JJMIE

Volume 20, Number 2, Special issue: Instrumentation & Measurements in Mechanical Engineering, April 2026
ISSN 1995-6665
Pages 127– 135
https://doi.org/10.59038/jjmie/200201
Editor-in-Chief: Prof. Faris M. AL-Oqla

Jordan Journal of Mechanical and Industrial Engineering

Design of Flexible Wheel for Harmonic Drive Based on Orthogonal Simulation Experiment

Jian Li[1,2], Yu Sun[3,*], Shuang Yu[1], Jing Chu[2], Xuan Yan[2], Lijun Yang[2], Ying Wang[1]

[1]Suzhou Vocational Institute of Industrial Technology, Institute of Intelligent Equipment, Suzhou 215104, China
[2]Leader Harmonious Drive Systems Co., Ltd, Suzhou 215101, China
[3]Geophysical Exploration Academy of China Metallurgical Geology Bureau, Baoding 071051, China

Received 15 Sep 2025 *Accepted 12 Feb 2026*

Abstract

Background: Taking the flexible wheel of the 17-series harmonic reducer of Leader Harmonious Drive Systems Co., Ltd with the designation LCD as the research object. Analyze the key design dimensions. Select the important dimensions as variables. And the maximum stress under rated load is taken as the optimization objective to achieve the maximum bearing capacity of the flexible wheel under rated load.

Methods: Firstly, the flexible wheel model is established in Workbench software, and the important dimension parameters are set as variables. Then according to the number of variables, the orthogonal test table of finite element simulation is designed, and the orthogonal simulation test is carried out by workbench. Finally extract the results of each simulation orthogonal experiment and analyze the influence of each variable on the flexible wheel stress.

Results: Obtain the best parameter matching through the analysis of the orthogonal experimental results, and summarize the optimal design method of the flexible wheel of harmonic reducer.

Conclusions: The finite element simulation with orthogonal test method is an effective means to optimize the design of the parameters of flexible wheel.

Keywords: Harmonic reducer, Flexible wheel, Orthogonal test, Finite element method, Optimal design.

1. Introduction

Harmonic gear reducers are widely used in the rotary joints of collaborative robots and humanoid robots, and are one of the core components of robots. Harmonic reducer is mainly composed of rigid wheel, and flexible wheel, and wave generator. Among them, flexible wheel is the key part of harmonic reducer, which transmits motion and power through the meshing of flexible wheel and rigid wheel. The flexible wheel is a thin-walled structure, with the thinnest part having a thickness of less than 1mm, making it prone to malfunction. When the flexspline fails, the robot joint will be unable to rotate normally, such as the joint getting stuck and unable to rotate or rotating uncontrollably. At the moment of joint failure due to jamming, a severe impact may occur, potentially causing damage to the entire machine. When the joint rotates uncontrollably, the end load will fall uncontrollably, causing impact and even harming the safety of the workers collaborating with the robot. Ensuring the reliability of the flexspline during the design process is crucial for the safe use of the robot. In the transmission process under rated load, reducing the maximum stress of the flexible wheel is one of the effective means to improve the service life of the flexible wheel [1-15].

Flexible wheels have many important dimensional parameters, and the maximum stress is determined by the coupling effect of each parameter when the load is given, and we want the value to be as small as possible. It is very difficult to obtain the minimum value of the maximum stress of the flexible wheel when driving under the rated load by changing a certain parameter alone.

In addition, the flexible wheel material is special. Its processing technology is complex, and its processing needs special processing equipment. So the production efficiency of flexible wheel is low. The cost is very high when the flexible wheel parameters are directly optimized through product iteration.

Therefore, in the process of flexible wheel optimization, there are many difficulties such as multi parameter coupling effect, high cost and low efficiency of product iterative optimization.

The 17-series harmonic reducer is one of products of Leader Harmonious Drive Systems Co., Ltd with the designation LCD and the pitch circle of the gear is 17in and named LCD17 by the company. The LCD17 harmonic reducer has flat shape. So it is particularly suitable for being arranged at the shoulder joint of a humanoid robot. LCD is

* Corresponding author e-mail: Ronaldobd0312@163.com.

the code name for the company's ultra-thin series harmonic reducers. 17 refers to the imperial size specification of the flexspline tooth profile pitch circle.

In this paper, the flexible wheel of LCD 17 harmonic reducer is taken as the research object and is studied from the point of view of statics. The relationship between the important dimension parameters of the flexible wheel and the maximum stress under rated load is studied by means of orthogonal test, and the mathematical model of the maximum stress of the flexible wheel is established, and the dimension parameters of the flexible wheel is determined.

2. LITERATURE REVIEW

Many experts and scholars have conducted research on harmonic gear transmission. Ivanov and Shen [1] and Shen and Ye [2] studied the basic principles, mathematical models, and design methods of harmonic reducers. He et al. analyzed optimization test design method [3] and this paper presents the orthogonal experimental method. Chai Wenjie studied deformation Characteristics of flexible wheel and the finite element simulation analysis of the flexible wheel model [3]. Gao et al. [5] and Zhang et al. [6] analyzed flexible gear parameters to its stress based on ANSYS and the reliability of the finite element analysis for harmonic reducers was verified. Ma analyzed the deformation Function of the flexspline and established mathematical equations for flexible wheel deformation [7]. Fu et al. analyzed the stress of flexspline and obtained the influence trend of flexible wheel size parameters on stress [8]. Dong et al. researched structural optimization of the flex spline cup proved the reliability of the orthogonal experimental method [9]. Wang et al. [10], Liu et al. [11] and Deng et al. [12] analyzed structural parameter of flexspline. Zhang et al. analyzed the buckling of flexspline structure [13]. Yang studied the short flexspline and obtained the design method for segment size flexible wheels [14]. Wang et al. researched the structure optimization design of the flexspline that proved the reliability of the orthogonal experimental method [15].

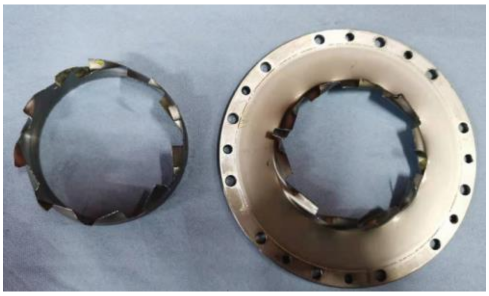
3. THEORETICAL ANALYSIS OF FLEXIBLE WHEEL STRENGTH

LCD 17 harmonic reducer is shown in Figure 1. As shown in Figure 2(a) and Figure 2(b), the common failure parts are the root of the gear tooth and the bottom of the cup body of flexible wheel. During the experiment, the harmonic reducer was installed on the frame and torque was applied to the output end to test whether the flexible wheel would fail under a given torque.

The schematic diagram of the connection structure of LCD17 flexible wheel is shown in Figure 3. The circular shape of A represents the installation surface of the flexible wheel. The shape of flexible wheel assembled with the wave generator is B, and the shape of B is determined by the shape of the wave generator [1, 2].

Figure 1. Harmonic reducer

(a) Picture of tear at the bottom of flexible wheel cup



(b) Picture of tear at the root of flexible gear teeth

Figure 2. Failure diagram of flexible wheel

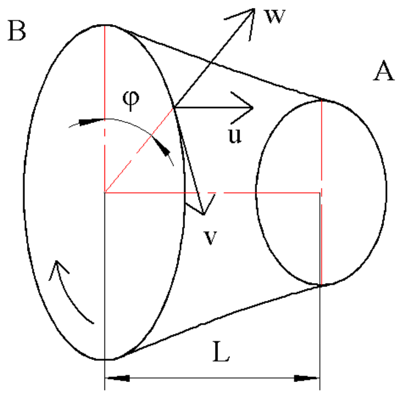


Figure 3. Diagram of LCD 17 flexible wheel

The wave generator limits the radial displacement w, but there are circumferential displacement v and axial displacement u, as well as the rotation of the generatrix of the generator. Under the assumption that the generatrix is not distorted and the assumption of semi moment free theory, the formula is obtained by Ivanov and Shen [1]:

$$v = -\int w \, d\varphi \tag{1}$$

$$u = -r\int \partial v/\partial x \, d\varphi \tag{2}$$

When the function *w* is known, the functions of *v* and *u* can be determined according to formula (1) and formula (2).

According to the condition of generatrix straightness, the function w changes linearly along X, as shown in equation (3).

$$w = \Phi(\varphi)x/L \tag{3}$$

When x = 1, the position of wave generator Φ(φ) Determined by the wave generator.

The axial strain equation in the layer with the distance *z* from the middle surface of the shell is as shown in (4).

$$\varepsilon_{xz} = \partial u/\partial x + z\partial^2 w/\partial x^2 \tag{4}$$

Circumferential strain of this layer is as shown in (5).

$$\varepsilon_{tz} = (1/r)(\partial u/\partial\varphi + w) - (z/r^2)(\partial v/\partial\varphi - \partial w^2/\partial\varphi^2) \tag{5}$$

Angular strain of this layer is as shown in (6).

$$\gamma_{xtz} = (\partial v/\partial x + \partial u/(r\partial\varphi) + 2(z/r)\partial^2 w/(\partial x\partial\varphi) \tag{6}$$

The stress of the layer z from the middle plane of the shell is shown in (7) [1].

$$\begin{cases} \sigma_{xz} = [E/(1-\mu^2)](\varepsilon_{xz} + \mu\varepsilon_{tz}) \\ \sigma_{tz} = [E/(1-\mu^2)](\varepsilon_{tz} + \mu\varepsilon_{xz}) \\ \sigma_{xz} = \{E/[2(1+\mu)]\}\gamma_{xtz} \end{cases} \tag{7}$$

For the stress of the cylindrical shell, according to the assumption of the semi moment free theory, the displacements W and V are functions of the first power of x, and the displacements u and x are independent.

Therefore, $\partial u/\partial x=0$ and $\partial^2 w/\partial x^2=0$. According to the condition of axial non elongation, obtained $\varepsilon_{xz}=0$。

When μ=0, it can be seen from formula (7) that $\sigma_x=0$. When z=±s/2, obtained (8) and (9).

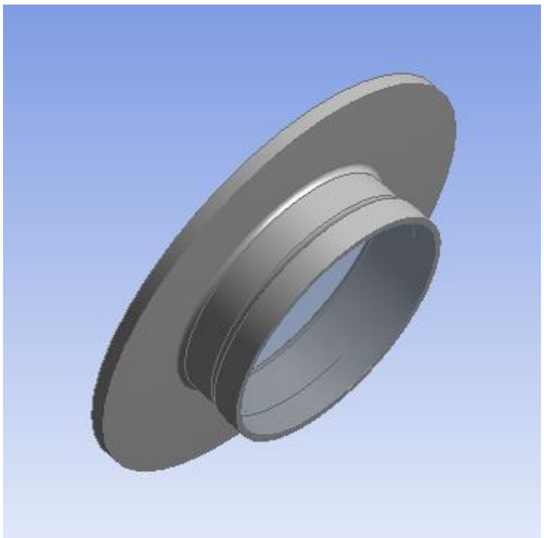
$$\sigma_t = \pm[E_s/(2r^2)](\partial^2\omega/\partial\varphi^2 + \omega) \tag{8}$$

$$\tau_{xt} = \pm[E_s/(2r)]\partial^2\omega/(\partial x\partial\varphi) \tag{9}$$

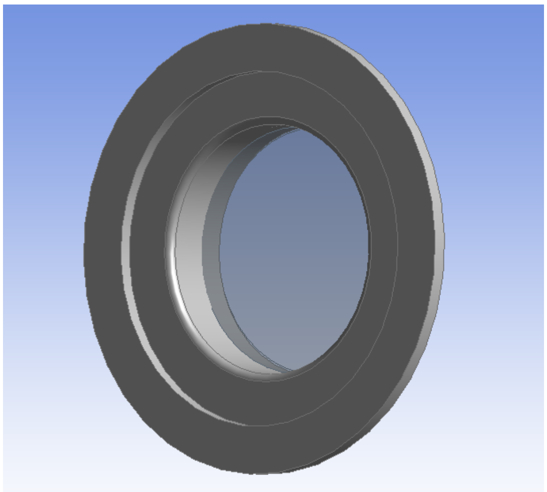
4. FINITE ELEMENT ANALYSIS OF FLEXIBLE WHEEL

4.1. Modeling

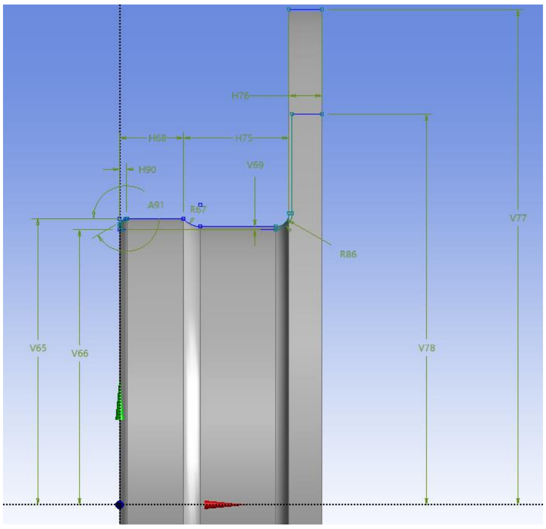
The flexible wheel model of LCD17 harmonic reducer is constructed as shown in Figure 4 (a). The construction of the flexible wheel model of LCD17 is realized by the rotation of the sketch. And the Back of LCD 17 flexible wheel model is shown in Figure 4 (b) The sketch is shown in Figure 4(c), and the meaning of each parameter is shown in Table 1. The wave generator is shown in Figure 5(a). The generator cam and flexible bearing are regarded as a whole and participate in the simulation as a rigid body. The simplified wave generator model is shown in Figure 5(b).



(a) LCD 17 Flexible Wheel Model



(b) Back of LCD 17 Flexible Wheel Model



(c) LCD 17 Flexible Wheel Model Sketch

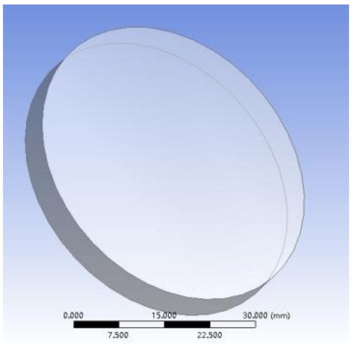
Figure 4. Structural drawing of flexible wheel

Analyze the flexible wheel size (Figure 6):

1. The final tooth shape will not change easily and will not be considered in the simulation of this project.
2. According to the actual experience and simulation test, it can be determined that the outer diameter and inner diameter of the fixed surface of the flexible wheel have little influence on the performance of the flexible wheel.
3. In Figure 4 (b), the dimension R86 is equal to R87, and the dimension H69 is equal to v89.

According to the above analysis, the flexible wheel parameters to be analyzed are simplified. And the important parameters of the flexible wheel are determined as shown in Table 2. These dimensions are set as parameters in the workbench model, as shown in Table 2. The important parameters are set as variable during modeling as shown in Figure 7 (a), and they can be seen in Figure 7 (b). The equivalent stress maximum, total deformation maximum, and equivalent elastic strain maximum are set as output parameters as shown in Figure 7 (b). The project scheme is shown in Figure 7 (c).

(a) Wave Generator



(b) LCD 17 Wave Generator Model

Figure 5. Simplified model of wave generator

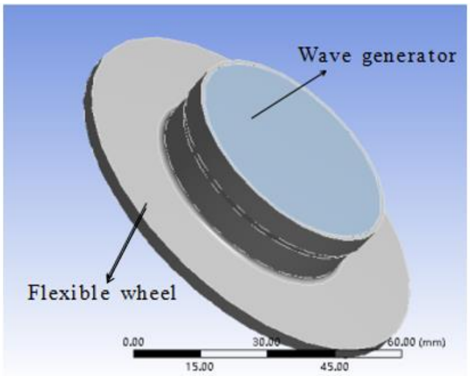


Figure 6. Assembly model of flexible wheel and wave generator

Table 1. Flexible wheel dimension parameters

Root fillet radius	Tooth height	Hollow hole	Tooth thickness	Cup height	Fillet radius of cup root		Thickness of flexible wheel cup wall	Thickness of flexible wheel cup bottom	Thickness of fixed surface	Outer diameter of fixed surface	Inner diameter of fixed surface
R67	V65	V66	H68	H75	R86	R87	H69	V89	V77	V77	V78

Table 2. Important parameters of flexible wheel

Root fillet radius	Cup height	Fillet radius of cup root	Thickness of flexible wheel cup wall	Thickness of flexible wheel cup bottom
R67	H75	R86/R87	H89/V69	V69

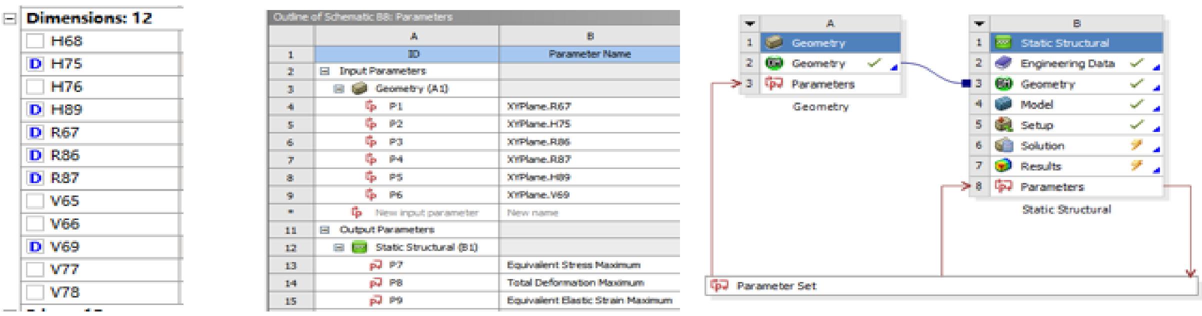


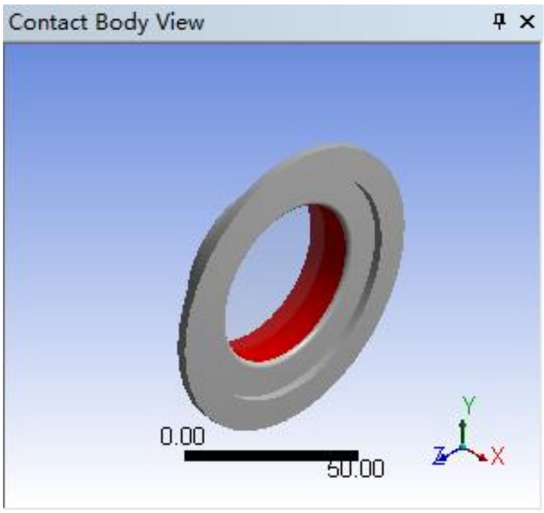
Figure 7. Parametric design of flexible wheel

4.2. Simulation settings

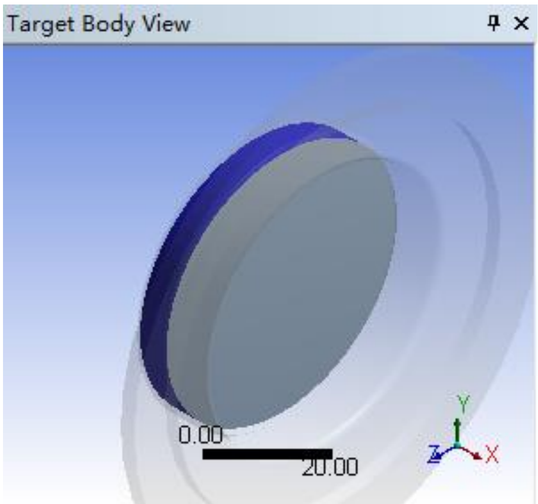
Material steel 45# was selected for analysis. The hollow hole of the flexible wheel and the outer surface of the wave generator form contact, and the hollow hole of the flexible wheel is set as the contact surface as shown in Figure 8(a), and the outer surface of the wave generator is set as the target surface as shown in Figure 8(b). The motion pair of the wave generator is set to be fixed, as shown in Figure 9. The applied torque is shown in Figure 10. The mesh is shown in Figure 11. In the mesh generation settings, the element size is set to 0.003mm, with a medium level of smoothness. There are a total of 23,252 nodes and 11,429 elements

4.3. Analysis of preliminary simulation results

The preliminary test simulation is conducted, and the stress cloud diagram is shown in Figure 12. The flexible wheel of harmonic reducer mainly fails at the place where the gear ring contacts the thin wall of the flexible wheel and the place where the thin wall of the flexible wheel contacts the bottom of the cup.



(a) Contact surface



(b) Target surface

Figure 8. Contact setting

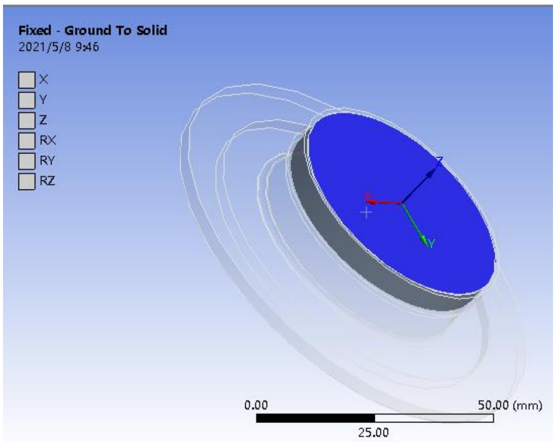


Figure 9. Kinematic pair setting of wave generator

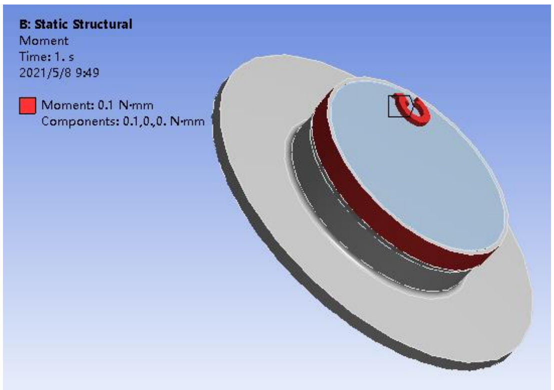


Figure 10. Kinematic pair setting of wave generator

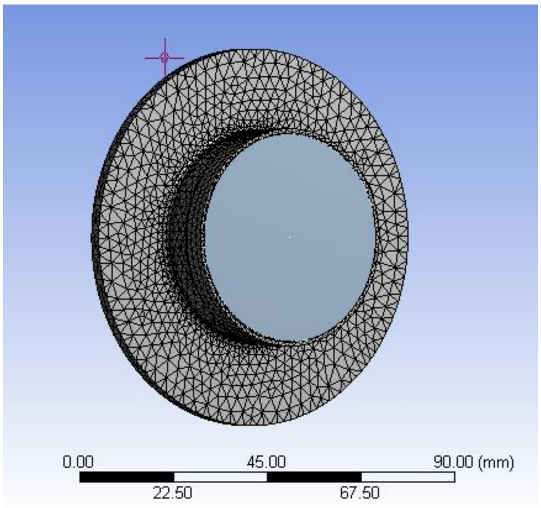


Figure 11. Meshing

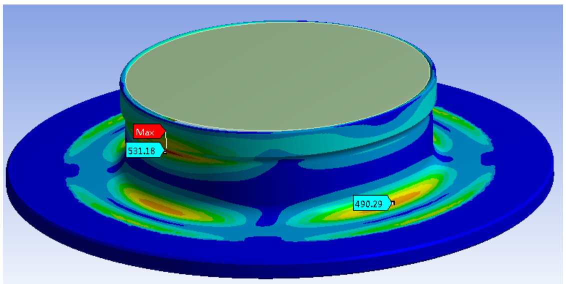


Figure 12. Stress contour diagram

5. ORTHOGONAL TEST SCHEME DESIGN

According to Table 2, six parameters need to be optimized. Therefore, the selection of five level tables is shown in Table 3 [3], which requires 25 simulations. Bring the data in Table 4 into Table 3, which is the parameter matching of 25 simulation tests.

6. ORTHOGONAL SIMULATION TEST RESULTS

The results and analysis of orthogonal simulation test are shown in Table 5. In Table 5, the intersection of each factor and Ki represents the sum of simulation results corresponding to its level i. And ki equal to the average value of Ki

In order to intuitively express the relationship between each parameter and the maximum stress, a broken line diagram of each index parameter k value and the maximum stress is established, as shown in Figure 13. The following conclusions are drawn:

1. Observing Figure 13(a), it is found that the root fillet and the maximum stress have a nonlinear relationship.
2. Looking at Figure 13(b), it is found that the cup height and the maximum stress approximately show an inverse linear relationship.
3. By observing Figure 13(c), it is found that the fillet radius at the root of the cup body and the maximum stress generally show an inverse linear relationship.
4. It is found from Figure 13(d) that the greater the thickness of the flexible wheel cup, the greater the maximum stress.

In general, the influence of the above four parameters on the maximum stress is sorted from the largest to the smallest is cup height, and flexible wheel cup thickness, and cup root fillet radius, and tooth root fillet radius.

Table 3. Five Level $L_{25}(5^6)$

Test No. / Serial number	1	2	3	4	5	6
1	1	1	1	1	1	1
2	1	2	2	2	2	2
3	1	3	3	3	3	3
4	1	4	4	4	4	4
5	1	5	5	5	5	5
6	2	1	2	3	4	5
7	2	2	3	4	5	1
8	2	3	4	5	1	2
9	2	4	5	1	2	3
10	2	5	1	2	3	4
11	3	1	3	5	2	4
12	3	2	4	1	3	5
13	3	3	5	2	4	1
14	3	4	1	3	5	2
15	3	5	2	4	1	3
16	4	1	4	2	5	3
17	4	2	5	3	1	4
18	4	2	1	4	2	5
19	4	4	2	5	3	1
20	4	5	3	1	4	2
21	5	1	5	4	3	2
22	5	2	1	5	4	3
23	5	3	2	1	5	4
24	5	4	3	2	1	5
25	5	5	4	3	2	1

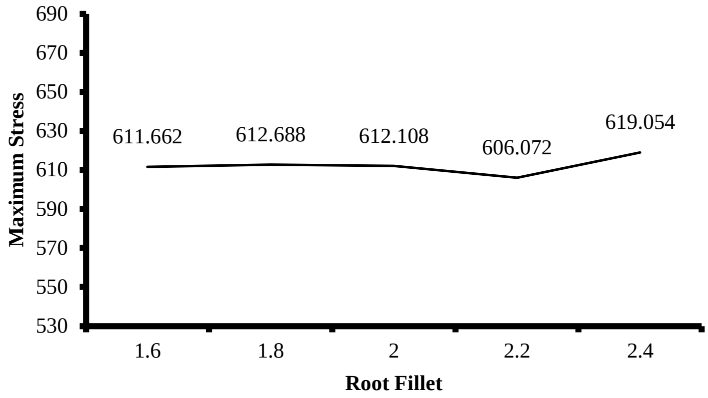
Table 4. Test parameter setting

Parameter / Serial number	Root fillet radius	Cup height	Fillet radius of cup root	Thickness of flexible wheel cup wall
	R67	H75	R86/R87	H69/V69
Item 1	A1	B1	C1	E1
step	0.2	0.6	0.2	0.01
Item i	Ai=A1+(i-1) *0.2	Bi=B1+(i-1) *0.6	Ci=C1+(i-1) *0.2	Ei=E1+(i-1) *0.01

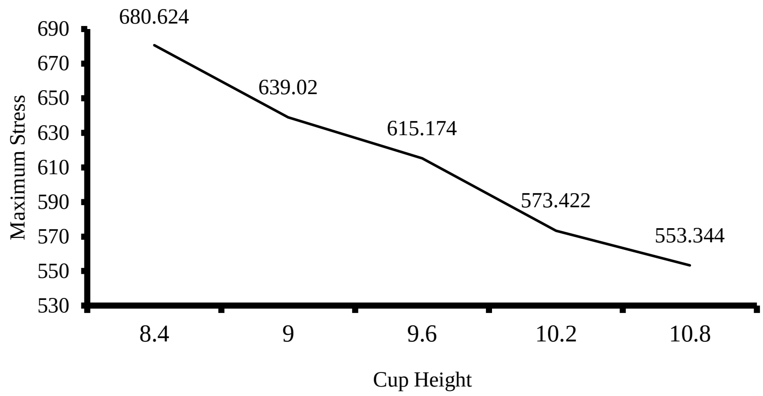
According to Figure 13, it is found that the weak point of the flexible wheel is at the transition circle of the tooth root and the transition circle of the cup bottom. According to the theoretical analysis, the failure of the flexible wheel mainly starts from these two points. Compared with the tear of the flexible wheel in the failure photo of the flexible wheel in the limit test, the theoretical analysis is consistent with the actual failure law of the flexible wheel. At last, set the parameters as shown in Table 6. However, virtual test data cannot be guaranteed to be directly applied to manufacturing, and parameters must be adjusted through sample testing. Virtual simulation data gives direction to the design. Through pendulum testing, the flexible wheel with these parameters has achieved a lifespan of 9000 hours under rated load conditions. The LCD17 harmonic gear is shown in Figure13.

Table 5. Results and analysis of orthogonal simulation test

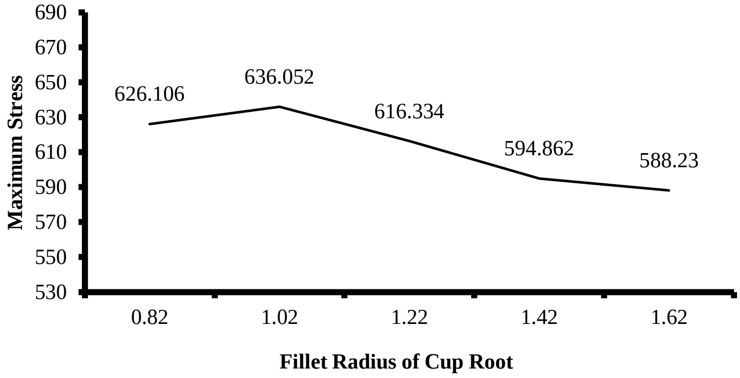
Factor / Serial number	Root fillet radius level	Cup height level	Fillet radius of cup root level	Thickness of flexible wheel cup wall level	Maximum stress/MPa
1	1.6	8.4	0.82	0.26	660.03
2	1.6	9	1.02	0.27	650.95
3	1.6	9.6	1.22	0.28	604.18
4	1.6	10.2	1.42	0.29	555.25
5	1.6	10.8	1.62	0.3	587.9
6	1.8	8.4	1.02	0.28	685.59
7	1.8	9	1.22	0.29	645.03
8	1.8	9.6	1.42	0.3	655.52
9	1.8	10.2	1.62	0.26	529.16
10	1.8	10.8	0.82	0.27	548.14
11	2	8.4	1.22	0.3	764.55
12	2	9	1.42	0.26	583.78
13	2	9.6	1.62	0.27	565.95
14	2	10.2	0.82	0.28	579
15	2	10.8	1.02	0.29	567.26
16	2.2	8.4	1.42	0.27	633.77
17	2.2	9	1.62	0.28	598.96
18	2.2	9	0.82	0.29	594.04
19	2.2	10.2	1.02	0.3	653.22
20	2.2	10.8	1.22	0.26	517.43
21	2.4	8.4	1.62	0.29	659.18
22	2.4	9	0.82	0.3	716.38
23	2.4	9.6	1.02	0.26	623.24
24	2.4	10.2	1.22	0.27	550.48
25	2.4	10.8	1.42	0.28	545.99
K1	3058.31	3403.12	3097.59	2913.64	
K2	3063.44	3195.1	3180.26	2949.29	
K3	3060.54	3042.93	3081.67	3013.72	
K4	2997.42	2867.11	2974.31	3020.76	
K5	3095.27	2766.72	2941.15	3377.57	
k1	611.662	680.624	619.518	582.728	
k2	612.688	639.02	636.052	589.858	
k3	612.108	608.586	616.334	602.744	
k4	599.484	573.422	594.862	604.152	
k5	619.054	553.344	588.23	675.514	
R	19.57	127.28	47.822	92.786	



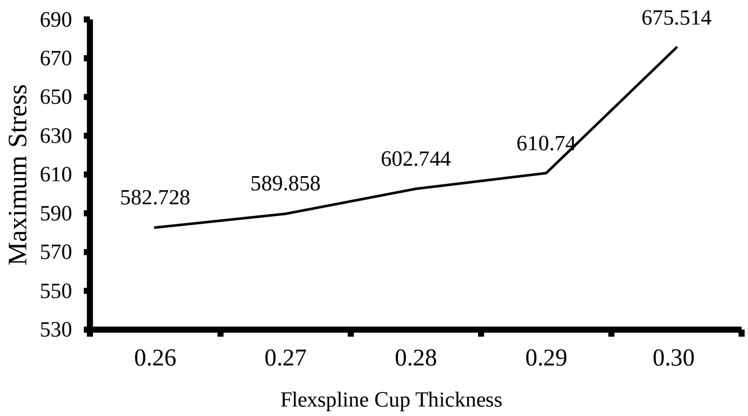
(a) Root Fillet - Maximum Stress



(b) Cup Height - Maximum Stress



(c) Fillet Radius of Cup Root - Maximum Stress



(d) Flexspline Cup Thickness - Maximum Stress

Figure 13. Each Index Parameter k Value and The Maximum Stress

Table 6. Parameter selection

Root fillet radius	Cup height	Fillet radius of cup root	Thickness of flexible wheel cup wall
2.2	10.8	1.62	0.26

Figure 13. LCD17 harmonic gear

7. CONCLUSION

Based on orthogonal simulation test, the flexible wheel of LCD 17 harmonic reducer has been successfully designed in one attempt, whereas in the past, it required at least 3 iterations.

Based on the research above, the following conclusions are drawn.

1. (1) The physical destructive test and simulation analysis show that the flexible wheel of harmonic reducer is mainly at the place where the gear ring contacts the thin wall of the flexible wheel and the place where the thin wall of the flexible wheel contacts the bottom surface of the cup body. These two positions are prone to failure.
2. (2) The cup height has a great influence on the value of the maximum stress, but due to the external dimension and processing difficulty of the reducer, the selection of the cup height parameter needs to be designed according to the load, which is one of the reasons for the large difference in the axial dimension between the light load harmonic reducer and the heavy load harmonic reducer.
3. (3) Simplifying each component model of harmonic reducer and conducting orthogonal simulation test research through workbench software can optimize the design of flexible wheel parameters of harmonic reducer, which is an efficient flexible wheel optimization design method.

The conclusions of this paper can be applied to the design of the flexible wheel of harmonic reducers, enhancing design efficiency and reducing the waste associated with physical trial-and-error iterations.

The harmonic reducer has more simulation parameters for the orthogonal test of the complete machine, so how to efficiently conduct the orthogonal test simulation of the complete machine is of greater significance.

Funding

This study was supported by Research on torque motor of high torque density and adaptive drive technology of collaborative robots (No.22KJA460008), Scientific research start-up fund project of Suzhou Vocational Institute of Industrial Technology (No.2017kyqd030) and Research on the structural construction scheme of the teaching team of higher vocational colleges under the background of “Double-high construction” (No. SGYJG2023A).

References

[1] Ivanov MH, Shen Y. Harmonic Gear Drive. Beijing: National Defence Industry Press, 1987, p.13-29.

[2] Shen YW, Ye QT. The Theory and Design of Wave Gear Transmission. 1rd ed. Beijing: Mechanical Machine Press; 1985, p.8.

[3] He W, Xue WD, Tang B. Optimization Test Design Method and Data Analysis. 1rd ed. Beijing: Chemical Industry Press, 2012, p.29-41.

[4] Chai W. Studies on Deformation Characteristics of Flexible Wheel in Harmonic Gear Drive. Beijing: China University of Geosciences (Beijing); 2017.

[5] H. Gao, Z. Li, Z. Deng, “Sensitivity Analysis of Cup-shaped Flexible Gear Parameters to its Stress Based on ANSYS”, Journal of Mechanical Engineering, Vol. 46, No.5, 2010, pp.1-7. https://doi.org/10.3969/j.issn.1004-2539.2008.04.007

[6] C. Zhang, S. Wang, J. Shao, “Simulation Analysis of Harmonic Gear Reducer Fatigue Life Based on ANSYS”, Hydraulics Pneumatics & Seals, Vol. 32, No.8, 2012, pp.72-74. https://doi.org/10.3969/j.issn.1008-0813.2012.08.024

[7] Ma D. Study of Kinematics and Meshing Characteristic of Harmonic Gear Drives Based on the Deformation Function of the Flexspline. Dalian: Dalian University of Technology; 2008, p. 18-20.

[8] J. Fu, H. Dong, Y. Shen, “Stress Analysis of the Flexspline in Harmonic Gearing by Using FEM”, China Mechanical Engineering, Vol. 18, No.18, 2007, pp.2214-2210. https://doi.org/10.3321/j.issn:1004-132x.2007.18.017

[9] H. Dong, D. Li, S. Qi, “Structural Optimization of the Flex Spline Cup in Harmonic Drives Based on Orthogonal Test and Finite Element Analysis”, Journal of Mechanical Transmission, Vol. 37, No.01, 2013, pp.34-38. https://doi.org/10.16578/j.issn.1004.2539.2013.01.013

[10] Y. Wang, S. Li, T. Xie, “Finite Element Mechanics Analysis of Flexspline in Harmonic Driver and Improvement of Structural Parameter”, Optics and Precision Engineering, Vol. 13(Supplement), 2005, pp.86-90. https://doi.org/10.3321/j.issn:1004-924X.2005.z1.019

[11] W. Liu, N. Zhang, C. Zhang, Y. Zhao, “Finite Element Calculation and Analysis on Column-shaped Flex Spline of Harmonic Drive”, Chinese Journal of Mechanical Engineering, Vol. 42, No. 4, 2006, pp.52-57. https://doi.org/10.3321/j.issn:0577-6686.2006.04.010

[12] J. Deng, R. Yang, X. Deng, X. Gao, “Stress Analysis and Parameter Optimization of Column-shaped Flexsplinein Harmonic Drive Based on ANSYS”, Machinery Design & Manufacture, Vol. 46, No.05, 2012, pp.59-61. https://doi.org/10.3969/j.issn.1001-3997.2012.04.022

[13] Q. Zhang, D. Huang, J. Gu, “Buckling Analysis for Flexspline Structure in Harmonic Drive with FEM”, Journal of Mechanical Transmission, Vol. 412, No. 2, 2017, pp.52-57. https://doi.org/10.16578/j.issn.1004.2539.2017.02.026

[14] Yang P. Harmonic Gear Driving Device and the Short Flexspline Study. Beijing: China Academy of Machinery Science and Technology, 2006, p.6-8.

[15] X. Wang, X. Zhou, C. Li, “The Sructure Optimization Design of the Flexspline of Harmonic Reducer”, Machinery Design & Manufacture, Vol. 47, No. 11, 2013, pp.67-69. https://doi.org/10.3969/j.issn.1001-3997.2013.11.020