Deformation and Stress Analysis of Flex spline in Harmonic Drive based on Finite Element Method

Chunjian Liu[1, a], Lijie Chen[1, b, *] and Cheng Wei [2, c]

[1]Department of Aeronautics, Xiamen University, Xiamen 361005, China

[2]Department of Aerospace Engineering, Harbin Institute of Technology, Harbin 150001, China

[a]liuchunjian@live.cn, [b]chenlijie@xmu.edu.cn, [c] weicheng@hit.edu.cn

Abstract. As a new driving method, harmonic drive has broad application prospects. This paper aims to analyze the deformation and stress of the flexspline, a main component of the harmonic drive gear to provide a basis for the assessment of its fatigue and fracture failure. First, the structures of a harmonic drive reducer is designed. Then, finite element method is used to simulate the assembling process of wave generator and the dynamic working process of harmony drive gear. The deformation and stress distributions of the flexspline are obtained. The simulation results show that the stress generated during the flexspline transmission is the main cause of the fatigue failure of the flexspline and the wear of the gear teeth. These simulation results provide an important reference for the flexspline's further experimental design of the fatigue and wear properties.

Keywords: harmonic drive, flex spline, finite element method, deformation, stress.

1. Introduction

Harmonic drive is a new driving method that utilizes the elastic deformation wave formed in a flexible component to realize motion or power transmission. Harmonic drive gear is widely used due to its advantages, such as simple structures and high single-stage transmission ratio. As an important component in harmonic drive, flexspline is an elastic thin-walled structure and works under alternating loads. As a result, fatigue failure of flexspline is one of the most common failure modes [1-3]. The research on deformation and stress distributions of flexspline is the basis for the evaluation of the flexspline's fatigue failure in the harmony drive gear's dynamic working process.

Deformation and stress distributions of flexsplines are usually determined with the empirical formula and experimental method. For example, Ivanov [3] and Shen et al. [4] provided a calculation model using the column shell theory based on elastic theory and some assumptions and made complements for the model according to the experimental results. In recent years, there are some works based on the finite element method to obtain more accurate deformation and stress distributions of flexspline. Kayabasiet et al. [5] conducted a stress–strain analysis of the flexspline based on the finite element method in order to calculate the stress on flexspline teeth and find optimum shape of teeth to maximize fatigue life. Zhang [6] defined the boundary conditions and initial load by surface-to-surface contact between flexspline and wave generator through the finite element contacting analysis. The distribution of flexspline deformation and stress can be acquired under different conditions, which provides valuable data for flexspline structural optimization. Chen et al. [7] built a finite element model of flexspline based on shell element. The deformation distribution of the flexspline in transmission state was obtained by nonlinear contact analysis. Zou et al. [8] established a detailed surface-to-surface contact finite element model of harmonic drive system and found that the deformation and stress at the flexspline gear cross-section changing geometrically with heightened load although the distributions of the deformation and stress increments remain unchanged. However, because of the high reliability and life, the above analysis still cannot meet the requirements of structural integrity assessment.

In this paper, we design a harmonic drive reducer and then perform simulations of wave generator assembling process and harmony drive gear dynamic working process to obtain the deformation and stress distribution of the flexspline. The results are very important for the experiments design of the flexspline's fatigue and wear properties.

2. Modeling

The Dimension of Harmonic Drive Gear. The dimension of flexspline structure is shown in Fig. 1. Its torque loaded is 5Nm, speed 2 degree per second, module $m = 0.2$, transmission ratio $i = 0.2$, pressure angle $\alpha = 20^{\circ}$, gear tooth number $z_1 = 200$, addendum coefficient $h_a^* = 1.0$, and radial clearance coefficient $c^* = 0.25$.

The radial deformation coefficient is a ratio between flexspline's maximum radial deformation and gear module, which is determined by $\omega_0^* = \frac{\omega_0}{m}$. ω_0 Is flexspline's maximum radial deformation and is equal to the difference between the radius of wave generator's long axis and the radius of flexspline's inner wall. m Is the flexspline's gear module. Generally, the meshing depth can be increased by enlarging deformation coefficient which results in the increase of the stress of flexspline. Excessive deformation coefficient is not recommended. Commonly, it is chosen as $\omega_0^* = 0.9 \sim 1.1$. In this design, we choose $\omega_0^* = 0.9$. Given the module $m = 0.2$, the flexspline's maximum radial deformation is $\omega_0 = 0.18mm$.

The wave generator is a component that makes flexspline generate continuous wave. Cam type wave generator mainly consists of cam and flexible rolling bearing with complex structure. To decrease the calculation time of finite element simulations, the wave generator can be simplified as a rigid structure in order to simulate the effect of the wave generator on the flexspline.

Here we take the outer contour line of the flexible bearing as the contour line of the wave generator. Thus the rigidization process of the wave generator is realized. The contour line of wave generator is:

$$\rho_R = R + \omega_0 \cos 2\varphi \quad (1)$$

Where R is 20mm and ω_0 is 0.18mm. The model of the harmonic drive gear is shown in Fig. 2.

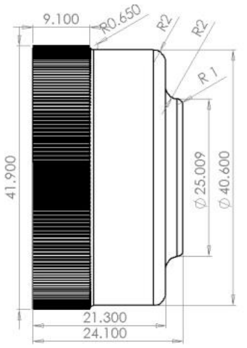

Fig. 1 The dimension of flexspline

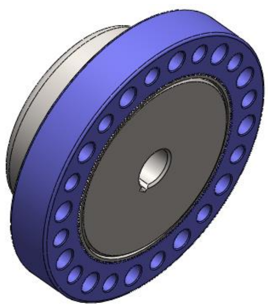

Fig. 2 Model of harmonic drive gear

Finite Element Modeling. The flexspline's is made of 40CrNiMoA steel. The material properties are: density 7850kg/m^3, elastic modulus 209GPa, and Poisson's ratio 0.295. The finite element analysis model is developed in the commercial software Patran.

This analysis is a rigid - flexible contact problem, in which the flexspline is a flexible deformation and wave generatior is a rigid-body. Due to the flexspline symmetry and the computational complexity, a quarter part of the flexspline is used for analysis of asembling process, as shown in figure 3. Finite element mesh of the flexspline is shown in Fig. 4.

After setting the distance of extrusion as 0.4mm, the contact relationship is defined by contact element between the flexspline and wave generatior. Symetric boundary conditions are applied, as shown in Fig. 5.

The solution is done by the software Nastran. By using implicit nonlinear module SOL 600 (MD Nastran provides this module to solve nonlinear issues), we simulate wave generator's assembling process. Importing assembling stress and deformation into dynamic simulation process as initial conditions, we use implicit nonlinear module SOL 600 to simulate harmonic gear dynamic working process. Then the deformation and stress distribution of the flexspline under alternating load can be solved.

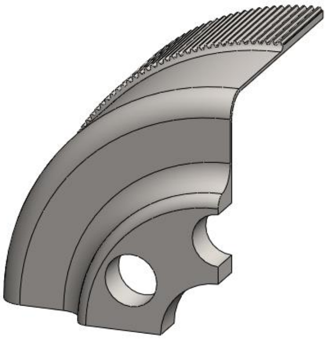

Fig. 3 A quarter of the flexspline

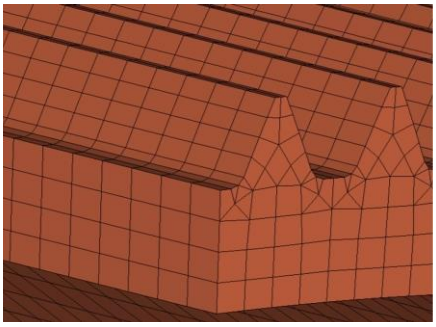

Fig. 4 Finite element mesh generation of the flexspline

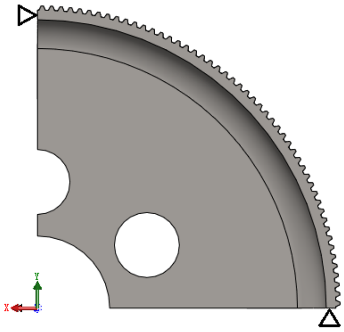

Fig. 5 The displacement constraints of the flexspline

3. Finite Element Analysis Results

Assembling Simulation Resluts of Wave Generator. During the assembling of the wave generator, the wave generator extrudes the flexspline and causes the initial deformation and assembling stressbecause the long axis of the wave generator is a little bit larger than the inner diameter of the flexspline. Thus, the analysis of the flexspline's initial deformation and the assembling stress are very important for the researches on the dynamic working performances of harmonic gear.

Fig. 6 is the contour plot of the deformation of the flexspline's initial deformation. It shows that the deformation of the flexspline is large at the position closed to the cup rabbet at the long and short axes, and the defamation decreases along the direction of tube. Fig. 7 shows the deformation in the long axis direction, which reaches a maximum value of 0.1918mm near cup rabbet. Fig. 8 shows the deformation in the short axis direction, which reaches a maximum value of 0.1654mm near cup rabbet.

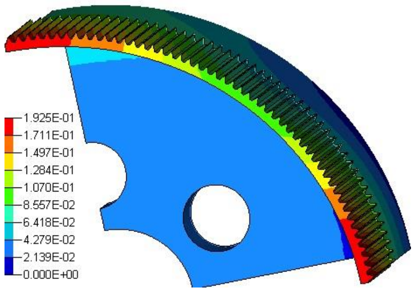

Fig. 6 Initial deformation of the flexspline

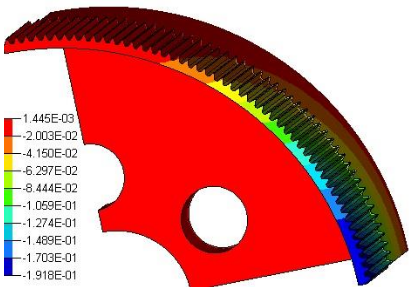

Fig. 7 Initial deformation of the flexspline at long axis (x)

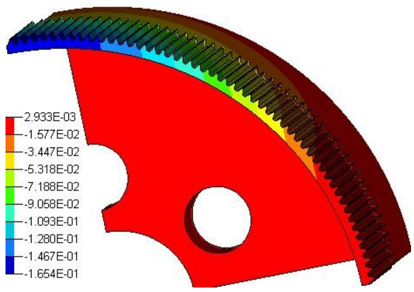

Fig. 8 Initial deformation of the flexspline at short axis (y)

The Von Mises stress and the first principal stress of the flexspline during the assembling are shown in Fig. 9 and Fig. 10, respectively. Correspondingly, Table 1 lists each stress component of the maximum stress point. Thus, the stresses at the long and the short axes in gear ring are both large. Von Mises stress near the cup rabbet at the long axis reaches a maximum value of 177.6Mpa and the maximum first principal stress is 210.0MPa. Because of the assembling of wave generator, the contact area for wave generator and flexspline is at the gear ring and the flexspline tube is extruded, which causes the normal and the shear stresses and the latter is larger than the former.

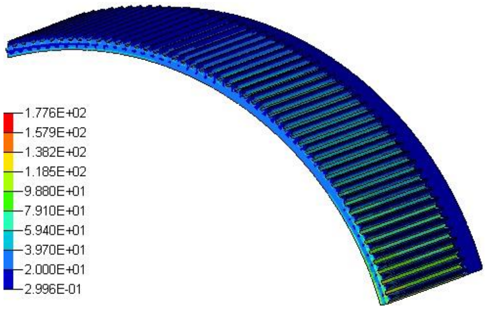

Fig. 9 The Von Mises stress of the flexspline during the assembling

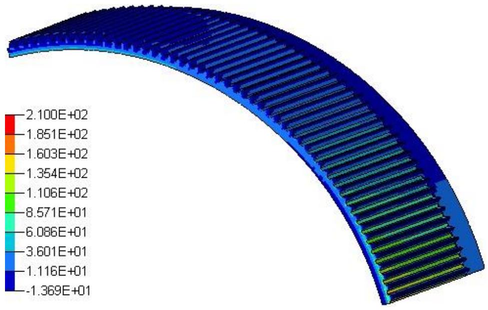

Fig. 10 The first principal stress of the flexspline during the assembling

Table 1 Stress components of the flexspline during the assembling

Stress (Mpa)	Node Number	Stress components (Mpa)					
		σ_{XX}	σ_{YY}	σ_{ZZ}	σ_{XY}	σ_{YZ}	σ_{ZX}
Maximum Von Mises: 177.6	626290	34.26	196.9	46.08	-47.50	-4.889	2.242
MaxShear: 94.3	627990	32.75	194.0	51.67	-46.69	-1.903	0.6493

Dynamic Simulation Results of Transmission. During harmonic gear transmission, the deformation and stress state of the flexspline are complex because of the following two reasons: 1) the initial deformation and stress caused by wave generator's assembly, 2) the complexity of the harmonic gear's structure and the meshing effect by the flexspline with the rigid gear. This alternating stress results in the fatigue failure of the flexspline, the wear of gear tooth and so on. Thus, it is necessary to do further researches on the deformation and stress state of the flexspline under the dynamic loading.

The distributions of the Von Mises stress and the first principal stress of the flexspline during transmission are shown in Fig. 11 and Fig. 12, respectively. Table 2 list each stress component of the flexspline during the transmission.

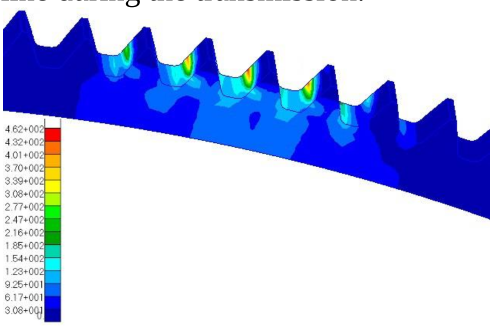

Fig. 11 The Von Mises stress of the flexspline during the transmission

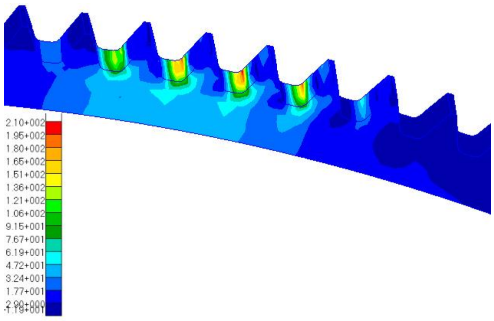

Fig. 12 The first principal stress of the flexspline during the transmission

Table 2 Stress components of the flexspline during the transmission

Stress (Mpa)	Node Number	Stress components (Mpa)					
		σ_{XX}	σ_{YY}	σ_{ZZ}	σ_{XY}	σ_{YZ}	σ_{ZX}
Maximum Von Mises: 462	620048	-245	-197	-42.5	-219	-14.8	18.2
Maximum Shear: 259	619726	-184	-170	-48.0	-238	-4.49	7.98

From these figures, the stress is large in gear ring. The maximum stress value is not right at the long axis, but at the gear root near the long axis. The maximum Von Mises stress and first principal stress are 462 MPa and 210MPa, respectively. During harmonic gear transmission, normal stress and shear stress are both large, which are caused by meshing of the flexspline with the rigid gear.

4. Conclusions

In this research, a harmonic drive reducer is designed and the assembling process of wave generator and the dynamic working process of harmonic gear transmission are simulated with finite element method in order to provide references for the experimental design of fatigue-wear properties of fatigue-wear properties of flexspline. The main conclusions are as follows:

(1) The analysis of the assembling process of wave generator and flexspline show that: i) the initial deformation of the flexspline is large near the cup rabbet at the long and the short axes, and the deformation decreases along the direction of tube; ii) the initial stress is high near gear ring at the long and the short axes, and it reaches the maximum value when the long axis becomes close to the gear root of the cup rabbet; iii) extrusion of the flexspline tube causes the normal the shear stresses and the latter is relatively larger than the former.

(2) Similar with the assembling process, in the process of harmonic gear transmission, the stress of the flexspline is strong near gear ring. The maximum value of flexspline's stress does not appear right at the long axis, but at the gear root with a few teeth deviation from the long axis. Due to the meshing of the flexspline and the rigid gear, the relatively larger normal stress and shear stress are caused at the gear root. Comparing to assembling process, the stress caused by harmonic gear transmission is much larger. This is the main reason of the fatigue failure of the flexspline and the wear of gear teeth.

Acknowledgements

This study was supported by the Fundamental Research Funds for the Central Universities and the National Basic Research Program of China (Grant No. 2013CB733004).

References

[1] Y.F. Wang, S.G. Li, T. Xie, Finite element mechanics analysis of flexspline in harmonic driver and improvement of structural parameter, Optics and Precision Engineering. 13 (2005) p.86-90. (In Chinese)

[2] W. Shi, Q. Zhang, X. Li, The research about deformation and stress of flexspline in harmonic drive based on ABAQUS, Mechanical Engineer. 6 (2010) p.44-47. (In Chinese)

[3] M.H. Ivanov, the Harmonic Drive, National Defense Industry Press, Beijing, 1987. (In Chinese)

[4] Y.W. Shen, Q.T. Ye, the Theory and Design of Harmonic Drive, China Machine Press, Beijing, 1985. (In Chinese)

[5] O. Kayabasi, F. Erzincan, Shape optimization of tooth profile of a flexspline for a harmonic drive by finite element modeling, Materials and Design. 28 (2007) p.441-447.

[6] S.M. Zhang, Stress Analysis of Flexspline in Harmonic Gear Drive Based on ANSYS, Master Dissertation, Shanghai Jiao Tong University, 2009. (In Chinese)

[7] X.X. Chen, S.Z. Lin, J.Z. Xing, Y.S. Liu, Deformation of harmonic drive in transmission state based on contact analysis with shell element tooth, Applied Mechanics and Materials. 86 (2011) p.771-774.

[8] C. Zou, T.T, G.D. Jiang, X.S. Mei, Deformation and stress analysis of short flexspline in the harmonic drive system with load, 2013 IEEE International Conference on Mechatronics and Automation, 2013, p.676-680.