

Fatigue Life Analysis of Flexspline Based on Finite Element Simulation Model with Non-Simplified Flexible Bearing Structure of Harmonic Reducer and Measured SN Curve

Liang-Shan Xiong*, Jun-Fan Xie**, Yong Wang** and Long-Jun Luo***

Keywords: flexspline, fatigue life, finite element dynamic simulation, SN curve

ABSTRACT

When predicting the fatigue life of the flexspline in a harmonic reducer using the simulation results from the dynamic numerical model of the harmonic reducer, it is common practice to simplify the flexible bearing structure significantly for convenience. Additionally, the SN curve of the material, obtained from tests conducted by others, is often directly used for calculations. This increases the likelihood of significant errors in the predicted fatigue life of the flexspline. To address this issue, a method is proposed that uses the stress simulation results from a harmonic reducer numerical model with a non-simplified flexible bearing structure, combined with the experimentally measured SN curve, to predict the fatigue life of the flexspline. Firstly, a finite element dynamic model of a hat-type harmonic reducer, which includes a complete flexible bearing structure (often simplified by researchers for computational convenience when constructing numerical simulation models of harmonic reducers) and accurately reflects the stress state of the flexspline, is established. Subsequently, the SN curve of the flexspline material 40CrNiMoA is obtained by single-point test. Finally, combined with the finite element stress analysis results and material SN curve, the fatigue life of the flexspline is calculated in nCode software.

Paper Received February, 2025. Revised June, 2025. Accepted August, 2025. Author for Correspondence: Liang-Shan Xiong and Jun-Fan Xie

* College of electromechanical and automation, Wuchang Shouyi University, Wuhan 430074, China

** School of Mechanical Science and Engineering, Huazhong University of Science and Technology, Wuhan 430074, China

*** Engineering practice innovation center, Huazhong University of Science and Technology, Wuhan, Hubei 430074, China

The results indicate that the fatigue life predicted by the proposed method is close to the average life of harmonic reducer products in China. The research can be applied to the structural parameter optimization of flexspline.

INTRODUCTION

The service life of harmonic reducers and their main components is a crucial factor determining the operational performance of many important electromechanical equipment in aerospace, industrial robotics, national defense, and other fields (Han, 2019; Deng, 2018). A harmonic reducer primarily consists of a flexspline, a circle spline and a wave generator (comprising a cam and a flexible bearing). Among them, the flexspline, serving as a weak link in the motion and power transmission of the harmonic reducer, is prone to fatigue failure (Ni, Zhou, Lu, and Lv, 2023; Zhang, Jiang, Zhang, Yun, and Mei, 2020). Its fatigue life directly determines the service life of the harmonic reducer and even the entire equipment. Therefore, investigating the factors and laws influencing the fatigue life of flexspline, as well as predicting and improving their fatigue life based on these laws, holds significant practical significance (Yan, Yang, Duan, and Qiu, 2013).

Currently, numerous scholars analyze and predict the fatigue life of the flexspline by acquiring the stress state of the flexspline and the fatigue life curve (SN curve) of flexspline materials (Cheng, 2024; Yi, 2023). Wu, Tao, Chen, Wu, Shu, Chen, and Zhang (2022) simplified the teeth of the flexspline into an equivalent ring and conducted finite element dynamic simulation on the assembly of the flexspline and the wave generator (neglecting the flexible bearings) to obtain the stress state of the flexspline. Then they derived the SN fatigue life curve according to the tensile limit of the flexspline material and established a fatigue life model of the flexspline in the Designlife module of the nCode fatigue analysis software. Yang, Li, Xu, Guo,

and Li (2022) similarly ignored the entire flexible bearing in their finite element numerical simulation of harmonic reducers to obtain the stress of the flexspline. They analyzed the influence of changes in the length and wall thickness of the flexspline cylinder on the fatigue life of the flexspline. Fu (2017) based on a three-dimensional model incorporating flexible bearings of a cup-type harmonic reducer, conducted finite element static simulation to obtain the stress state of the cup-type flexspline. He then approximated the strain amplitude and stress fatigue life curve based on flexspline material parameters such as the tensile strength and elastic modulus, and subsequently calculated the fatigue life of the flexspline. Huang, Wu, and Liao (2021) conducted finite element dynamic simulation on hat-type harmonic reducers under the condition that the cage composed of the flexible bearing was ignored and the friction contact between the rollers and the inner ring of the flexible bearing was simplified into binding contact. They provided a relatively accurate prediction of the flexspline fatigue life by referencing fatigue life data of the flexspline material provided in other literature. Cheng, Yuan, Zhang, and Liu(2024) further established the corresponding relationship between the flexspline structural parameters and its fatigue life through a BP neural network after calculating the fatigue life using the finite element stress calculation results and SN curve of the flexspline.

Although researchers have done a lot of effective work in the study of fatigue life prediction of the flexspline and achieved a lot of valuable research results, there are still two problems to be solved. First, researchers often simplify the geometric structure of flexible bearings (such as ignoring the cage inside the flexible bearing or even ignoring the entire flexible bearing structure) and the connection relationships between its internal components (such as simplifying the rollers and the inner and outer rings of the flexible bearing into binding contact) to improve the efficiency of finite element dynamic simulation calculation and quickly obtain the stress state of the flexspline. However, previous studies have shown that the stress on the flexspline is influenced by the rollers in the flexible bearing (Lyuminarsky, Lyuminarsky, and Lyuminarskaya, 2019). Simplifying the flexible

bearing structure or the connection relationships between internal components inevitably leads to distortion of the calculated stress state of the flexspline and deviations in the predicted fatigue life of the flexspline. Therefore, it is urgent to establish the finite element dynamic simulation model of the harmonic reducer based on the complete flexible bearing structure, and use the simulation results of the model to accurately predict the fatigue life of the flexspline. Second, in order to reduce the experimental cost, researchers often use a simple estimation of the SN curve coefficients for the flexspline material or directly reference coefficients of similar material SN curve equations from other literatures to determine the coefficients of the SN curve equation for flexspline materials and use them for fatigue life calculations. Obviously, this will lead to significant errors in the calculated fatigue life of the flexspline. It is well known that even materials of the same brand but from different production batches, the coefficient of the SN curve equation will be greatly different. Therefore, the accurate measurement of SN curve equation coefficient of the same batch material with the predicted life of the flexspline has become a key problem that must be solved to accurately predict the fatigue life of the flexspline.

To address these challenges, this study successfully achieved accurate prediction of the fatigue life of a hat-type harmonic reducer (in which the hat-type flexspline is manufactured using 40CrNiMoA material) using a finite element numerical simulation model with a non-simplified flexible bearing structure of the harmonic reducer and experimentally obtained SN curve of the flexspline material. The analysis process as shown in Figure 1. Firstly, a finite element mesh model of the hat-type harmonic reducer including the complete flexible bearing structure is established by manual method, and imported into Abaqus software to simulate the stress under the working condition of the flexspline. Then, the SN curve of 40CrNiMoA material is obtained through single-point testing. Finally, using nCode software and the obtained SN curve, the prediction of the fatigue life of the flexspline is realized, yielding satisfactory results.

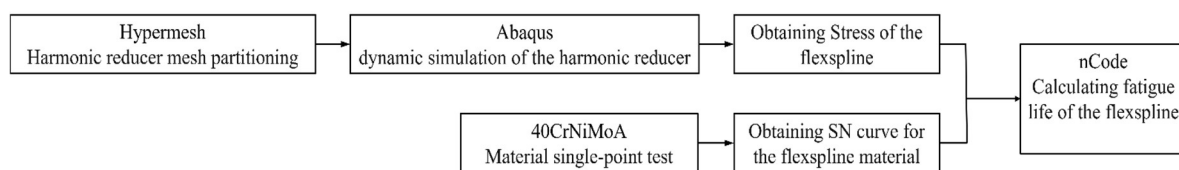


Fig 1. Flow chart of flexspline fatigue life analysis

THE ESTABLISHMENT OF FINITE ELEMENT MODEL WITH NON-SIMPLIFIED FLEXIBLE BEARING OF THE HAT-TYPE HARMONIC REDUCER AND THE ACQUISITION OF THE FLEXSPLINE STRESS STATE

The Establishment of Three-dimensional Model of the Hat-type Harmonic Reducer

The flexspline, intended for fatigue life analysis and prediction, belongs to a certain hat-type harmonic reducer produced by a certain enterprise in China. As mentioned above, this hat-type harmonic reducer is mainly composed of a wave generator (consisting of a cam and a flexible bearing), a flexspline and a circular spline, as shown in the three-dimensional model in Figure 2. The flexspline has 200 teeth with a modulus of 0.3mm, and its structural schematic is illustrated in Figure 3, with dimensions provided in Table 1. The flexible bearing consists of an inner ring, an outer ring, a cage and 23 rollers distributed along the circumference.

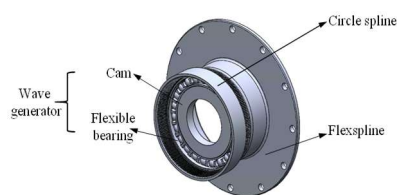


Fig 2. Harmonic reducer 3D model diagram

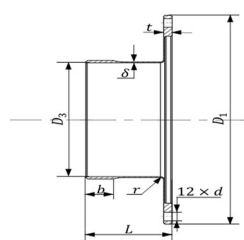


Fig 3. Schematic diagram of the flexspline structure

Table 1. The flexspline sizes

Parameter	Symbol	Value (mm)
Flexspline tooth width	b	13
Flexspline cylinder length	L	34
Flexspline wall thickness	δ	0.4
Flexspline inner diameter	D_3	60
Flexspline outer diameter	D_1	110
Inside diameter of cylinder bottom flange	D_2	88
Bottom flange height	t	3.5
Round corners at the bottom of the cylinder	r	1.3
Mounting hole diameter	d	4.5

The Material and Properties of Each Component of the Hat-type Harmonic Reducer

The material of the flexspline is 40CrNiMoA alloy steel; the material of the inner and outer rings of the flexible bearing is GCr15 bearing steel; the cage is made of nylon. The main physical properties of these materials are shown in Table 2.

Table 2. Material property

Part	Material	Density (kg/m ³)	Elastic modulus (GPa)	Poisson's ratio
Flexspline	40CrNiMoA	7850	200	0.29
Flexible bearing inner and outer rings	GCr15	7810	210	0.3
Flexible bearing cage	nylon	1320	140	0.24

During the operating process of the harmonic reducer, the flexspline and the inner and outer rings of the flexible bearing undergo large deformations, whereas the deformations of the cam, circle spline, and bearing rollers can be neglected. Therefore, in the upcoming dynamic simulation of the cap-type harmonic reducer, in order to reduce the computational workload, the cam, circle spline, and bearing rollers are all modeled as rigid bodies, following the approach of previous studies (Ni, Zhou, Lu, and Lv, 2023; Wang, Zhao, Li, Zhou, Yan, and Hu, 2023).

The Establishment of Finite Element Dynamic Analysis Model of The Hat-type Harmonic Reducer

(1) Mesh partitioning

The first step in establishing a finite element dynamic simulation model for the hat-type harmonic reducer, given an existing 3D model, is to mesh the components of the harmonic reducer. Under the constraint of limited computational resources, the difference of mesh partitioning will directly affect the accuracy and efficiency of finite element simulation. To appropriately improve simulation efficiency while ensuring simulation accuracy, the following two points should be considered when setting mesh sizes for the simulation model of the harmonic reducer: (1) The hat-type harmonic reducer contains a lot of thin-wall structure, and the tooth shape of the flexspline and circular spline is complex. Only by making the established mesh model closely approximate the geometric structure characteristics of the components of the hat-type harmonic reducer, can the accurate stress simulation results be obtained, which requires that the mesh size should not be set too large during meshing; (2) if using the automatic mesh partitioning method, and the goal is to improve the computational efficiency of finite element simulation, the mesh size of the entire harmonic reducer can not be set too small.

In this paper, the CAE pre-processing software Hypermesh is used to manually mesh the three-dimensional geometric model of the harmonic reducer. The method is as follows: (1) In the thin-walled structure area (such as the flexspline wall thickness of 0.4 mm), ensure that at least 2 layers of meshes cover the wall thickness direction; (2) In the tooth surface area, divide at least 5 layers of meshes along the tooth height direction, and in the curved contour area, divide at least 3 segments of mesh edges to capture stress concentration details; (3) For areas with large dimensions or simple contours, larger-sized meshes are assigned; (4) In regions with significant changes in mesh size (where one side of the region has larger meshes while the other side has smaller meshes), a transition mesh consisting of a combination of trapezoidal and square meshes is set. During mesh partitioning, non-critical stress areas such as assembly holes are removed. Following the above method, the hat-type harmonic reducer to be studied is divided into a total of 682903 hexahedral meshes (Figure 4).

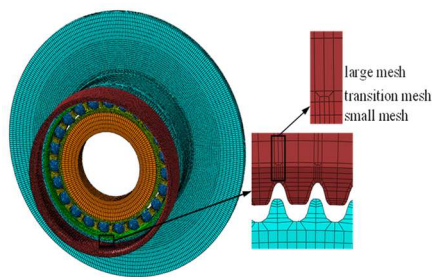


Fig 4. Harmonic reducer grid model and local magnification diagram

(2) Contact setting

After the mesh model of the proposed hat-type harmonic reducer is imported into the Abaqus 2022 software system, it is necessary to set up contacts between the components, including the following contact types: contact between the cam and the inner ring of the flexible bearing, contact between the inner ring of the flexible bearing and the rollers, contact between the rollers and the outer ring of the flexible bearing, contact between the outer ring of the flexible bearing and the flexspline, contact between the teeth of the flexspline and the teeth of the circular spline. In order to ensure the simulation accuracy, all of the above contact types are set as friction contact based on the actual situation, and all the friction coefficients are set as 0.1 according to the practice of predecessors (Wang, Zhao, Li, Zhou, Yan, and Hu, 2023).

(3) Load and constraint settings

The hat-type harmonic reducer to be studied is fixed with circular spline, with motion input from the wave generator and output from flexspline. To simulate the working state of this harmonic reducer

and ensure convergence of the finite element calculation process in Abaqus software, the dynamic implicit analysis solver is employed. The analysis is divided into 4 steps, with each step having specific constraints and loads: In the first step, a fixed constraint is applied to the output end of the flexspline allowing deformation of the teeth of the flexspline under the action of the wave generator. In the second step, an axial displacement is applied to the circular spline to engage the teeth of the flexspline with the teeth of the circular spline. In the third step, the axial rotational degrees of freedom of the flexspline are released, and a rated torque of 67 N·m is applied to the output end of the flexspline to simulate the effect of the rated load on the flexspline. In the fourth step, a rated speed of 2000 rpm is applied to the elliptical cam of the wave generator, and the cam is rotated for 0.03 seconds at this speed, equivalent to one revolution.

After completing the above setup, create a job in Abaqus software and submit the calculation.

The Simulation Results of Stress State of The Flexspline

The stress state of the flexspline simulated by the above model under working state is shown in Figure 5. The simulation results indicate the maximum stress during working rotation is located at the rear end of the tooth ring in the transition region between the long axis and the short axis of the deformed flexspline. The maximum equivalent stress of the flexspline is lower than the yield strength of the material (the yield strength of 40CrNiMoA is greater than 785 MPa), indicating that the flexspline is in the elastic deformation stage, but prone to fatigue failure.

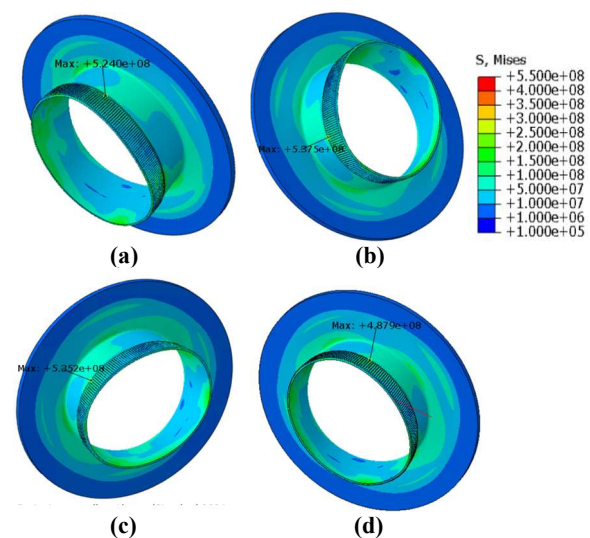


Fig 5. Equivalent stress cloud diagram of flexspline: (a) Stress state contour plot of the flexspline when the wave generator rotates by about 90 degrees. (b) Stress state contour plot of the flexspline when the wave generator rotates by about 180 degrees. (c) Stress state contour plot of the flexspline when the wave generator rotates by about 270 degrees. (d) Stress state contour plot of the flexspline when

the wave generator rotates by about 360 degrees

EXPERIMENTAL ACQUISITION OF THE SN CURVE OF THE FLEXSPLINE MATERIAL

SN Curve Test Method

To predict the fatigue life of flexspline, it is necessary to know not only the stress state of the flexspline, but also the SN curve of the flexspline material.

Therefore, a specially designed fatigue life test is conducted to obtain the SN curve of the flexspline material, 40CrNiMoA, used in the hat-type harmonic reducer under study.

As it is well known, there are various methods to obtain the SN curve of the flexible gear material, such as the single-point test method and the batch test method. Due to its higher efficiency and lower cost compared to the batch test method, the single-point test method is chosen here.

The test is planned to be conducted on the Shimadzu EHF-UV100k2-040-1A fatigue test machine. The test specimen is designed based on the GB/T 3075-2008 "Method for Axial Force Control in Fatigue Testing of Metal Materials", with its structure shown in Figure 6 and key structural parameters listed in Table 3.

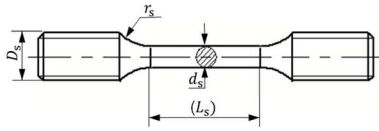


Fig 6. Schematic diagram of the test specimen structure

Table 3. Key structural parameters of the test specimen

Parameter	Symbol	Value/mm
External diameter	D_s	8
Outer transition arc radius	r_s	12
Measuring diameter	d_s	3.5
Reduced length	L_s	14

The Parameter Settings for The Fatigue Testing Machine

In the experiment, a stress ratio of $R = -1$ is selected, indicating that the alternating load is set as a symmetric cyclic load. The upper and lower limits of alternating load are as follows:

$$\begin{cases} \text{Upper limit: } P_{\max} = Sd \\ \text{Lower limit: } P_{\min} = RP_{\max} = -P_{\max} \end{cases} \quad (1)$$

Where d represents the measured diameter of the 40CrNiMoA material test specimen, with unit of mm; S represents the maximum value of stress levels borne by the specimen, with unit of MPa. The stress

levels are set according to the recommended principles of the single-point test method (Ling, 1987) and practical experience summarized during the experimental process, as detailed in Table 4.

Data processing and results of the SN curve testing

The fatigue load cycle counts endured by each test specimen at corresponding stress levels are presented in Table 4 below.

Table 4. Fatigue test data of 40CrNiMoA material

Serial number	Stress S /MPa	Cycle number N /cycles
1	715	4998
2	570	17748
5	357	10000000
6	430	1516719
7	500	269055
8	405	3515632

The functional relationship between the number of cycles N and the stress value S is set to the commonly used power function form, that is, $N = aS^{-b}$. Two undetermined coefficients are fitted using the stress-fatigue life test data obtained by the experiment and 1stOpt software. Consequently, the stress-fatigue life function relationship (SN curve) of 40CrNiMoA material in the high-cycle fatigue zone is determined as follows:

$$N = 5.14 \times 10^{30} S^{-9.29} \quad (2)$$

Where N represents the fatigue life in cycles, with unit of cycles; S represents the maximum alternating stress, with units of MPa.

The relationship between the experimental data and the fitted function is illustrated in Figure 7.

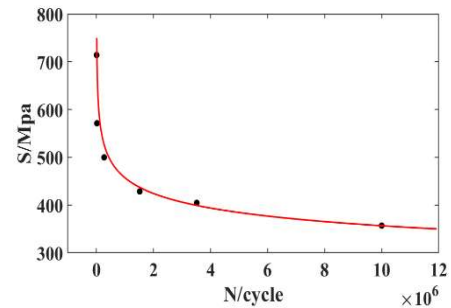


Fig 7. Experimental data and fitted SN curve

FATIGUE LIFE OF THE FLEXSPLINE BASED ON THE MEASURED SN CURVE

The fatigue life of the flexspline is analyzed and predicted using the nCode software. The method and process are outlined as follows:

(1) Create a material for 40CrNiMoA in the MaterialsManager module in the nCode software.

Based on the results of the aforementioned fatigue tests, equation (2) is utilized to define the stress-fatigue life parameters of the 40CrNiMoA material. Choose the material type as "nCode SN R-Ratio Curves" and set the stress ratio to -1.

(2) Import the finite element dynamic analysis results of the hat-type flexspline stress into the DesignLife module. Select the analysis method as "MultiRRatioCurve". Set the stress correction method to "Goodman", the stress combination method to "AbsMaxPrincipal" (Chen, 2021), and the load mapping type to "Time Step".

(3) Use the stress calculation results of the flexspline obtained from the fourth analysis step of the finite element analysis (which simulates the stress of the flexspline at a cam speed of 2000 rpm) as the input load into the nCode software.

The flexspline fatigue life calculation model is solved and the fatigue life contour map of the flexspline is obtained, as shown in Figure 8. From Figure 8, it can be observed that the flexspline is prone to fatigue failure at the inner wall of the tooth ring (a), the back end (b), and transition arc between the cylinder and the cap bottom (c), where the number of loading cycles is relatively low. The minimum number of cycles is 3.956×10^8 , with each cycle time being 0.03 s. Therefore, the fatigue life of the flexspline is calculated as $3.956 \times 10^8 \times 0.03 / 3600 = 3296.7$ hours.

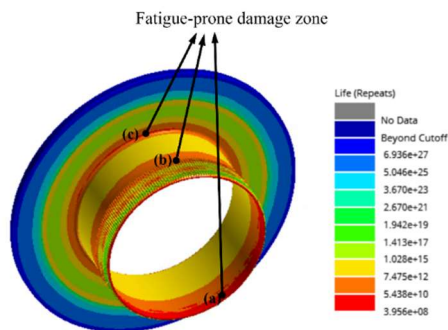


Fig 8. Life contour plot of the flexspline (Number of cycles)

Through literature review, it was found that the predicted result mentioned above is very close to the common fatigue life of harmonic reducer products in China (3000 hours (Wang, 2018)), as well as the actual fatigue life obtained during the product's usage.

CONCLUSIONS

The conclusions are as follows:

(1) A method is proposed to predict the fatigue life of the flexspline by using the stress simulation results from a harmonic reducer numerical model with a non-simplified flexible bearing structure, combined with the experimentally measured SN curve. This method, along with the SN curve parameters of the flexspline material obtained through single-point testing, enables accurate prediction of the flexspline's fatigue life.

(2) The maximum stress of the flexspline during working rotation is located at the rear end of the tooth ring of the transition area between the long axis and the short axis after the flexspline deformation. Significant stresses are also observed at the front end of the gear ring between the long and short axes after deformation and at the transition arc between the barrel and the cap bottom.

(3) According to the proposed prediction method, the fatigue life of the flexspline in the certain cap-type harmonic reducer is about 3296.7 hours, which is close to the fatigue life of harmonic reducer products in China and the actual fatigue life obtained during the product's usage. The validity of this prediction method is demonstrated.

REFERENCES

- Cheng, C., "Development of Short Cylinder Hat-type Harmonic Reducer and Life Testing System," Nanjing University of Information Science and Technology, China, (2024).
- Chen, S.J., "Study on Anti-fatigue Characteristics and Parameter Optimization of Flexspline in Harmonic Reducer," Guangdong University of Technology, China, (2021).
- Cheng, Y.B., Yuan, W.P., Zhang, T., and Liu, Z.F., "Research on fatigue life prediction of flexspline of harmonic reducer based on BP neural network," Modern Manufacturing Engineering, No. 4, pp. 140-145 (2024).
- Deng, X., "Mechanical Analysis and Fatigue Life Research of Harmonic Flexspline," Hunan University, China, (2018).
- Fu, Z.H., "Dynamic Simulation of Harmonic Drive," Xiamen University, China, (2017).
- Huang, Q.W., Wu, B.S., and Liao, D.L., "Life Prediction of Flexspline based on the Modified P-S-N Curve," Mechanical Transmission, Vol. 45, No. 11, pp. 161-165 (2021).
- Han, X.R., "Life Analysis And Parameter Optimization Of Harmonic Drive Flexspline In Glass Bottle Making Machine," Huazhong University of Science and Technology, China, (2019).
- Ling, S.S., "Conventional Fatigue Testing Methods and Data Processing, Lecture 5," Physical Testing, Physical Section, Vol. 23, No. 5, pp. 62-65 (1987).
- Lyuminarsky, I.E., Lyuminarsky, S.E., and Lyuminarskaya, E.S., "Increased Durability of Flexible Gear of Harmonic Drive Due to Flexible Bearing Modification," Proc., of International Conference on Industrial Engineering, Cham, D.C. (2019).
- Ni, K., Zhou, F., Lu, R.G., and Lv, M.J., "Motion Analysis Research of Harmonic Reducer Based on Abaqus," International Journal of Science, Vol. 10, No. 5, pp. 88-95 (2023).
- Wu, C., Tao, M.L., Chen, D.F., Wu, J.F., Shu, W.M.,

- Chen, F., and Zhang, H.M., "Analysis on Fatigue Life and Mechanical Property of Flexspline of Harmonic Reducer," *Mechanical Transmission*, Vol. 46, No. 1, pp. 115-119 (2022).
- Wang, P.S., Zhao, C., Li, J.Y., Zhou, Z., Yan, H., and Hu, J.C., "Meshing Characteristics and Stress Deformation Analysis on Harmonic Reducer," *Aerospace Science and Technology*, Vol. 34, No. 7, pp. 73-80 (2023).
- Wang, J.X., Hu, R.K., Zhou, G.W., Zhang, Z.H., and Pei, X., "Analysis on mixed lubrication at the conjugate meshing domain of harmonic gear with double-circular-arc tooth profile," *Journal of Harbin Institute of Technology*, Vol. 50, No. 7, pp. 169-176 (2018).
- Yang, H.X., Li, X.P., Xu, J.C., Guo, Y.J., and Li, B.T., "Modeling and Fatigue Characteristic Analysis of the Gear Flexspline of a Harmonic Reducer," *Mathematics*, Vol. 10, No. 6, p. 868 (2022).
- Yan, F., Yang, W., Duan, C.C., and Qiu, J.X., "Fatigue Life Analysis of Flexspline in Harmonic Drives," *Modern Manufacturing Engineering*, No. 10, pp. 17-19 (2013).
- Yi, W.F., "Finite Element Analysis and Structural Optimization of Harmonic Reducer," *Kunming University of Science and Technology, China*, (2023).
- Zhang, X., Jiang, G.D., Zhang, H., Yun, X.L., and Mei, X.S., "Time-dependent reliability analysis of harmonic drive based on transient FEA and accelerated life test," *Engineering Computations*, Vol. 37, No. 7, pp. 2293-2317 (2020).
- b — Flexspline tooth width (mm).
- d — Flexspline Mounting hole diameter (mm).
- D_1 — Flexspline outer diameter (mm).
- D_2 — Inside diameter of cylinder bottom flange of Flexspline (mm).
- D_3 — Flexspline inner diameter (mm).
- D_s — External diameter of the test specimen of Flexspline (mm).
- L — Flexspline cylinder length (mm).
- δ — Flexspline wall thickness (mm).
- t — Flexspline Bottom flange height (mm).
- r — Flexspline Round corners at the bottom of the cylinder (mm).
- r_s — Outer transition arc radius of the test specimen (mm).
- d_s — Measuring diameter of the test specimen (mm).
- L_s — Reduced length of the test specimen (mm).
- S — The maximum alternating stress (MPa).
- N — The fatigue life in cycles.
- R — Stress ratio of the alternating load.
- $P_{\max}$ — Upper limit of the alternating load (kN).
- $P_{\min}$ — Lower limit of the alternating load (kN).

NOMENCLATURE