis of the crankshaft, analyzed the influence of vibration. Qin [12] made modal analysis using ANSYS, achieved the five natural frequency, and the results showed that the structural strength of the crankshaft is safe and reliable.

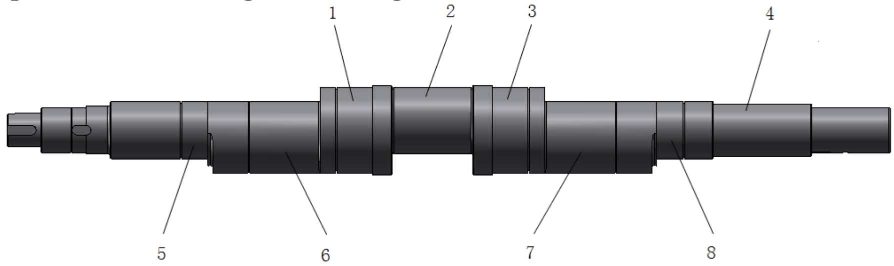
This dissertation mainly carries out a static analysis of crankshaft for high-speed punch under different strokes.

2. Finite Element Analysis modeling of crankshaft

2.1 Structure of crankshaft

The structure sketch of crankshaft is shown as Fig. 1. This type of crankshaft is used for 800KN high-speed punch made by a company. The working conditions of the punch are as follows: Nominal Pressure is 800 KN, Nominal Pressure Stroke is 2 mm, Slider Stroke is 30 mm, number of strokes is 600 SPM (Strokes per Minute), the thickness of punched part is 2 mm, the material of crankshaft is 40Cr.

The crankshaft transfer drive torque from the flywheel, which is installed in location 4 with clutch. The four groups of support bearings, which are installed in location 1, 3, 5, 8, form a revolute pair with the punch frame. The two groups of support bearings, which are installed in location 6, 7, form a revolute pair with main connecting rod. One group of support bearing, which is installed in location 2, form a revolute pair with balancing connecting rod.



1、3、5、8—Installation locations of support bearing

2—Installation location of connecting rod bearing of Balancing slider

Fig. 1 Structure sketch of crankshaft

2.2 Model of crankshaft

The crankshaft, which has complex minute structure, is a key part of main transmission system for high-speed punch. Due to the complicated structure of the crankshaft, some simplifications (ignore all screw holes, small holes and fillets) are adopted in order to make the analysis more convenient. The finite element model of crankshaft is shown in Fig. 2. Solid95 element is used to mesh the model, which is adopted for the complex stress and strain condition. The mesh model of crankshaft is shown in Fig. 3.

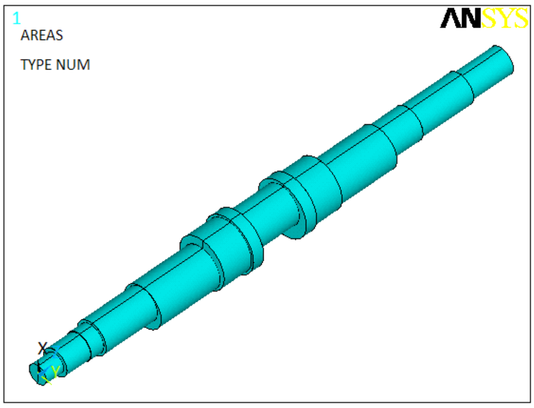


Fig. 2 The finite element model of crankshaft

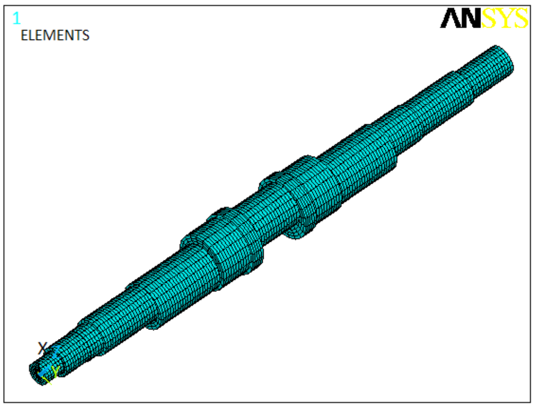


Fig. 3 The mesh model of crankshaft

2.3 Boundary conditions

For the connecting rod under uniaxial tension and uniaxial compression, the distribution of the force acting on connecting rod hole is as follows: the force is dominated by a parabolic rate law along axis direction of hole and is dominated by a cosine law along the circumference of hole in the limited 120 angle range. The diagram of distribution of the force is shown in Fig. 4. The force on the neck journal of crankshaft can be obtained by the interaction of force. The distribution of load equation is expressed as

$$q(x,\theta)=\frac{9P}{8lr}(1-\frac{4x^2}{l^2})\cos(1.5\theta) \quad (1)$$

where P is total load on the neck journal of crankshaft, r is the radius of the neck journal, l is the length along the axis of the connection between the neck journal and connecting rod, $x=-l/2\sim l/2$ is the variable of the length, $\theta=-60^{\circ}\sim 60^{\circ}$ is the variable of angle of the neck journal along the circumference.

The pressure on location 6 and 7 of crankshaft is vertically downward at the top dead center. The pressure on location 2 of crankshaft is vertically upward at the top dead center. But at the bottom dead center, it is just the opposite.

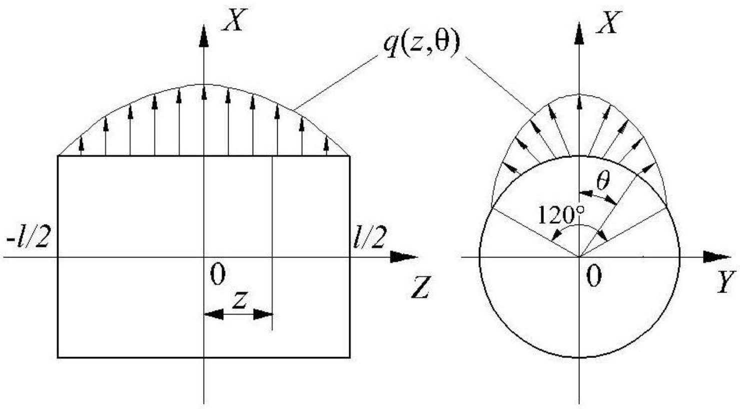


Fig. 4 The distribution of stress of crankshaft in neck journal

3. Results and discussion

The finite element analysis of crankshaft at the top and bottom dead center are carried out. The strength and stiffness of crankshaft under different strokes are obtained by static analysis. The displacement and stress of crankshaft at the top and bottom dead center in vertical direction under 1000 SPM are shown in Fig. 5~ Fig. 8. The data of the analysis of crankshaft in two limit positions (the top dead center and the bottom dead center of slider) are given in Table 1. The change of the maximum displacement and stress of crankshaft in vertical direction under different strokes are shown in Fig. 9 and Fig. 10.

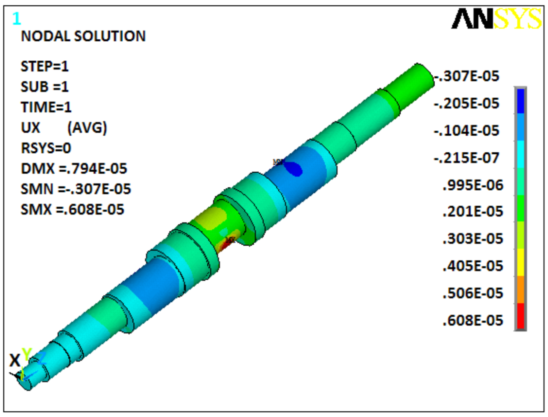


Fig. 5 The displacement of crankshaft at the top dead center in vertical direction under 1000 SPM

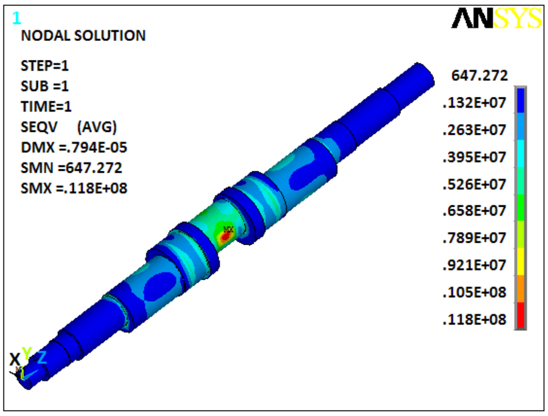


Fig. 6 The stress of crankshaft at the top dead center in vertical direction under 1000 SPM

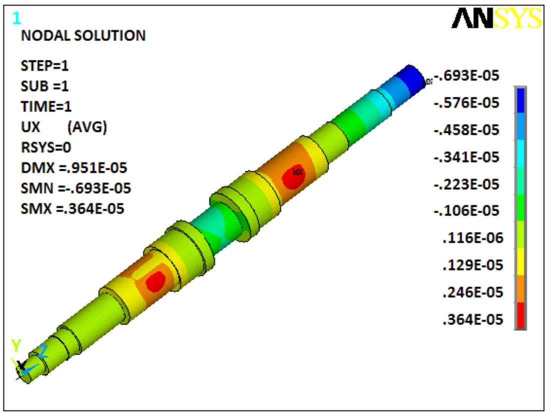


Fig. 7 The displacement of crankshaft at the bottom dead center in vertical direction under 1000 SPM

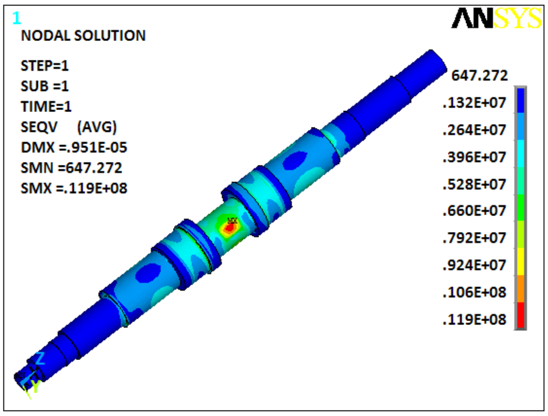


Fig. 8 The stress of crankshaft at the bottom dead center in vertical direction under 1000 SPM

Table 1 The analysis result of crankshaft under different strokes

Number of strokes /SPM	The maximum displacement of crankshaft in vertical direction / μm		The maximum stress of crankshaft / MPa	
	The top dead center location	The bottom dead center location	The top dead center location	The bottom dead center location
300	0.75	0.48	1.52	1.26
400	1.16	0.73	2.32	1.87
500	1.69	1.04	3.34	2.66
600	2.33	1.42	4.58	3.95
700	3.09	2.22	6.06	5.41
800	3.97	2.39	7.76	7.42
900	4.97	2.98	9.68	9.52
1000	6.08	3.64	11.8	11.9

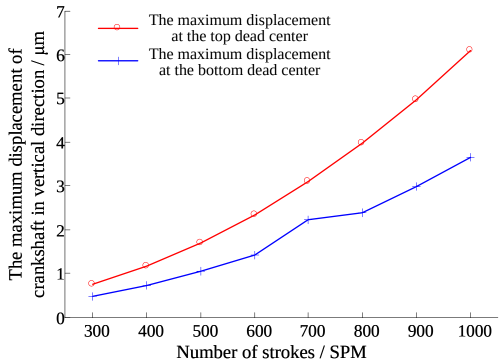


Fig. 9 The change of the maximum displacement of crankshaft in vertical direction under different strokes

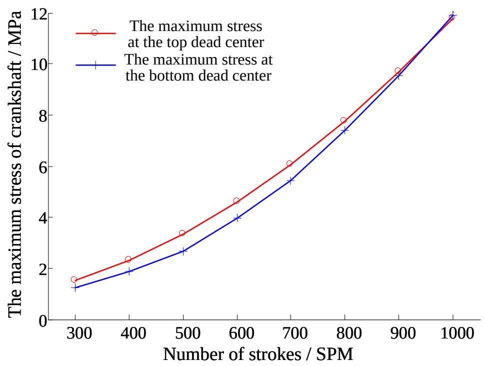


Fig. 10 The change of the maximum stress of crankshaft in vertical direction under different strokes

As can be seen from the chart, the maximum displacement and stress of crankshaft in vertical direction under different strokes are all very small. Under 1000 SPM, the maximum displacement at the top dead center is 6.08 μm, and the value at the bottom dead center is 3.64 μm. The maximum stress at the top dead center is 11.8 Mpa, and the value at the bottom dead center is 11.9 Mpa. The result shows that the strength and stiffness of the crankshaft can well meet the requirements even under 1000 SPM. The maximum displacement and stress of crankshaft increase with the value of the punch stroke.

4. Conclusion

After analyzing the structure of crankshaft for high-speed punch, the finite element analysis model of crankshaft is built in ANSYS. The intensity and rigidity analysis of crankshaft under different strokes (from 300 SPM to 1000 SPM, the incremental change is 100 SPM) are carried out. According to the static analysis results, the crankshaft can well meet the requirements even under 1000 SPM and some optimizations can be made for reducing costs. The finite element analysis results provide a basis for the reliability design and optimization of crankshaft for high-speed punch.

Acknowledgements

This work is financially supported by Fundamental Re-search Project of the National Natural Science Foundation of China (Grant No. 71601086), the National Natural Science Foundation of Jiangsu Province (Grant No. BK20160485), China Postdoctoral Science Foundation (Grant No. 2015M581748), the Natural Science in Colleges of Jiangsu Province of China (Grant No. 15KJB460005 & 16KJB140002), Jiangsu Planned Projects for Postdoctoral Research Funds (Grant No. 1501102C), Advanced Talent Foundation of Jiangsu University of China (Grant No. 14JDG138 & 15JDG033).

References

[1] S.K. Kernosky, K.J. Weinmann, J.R. Michler, et al. Development of a die shoulder force transducer for sheet metal forming research, Journal of Manufacturing Science and Engineering, vol. 120(1988), p.42-48.

[2] J. Breitling, D. Wallace, T. Altan. Investigations of different loading conditions in a high speed mechanical press, Journal of Materials Processing Technology, vol. 59(1996), p.18-23.

[3] M.C. Doolan, S. Kalyanasundaram, P. Hodgson, et al. Identifying variation in sheet metal stamping, Journal of Materials Processing Technology, vol. 115(2001), p.142-146.

[4] R. Hambli, F. Guerin, B. Dumon. Numerical evaluation of the tool wear influence on metal-punching processes, The International Journal of Advanced Manufacturing Technology, vol. 21(2003), p.483-493.

[5] R. Du, W.Z. Guo. The Design of a New Metal Forming Press with Controllable Mechanism, Journal of Mechanical Design, vol. 125(2003), p.582-592.

[6] H.S. Fang. The analysis and structural optimization of the crankshaft based on finite element method, Hangzhou: Zhejiang University of Teehnology, 2009. (In Chinese)

[7] Q.H. Yu, S.M. Liu, Y.H. Liu, et al. Design and Analysis of Crankshaft Based on UG and ANSYS, Journal of Shanghai University of Engineering Science, vol. 27(2013) No. 4, p.302-305. (In Chinese)

[8] H. Pu, X.X. Liu, J. Liu. Dynamic Characteristics Analysis on Crankshaft of L Type Air Compressor Based on SolidWorks and ANSYS, Mechanical Engineer, (2015) No. 12, p.84-86. (In Chinese)

[9] J.Y. Wang, W.C. Ding, H.T. Wang. Finite Element Analysis of the Vehicle Crankshaft Based on Pro /E and ANSYS, Machine Engineering & Automation, (2007) No. 1, p.54-56. (In Chinese)

[10] X. Wang. Finite Element Analysis and Optimization Research on Crankshaft Based on ANSYS, Shenyang: Northeastern University, 2013. (In Chinese)

[11] R.X. Niu, J.Y. Zhan, T.S. Zhong. Static and dynamic analysis of crankshaft in high-speed press based on the FEM, China Metalforming Equipment & Manufacturing Technology, vol. 50(2015) No. 1, p.29-31. (In Chinese)

[12] W.Q. Qin, S.H. Wang, T.F. Li, et al. Finite Element Analysis of Press Crankshaft by ANSYS, Coal Mine Machinery, vol. 32(2011) No. 9, p.98-100. (In Chinese)