ISSN: 1813-4890

Motion Analysis Research of Harmonic Reducer Based on Abaqus

Kun Ni [a], Fa Zhou [b], Ruigang Lu [c], Mengjiao Lv [d]

School of Qingdao University of Technology, Qingdao 266520, China

[a]1626896008@qq.com, [b]2293566131@qq.com, [c]519190901@qq.com,

[d] 1850854998@qq.com

Abstract

the flexible wheel is a thin-walled component that is prone to failure in the harmonic reducer. In order to study the deformation and stress variation characteristics of the flexible wheel in the process of motion, a harmonic reducer is established in this paper based on the equivalent three-dimensional model of the speed reducer, the kinematics of the flexible wheel is analyzed by using ABAQUS, and the shape change and stress distribution law of the flexible wheel in the motion process of the harmonic reducer are obtained. Research. It is found that before the cam rotates, the radial deformation of the flexible wheel is relatively gentle, and the deformation directions of the long shaft and the short shaft are opposite, and the tangential deformation is zero. In the stage of cam rotation, the radial deformation has a period of 0.5 seconds Periodic change of. The stress of the flexible tooth profile and the cup bottom changes in a sawtooth shape. The stress changes periodically with a period of 0.5 seconds in the cam rotation stage, and the maximum stress is large.

Keywords

Flexible wheel; Kinematics; Deformation analysis; Stress distribution.

1. Introduction

Harmonic transmission technology is a new transmission technology developed in the mid-1950s, and because of its simple structure and small volume, high transmission accuracy, large transmission ratio, high load capacity and high transmission efficiency [1-4], after the appearance of harmonic reducer, countries around the world have strengthened the research and production of harmonic reducer. Harmonic transmission has various structure types, such as harmonic helical transmission, harmonic gear transmission, etc., among which, harmonic gear transmission is the most widely used one. Harmonic reducer is composed of 3 parts, mainly including wave generator, flexible wheel and rigid wheel. Among them, the flexible wheel as the main working member, its length determines the length of the harmonic reducer, and its working life also determines the working life of the harmonic reducer. And in actual operation, the flexible wheel is more prone to damage than other parts of the harmonic reducer, so improving the fatigue life of the flexible wheel has been the main problem of the harmonic reducer [5-6].

The failure of the existing long-cylinder type and short-cylinder type flexure wheels mainly occurs in the ring part and the bottom part of the cylinder, which are also the two parts of the flexure wheel with large medium-effect forces, and the maximum stress point of the flexure wheel generally occurs in the part where the ring of the flexure wheel is connected to the cylinder body. The flexure wheel is a flexible body member, and the stress deflection calculation is extremely complicated, and the ratio of the maximum radial deformation to the thickness of the flexible wheel is large, so its stress and deflection cannot be obtained accurately in theory and can only be determined by experiment [7-8].

2. Modeling and pre-processing

2.1. Method of modeling

SolidWorks is a major product of Dassault Systems S.A. It is the world's first Windows-based 3D CAD system that integrates CAD/CAE/CAM functions and has won the world's largest installed base with its simple interface style, easy-to-use operating experience and powerful plug-in modules. The latest version of SolidWorks is also the latest version of SolidWorks. The latest version of SolidWorks software integrates CAD/CAE/CAM functions, enabling industrial products to be operated electronically from the initial design to the whole life cycle of the process, greatly enhancing the efficiency of product development and production.

As the main function of SolidWorks software, its CAD modeling capability is very powerful, which can accurately and quickly model the design of UAVs. For the field of UAV design, SolidWorks CAE plug-ins mainly include Simulation (mechanics), Flow simulation (fluids), etc. Compared with ANSYS, ABAQUS and other old large-scale finite element simulation software, SolidWorks CAE functions are not particularly comprehensive, and the accuracy is not as good as the former, but no tedious However, there is no need for cumbersome setup, high degree of automation, and very fast solution speed, which can greatly improve the efficiency in the initial design of UAV that requires constant iterative verification.

In this paper, SolidWorks is used to build a 3D model to form an assembly for pre-processing interference experimental verification, and its structural performance is verified by joint abaqus simulation.

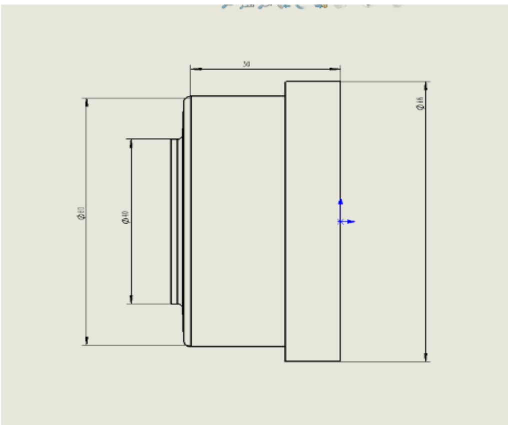

Fig.1. Harmonic reducer model side view

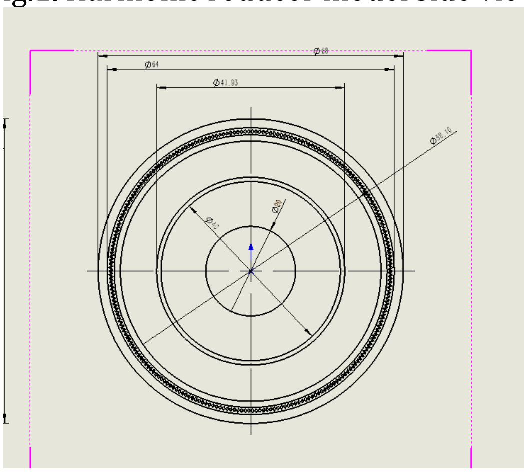

Fig.2. Harmonic reducer model main view

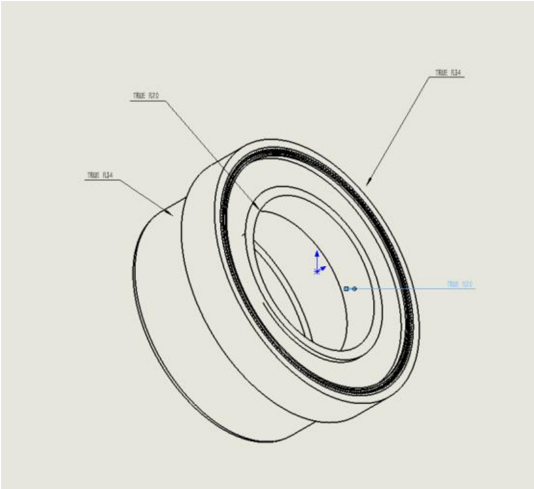

Fig.3. Axonometric view of the left and right diagonal of the harmonic reducer model

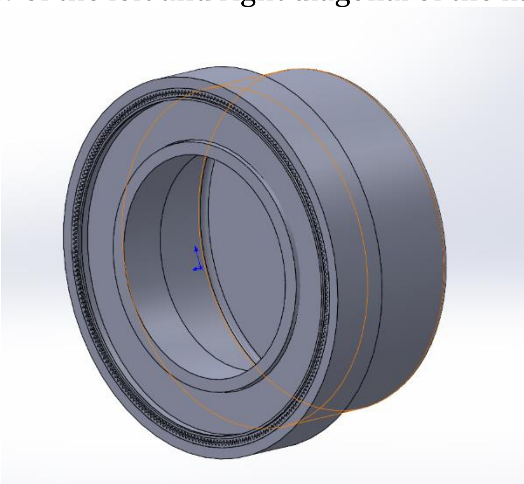

Fig.4. Harmonic reducer model assembly drawing

Figure 1 represents the side view of the harmonic reducer model; Figure 2 represents the main view of the harmonic reducer model; Figure 3 represents the left and right diagonal axonometric view of the harmonic reducer model; and Figure 4 represents the assembly drawing type in the harmonic reducer. The article is all modeled and assembled in SolidWorkers software according to the dimensions of the steel wheel, the flexible wheel and the wave generator.

The dimensions of each part of the harmonic reducer are shown in Table 1.

Table1. Harmonic reducer parameters

Input speed3000rpm	Rated input torque	N·m	1.5
	Output power	rpm	60
	Rated input power	kw	0.012
Input speed1500rpm	Rated input torque	N·m	2
	Output power	rpm	30
	Rated input power	kw	0.010
Input speed1000rpm	Rated input torque	N·m	2.5
	Output power	rpm	16
	Rated input power	kw	0.006

2.2. Harmonic reducer model interference test

According to the gear manual, if the established model does not satisfy one of the following conditions, the model interferes, and then the model cannot be used as the basis for theoretical studies.

The condition that no involute interference is generated is:

$$Z_2 \geq [Z_1^2 \sin^2 \alpha - 4(h_a / m)^2] / [2Z_1 \sin^2 \alpha - 4(h_{a2} / m)] \tag{1}$$

If this condition is not met, then the involute interference is generated, and such interference problem can be solved by increasing the displacement factor of the gear.

The conditions that do not produce superimposed interference of tooth profiles are:

$$\delta_1 = \cos^{-1}[(r_{a2}^2 - r_{a1}^2 - a^{r2}) / (2r_{a1}a^{'})] \tag{2}$$

$$\delta_2 = \cos^{-1}[(a^{r2} + r_{a2}^2 - r_{a1}^2) / (2r_{a2}a^{'})] \tag{3}$$

There are two cases that do not produce excessive curve interference.

The first one is the condition of not producing excessive curve interference as:

$$(Z_2 - Z_1)\tan\alpha^{'} + Z_1 \tan\alpha_{a1} \leq (Z_2 - Z_1)\tan\alpha_{02}^{'} + Z_{02}\tan\alpha_{a02} \tag{4}$$

The second condition that does not produce tooth root curve interference is:

$$Z_2 \tan\alpha_{a2} - (Z_2 - Z_1)\tan\alpha^{'} \geq (Z_1 + Z_{01})\tan\alpha_{01}^{'} - Z_{01}\tan\alpha_{a01} \tag{5}$$

Where, Z_{01}, Z_{02} are the number of teeth of the tooth cutter for gear processing; α_{01}', α_{02}' are engagement angle when machining gears; α_{a01}, α_{a02} are pressure angle when machining gears.

If one of the conditions is not met, excessive curve interference occurs, which can be reduced by increasing the coefficient of variation of the rigid wheel or reducing the top height of the teeth.

The condition of no radial interference is:

$$Z_2 - Z_1 \geq [2(h_{a1} + h_{a2})] / (m \sin^2 \delta) \tag{6}$$

$$(2\delta - \sin 2\delta) / (1 - \cos 2\delta) = \tan\alpha \tag{7}$$

If this condition is not met, radial interference occurs, and the same can be solved by increasing the tooth profile angle and reducing the top height of the tooth.

3. Analysis of finite element model

3.1. Finite element modeling

The results of the parametric 3D modeling of the long cylindrical flexure wheel and the new flexure wheel into the FEA software in SolidWorks are shown in Figure 5. Firstly, a gear with m=0. 5 and z=200 teeth was imported from Toolbox tool library, and then the flex wheel was built by using tools such as stretching and cutting. After that, the wall thickness T, the radius $r3$ and the width b of the gear ring are set as variable parameters.

3.2. Finite element model meshing

Since the quality of the mesh after the structure is discretized directly affects the solution time and the correctness of the solution results, the software developers have increased their investment in mesh processing, so that the quality and efficiency of mesh generation have been greatly improved. For tetrahedral cells, if intermediate nodes are not used, incorrect results will be generated in many problems, and if intermediate nodes are used, it will cause a series of problems in terms of solution time and convergence speed, so people are eager to see the emergence of automatic hexahedral meshing function. Adaptive meshing is a cyclic process of estimating computational errors, re-meshing and re-computing based on the existing mesh, based on the finite element calculation results. For many practical engineering problems, some regions of the model will generate large strains throughout the solution process, causing cell distortion, which leads to the solution cannot proceed or the solution result is incorrect, so automatic mesh re-gridding must be performed.

A parametric 3D model was established using the association between SolidWorks and Abaqus software, and the long cylindrical flexure wheel cylinder (semi-circular structure of the new flexure wheel), the flexure wheel ring and the bottom part of the flexure wheel cylinder were divided with a swept mesh using Solid186 hexahedral cells. Since the bottom table of the flexure wheel is bolted to the output end, the bottom flange of the flexure wheel is meshed with free meshing as shown in Figure 6.

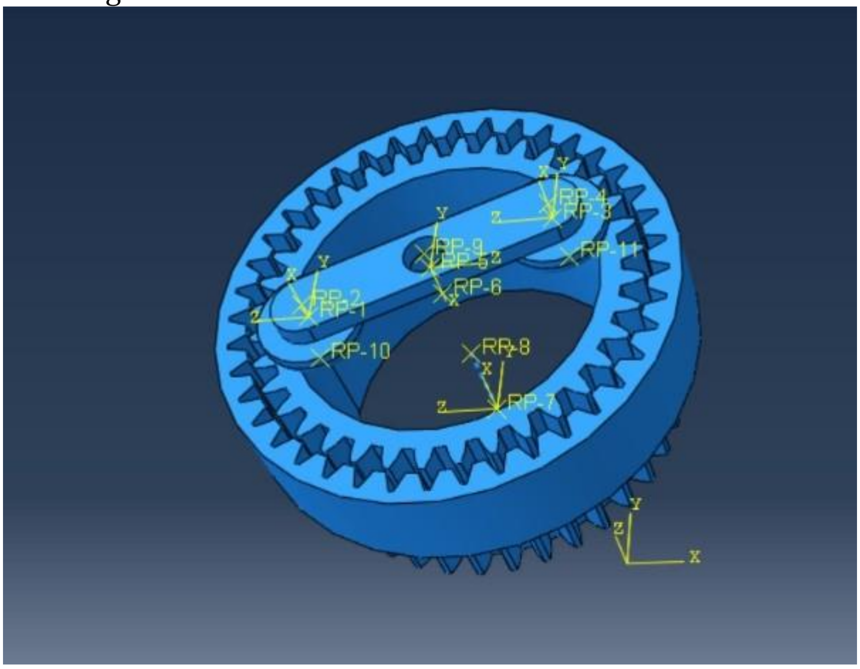

Fig.5. Harmonic reducer finite element model

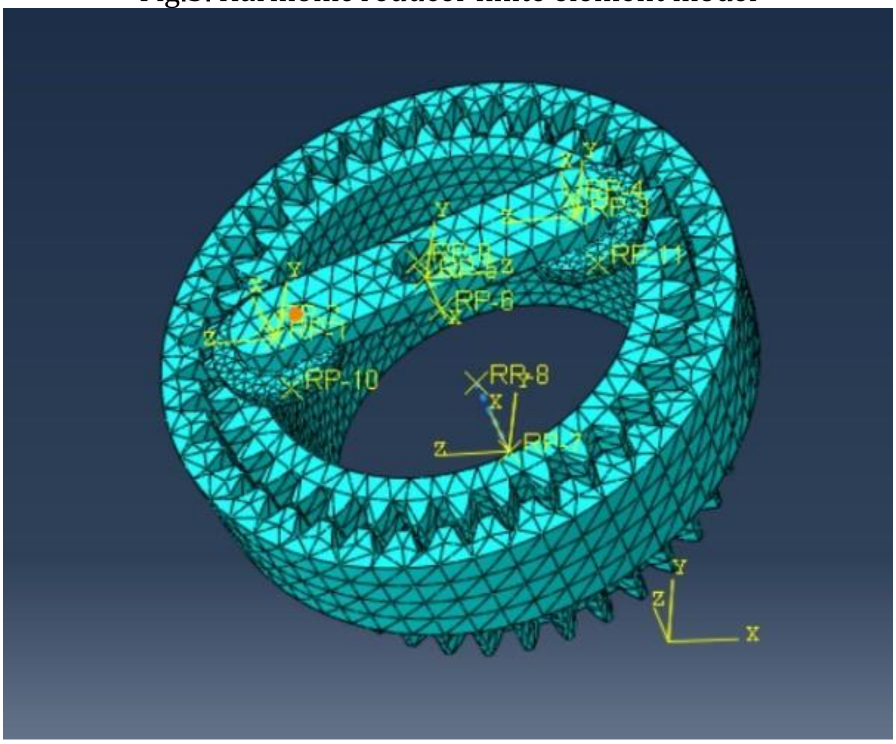

Fig.6. The harmonic reducer model after dividing the mesh

3.3. Add contact and loading to the model

The flexible wheel of the harmonic reducer belongs to a flexible body, while the wave generator, which is in indirect contact with the flexible wheel, can be defined as a rigid body, so the contact problem of the flexible wheel-wave generator can be considered as a rigid-flexible body contact problem, defining the inner ring of the tooth ring part of the flexible wheel as the contact surface, the outer ring of the wave generator as the target surface, and the contact mode as face-to-face contact . The torque of the new flexible wheel is 100·m. as shown in Figure 7.

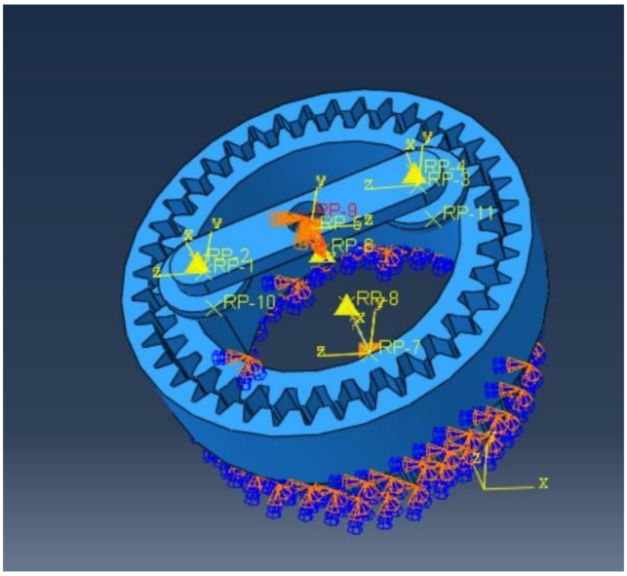

Fig.7. Model after application of load and contact

3.4. Analysis of results

The nodes are selected at the maximum model stress of the whole machine to observe the stress changes in the operating range, and to observe whether the harmonic reducer can operate smoothly under normal operating conditions, which is conducive to better improving the reduction efficiency of the reducer. It is shown in Figure 8 below.

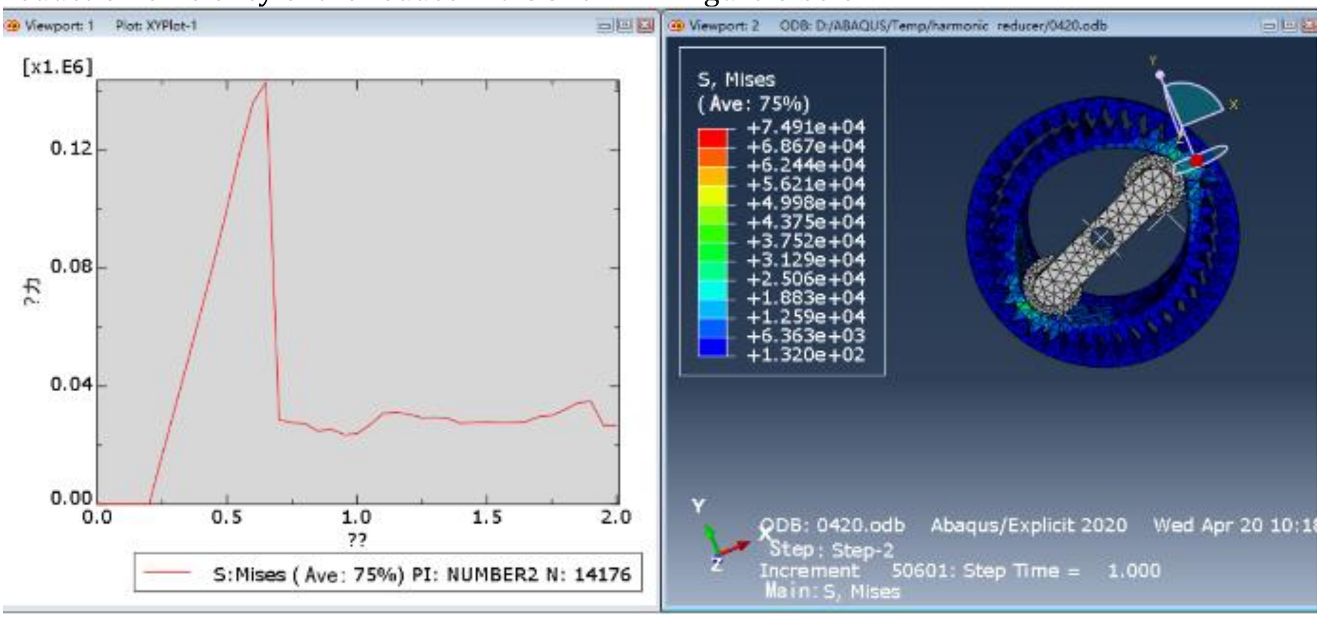

Fig.8. Stress variation diagram of the model

We set the wall thickness T=0. 5 mm, the radius of arc $r3$=7. 5 mm, and the other parameters remain unchanged, and the analysis range of the ring width b is set to 12~17 mm. In Fig. 8, the maximum stress value $\sigma 2$=466. 44 MPa for the new type of flexure wheel with b=16 mm and

$\sigma 2$=472. 56 MPa for the new type of flexure wheel with b=16 mm. Fig. 8 shows the fitted curve of the maximum stress value. When the width of the ring is larger than 14 mm, the stress value at the position of the ring tends to level off.

Of course, factors such as the radius of the fillet and the length of the output end of the flexure wheel have effects on the flexure wheel, but their effects are very small and are not elaborated here. The simulation analysis of different parameters of the new type of flexure wheel on the maximum stress of the flexure wheel can show that the radius of the new type of flexure wheel and the wall thickness have a great influence on the maximum stress of the flexure wheel, and the width of the tooth ring has a relatively large influence on the maximum stress of the flexure wheel. Among them, the most influential one is the radius of the new type of flexible wheel. When the radius of the circle is gradually increased from 5 mm to 10 mm, the stress change of the flexure wheel is relatively large. The larger the radius of the circle at the bottom of the new type of flexure wheel, the smaller the overall stress value of the flexure wheel and the smaller the maximum stress value. However, when the radius of the arc increases, the new type of flexure wheel will become longer, which will increase the overall aspect ratio of the new type of flexure wheel, resulting in the overall size of the harmonic reducer becoming larger, which is not in line with the expected design, so the size of the arc should not be too large. When the wall thickness of the flexure wheel gradually increases from 0. 2 mm to 0. 6 mm, the stress of the flexure wheel first drops sharply, then rises and finally rises slowly. The stress is relatively low until the wall thickness is 0. 25 mm to 0. 3 mm. Due to the problem of existing manufacturing process, the wall thickness of the flex wheel should not be too thin, so the wall thickness of the flex wheel can be reduced appropriately when the conditions allow.

4. Conclusion

(1) The flexure structure of the existing long-cylinder type flexure wheel and short-cylinder type flexure wheel is optimized, and the original right-angle cylinder wall is replaced by the semi-circular transition structure of the existing new type of flexure wheel. The new type of flexure greatly reduces the length-to-diameter ratio of the flexure, which is about 0.24 times of the original long-cylinder type, and the stress value of the tooth ring position of the new type of flexure, i.e., the maximum stress value of the flexure, is reduced by 5% to 10%.

(2) The overall stress value position of the new type of flexible wheel has been changed. The maximum stress value of the previous long cylindrical type of flexible wheel occurs at the connection between the tooth ring and the cylinder body, followed by the transition arc at the bottom of the cylinder. This structure greatly improves the meshing efficiency of the teeth of the flexible wheel and the rigid wheel, reduces the energy lost from the heat of the teeth of the flexible wheel, and improves the efficiency of the harmonic reducer.

Acknowledgements

The authors declared no potential conflicts of interest with respect to the research, authorship, and/or publication of this article. The authors thank Qingdao University of Technology and Shandong Jiao tong University for the support and contribution.

Funding: This work was supported by the National Natural Science Foundation of China (grant number 52202508); the Key Lab of Industrial Fluid Energy Conservation and Pollution Control, Ministry of Education (grant number 2022-JXGCKFKT-YB-04); the Key Laboratory of Transportation Industry for Transport Vehicle Detection, Diagnosis and Maintenance Technology (grant number JTZL2202).

References

[1] ZHANG Liyong, LIU Xinmeng, WANG Changlu, et al. Analysis of the effect of radial deformation on the meshing characteristics and flexure wheel stress of harmonic reducer [J]. Mechanical Transmission, 2017, 41(9): 166-169.

[2] Shen Yunwen . Theory and design of harmonic gear transmission [M]. Beijing: Machinery Industry Press, 1985: 21-40.

[3] M H Ivanov. Harmonic gear transmission [M]. Shen Yunwen, Li Kemei, translation. Beijing: National Defense Industry Press, 1987: 129-151.

[4] FAN Ming-Long, GU Ya-Wen. The effect of radial deformation coefficient on the meshing characteristics of harmonic gear transmission [J]. Mechanical Transmission, 2016, 4(2): 116-120.

[5] Dong Huimin, Li Deju, Qi Shuxue. Structural optimization of flexure wheel cups for harmonic transmission based on orthogonal test and finite element analysis [J]. Mechanical Transmission, 2013, 37(1): 34-38.

[6] Xia Tian, Yang Shiyong, Wang Yafeng . Fatigue analysis of harmonic reducer flexible wheel [J]. Mechanical Transmission, 2017, 41(10): 133-135.

[7] Liang Y, Fan Yuanxun . Analysis of deflection phenomenon of harmonic drive flexible wheel teeth and its influence [J]. Mechanical Transmission, 2018, 42(2): 31-33.

[8] Fu Junfeng, Dong Haiyang, Shen Yunwen. Finite element analysis of flexure wheel stress in harmonic gear transmission [J]. China Mechanical Engineering, 2007, 18(18): 2210-2214.

[9] Gao HB, Li CG, Deng ZONGQuan . Sensitivity analysis of the structural parameters of cup-shaped flexible wheel to the stress of flexible wheel based on ANSYS [J]. Journal of Mechanical Engineering, 2010, 46(5): 1-7.

[10] Xie Hengpeng . Analysis of mechanical characteristics of harmonic reducer flexure wheel [D]. Chongqing: Chongqing University, 2015: 19-30.