Static Analysis of Gear Periodic Symmetrical Structure Based On ANSYS

Jing Ma [1, *], Wenhui Zhang [2, a], Shuaibo Ding [3, b], Dajian Yi [3, c], Maojun Wu [3, d], Dajian Yi [3, e], Rui Chen [3, f], Xiaochen Huang [3, g]

1 Nanjing Xiaozhuang University, Nanjing, China

[2] Nanjing Xiaozhuang University, Nanjing, China; State Key Lab. for Novel Software Technology, Nanjing University, Nanjing, China

[3] Nanjing Xiaozhuang University, Nanjing, China; Zhejiang sci-tech university, Hangzhou, China

* Corresponding Author Email: 715685317@qq.com, [a] hit_zwh@126.com, [b] 2637208989@qq.com, [c] 1176540684@qq.com, [d] 3141793829@qq.com, [e] 1176540684@qq.com, [f] chen1716707642@qq.com, [g] 1176540684@qq.com

Abstract. Gears are important components of transmission mechanisms, and it is crucial to analyze the stress and strain of gears using ANSYS to obtain their mechanical characteristics for product research and development in production. After establishing a mechanical model of gears based on material characteristics, define constraints and boundary critical conditions. Through analysis and solution, the deformation diagram and stress distribution diagram of the gear are obtained. Using the characteristic of periodic symmetry can simplify the analysis of the mechanical characteristics of gears, reduce the cycle time and cost of prototype development. Of course, this method can be applied to all similar institutional developments with this characteristic, which provides convenience for product design and improvement and saves cost.

Keywords: Static Analysis, gear, periodic symmetrical, Finite Element, ANSYS.

1. Introduction

If the result is that after rotating an angle around an axis, the structure is exactly the same as that before rotation, this structure is called periodic symmetrical structure [1]-[3], the minimum rotation angle meeting this condition is called rotation period, and the included angle arbitrarily taken out from the structure is α All parts of the structure can be called the basic sector of the structure. The complete structure can be obtained by rotating and copying N copies of the basic sector around its axis [4]-[5].

A gear is analyzed by using the periodic symmetry of the structure [6]. Using this method, the scale of price analysis can be reduced and the calculation cost can be saved. Characteristics of periodically symmetric structures: if the structure rotates around its axis α If the angle and structure (including material constants) are exactly the same as before rotation, this structure is called periodic symmetric structure (cyclic symmetric structure) [7].

For the gear with round hole and thick tooth edge, the force analysis is carried out. Standard gear, maximum speed 62.8rad/s. calculate the stress distribution.

Tooth tip diameter: 48mm
Tooth bottom diameter: 40mm
Number of teeth: 10
Elastic modulus: 2.06e11pa
Density: 7.8e3

2. Establish finite element model

2.1. Define structure type

To analyze a rotating gear, firstly, the element attribute is defined as brick 8node 185. first established the points, and connected the points as lines. Then turn the line into a face and finally stretch it into a body. Deformation diagram are obtained. The following is the specific process.

2.2. Define material performance parameters

Using the stress analysis module of ANSYS to analyze gears, it is necessary to define material properties such as elastic modules.

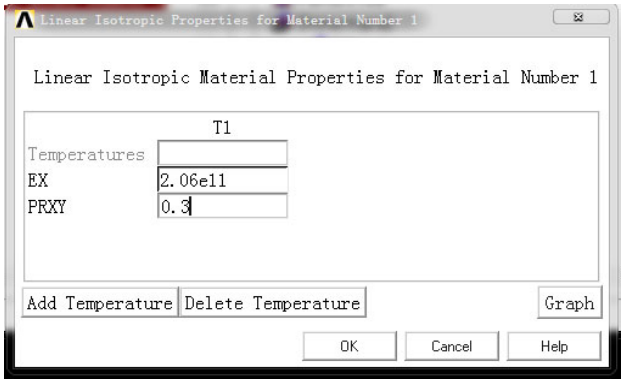


Fig. 1. Define material performance parameters

2.3. Established the sector model of gear

Select unities Menu, define Global cyclical. Command, Establish a key point. Select type 1, x = 15, y = 0 in the text box.

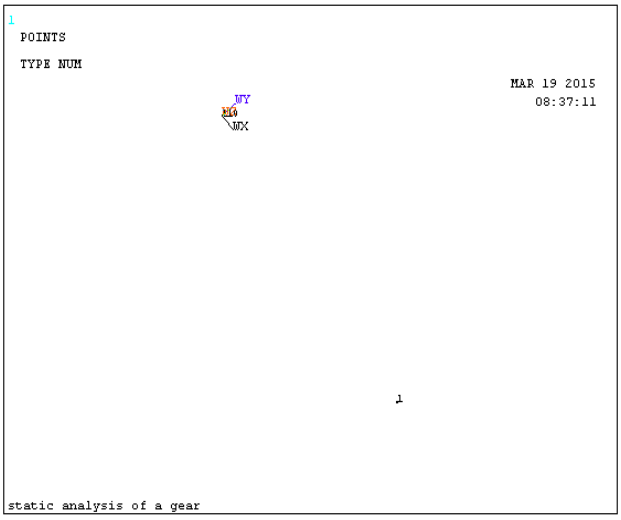


Fig. 2. Establish an auxiliary point

Establish an auxiliary point. Select type 110, x = 12.5, y = 40. Click OK As shown in the figure.

Offset the work plane to a given location. Unities Menu > working plane > offset WP to > key points command. Select point 110 in the drawing. Click OK.

Rotate the work plane. Unities Menu > working plane command, Select type "- 50, 0, zxangels" and click OK.

Select unities Menu > working plane > change active CS to >Working Plane. Command.

Establish the second key point. Select type 2 in the text box, x = 10.489, y = 0. Click OK.

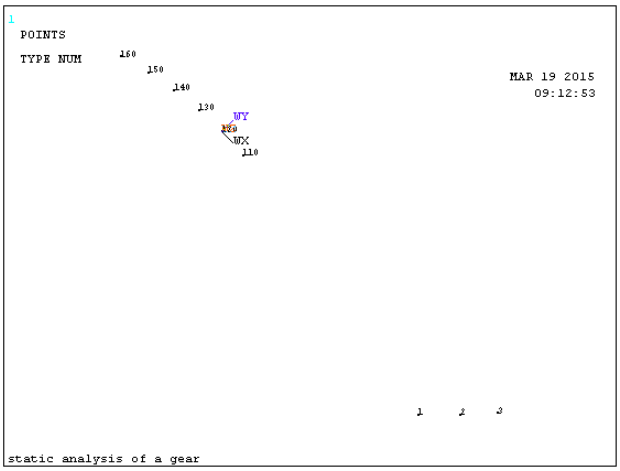


Fig. 3. Establish the second key point

Establish the third key point. Select type 4, x = 14.182, y = 0 in the text box. Click OK. Establish 4, 5, 6 and 7 key points. The method is the same.

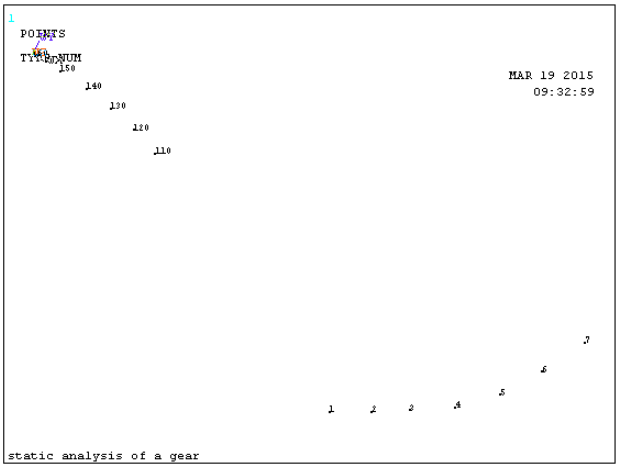


Fig. 4. Establish the third key point

Select unities Menu > working plane > change active CS to > Global cyclical. Command. Establish the 8th, 9th and 10th key points.

To create an arc line, select straight line command. Connect 10 and 1, 1 and 2, 2 and 3, 3 and 4, 4 and 5, 5 and 6, 6 and 7, 7 and 8, 8 and 9. Add the lines on the edge of the gear and select lines command

Rotate the work plane 9.87 degrees. Untities Menu > working plane and click OK.

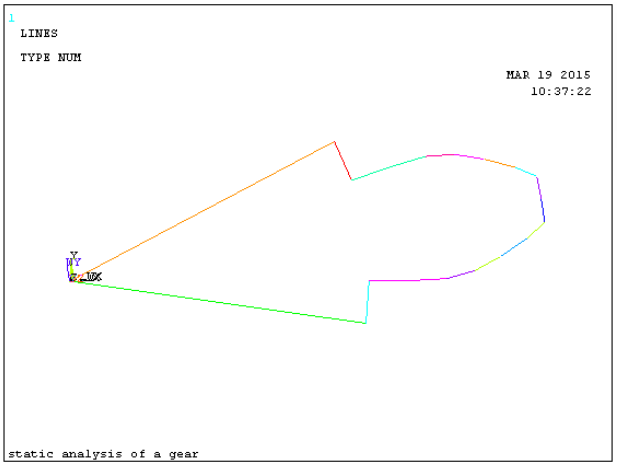


Fig. 5. Create an arc line

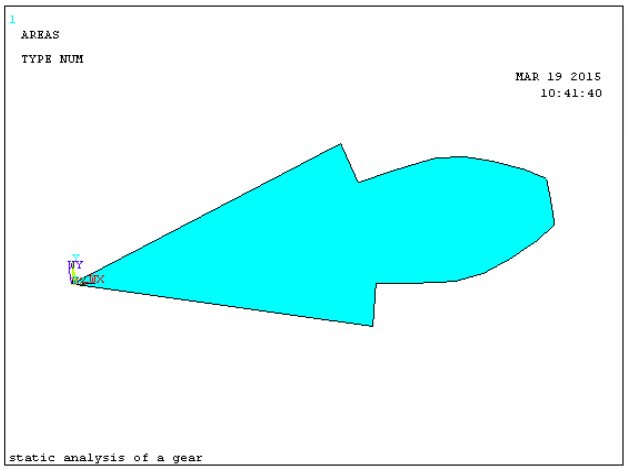


Fig. 6. Creates a line as a face

Creates a line. Extrude faces into bodies. Select the command. As shown in the figure, the results after stretching are obtained.

Create two cylinders. Select solid cylinder command.

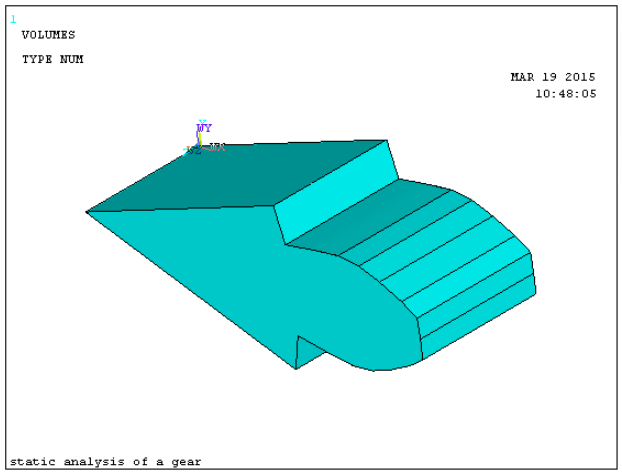


Fig. 7. Extrude faces into bodies

Subtract three cylinders from the gear body, Select the gear body, click apply, select two large cylinders, then select small .

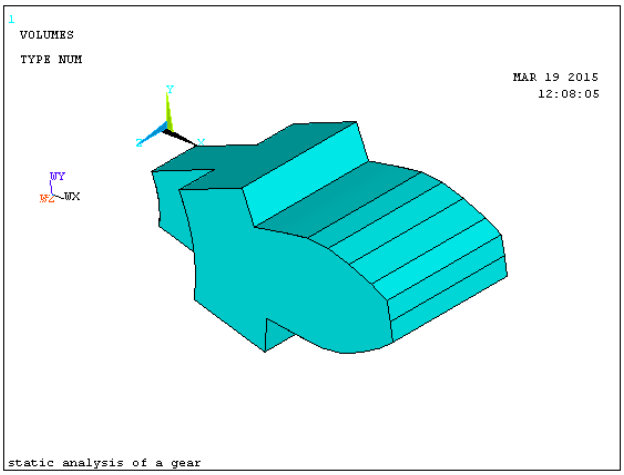


Fig. 8. Create two cylinders

Select unities Menu > working plane > change active CS to > Global cyclical.

Type 1000 in the text box, x = 7.5, y = - 5. Click OK. Offset the work plane, select the unities Menu > working plane > offset WP to > key points command, select the point, select untities Menu > working plane.

Create a cylinder. Subtract the gear body from the cylinder.

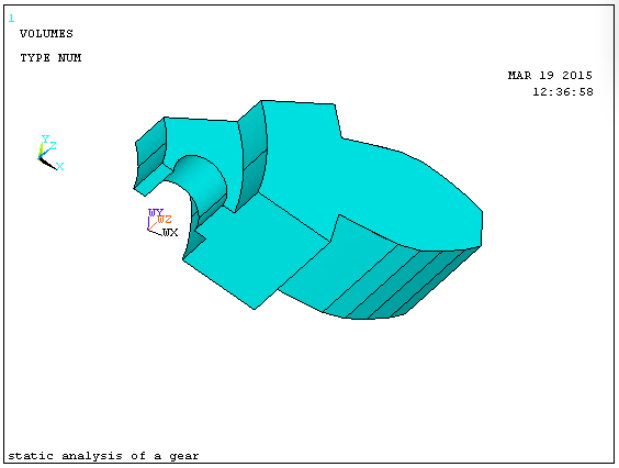


Fig. 9. Create a cylinder

Select activate smart size. Set it to 3 and click mesh. A dialog box pops up. Click picks all.

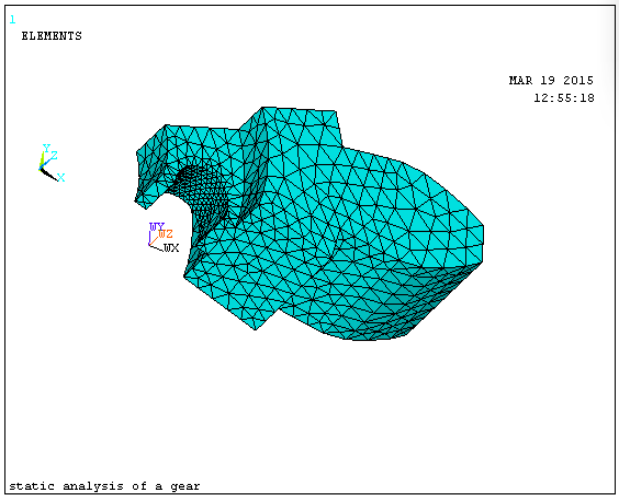


Fig. 10. Finite element mesh generation

3. Finite Element Solution and Analysis

3.1. Radial displacement analysis

It can be seen from the image above that the gear has radial displacement,

(1) From the figure, it can be seen that the maximum deformation is 0.829753.

(2) The maximum solution for node displacement is 0.711215.

(3) The minimum solution for node displacement is 0.450281.

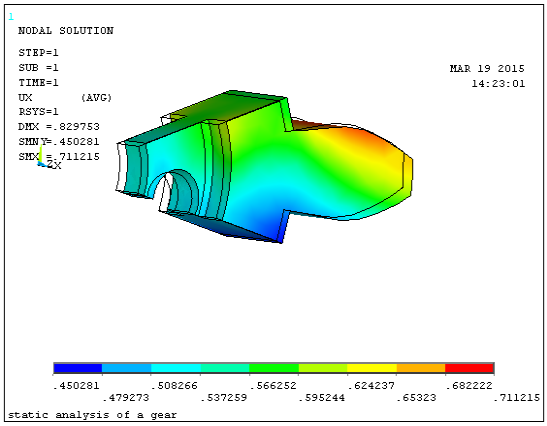


Fig. 11. Radial displacement diagram

3.2. Radial stress analysis

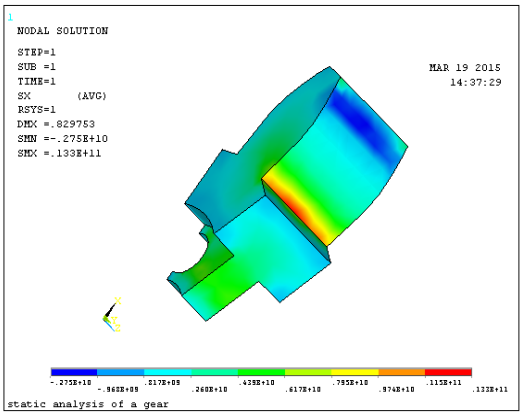


Fig. 12. Radial stress diagram

(1)The maximum radial stress can be seen from the figure as 0.829723;
(2)The maximum solution of node radial stress is 0.133e+11pa;
(3) The minimum solution for node radial stress is -0.275e+10Pa.

3.3. Equivalent stress local analysis

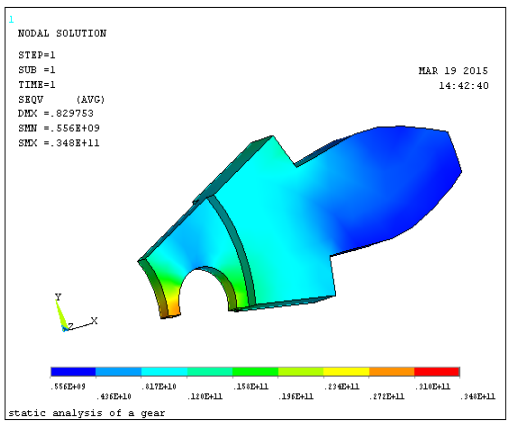


Fig. 13. Partial diagram of equivalent stress

(1)The maximum equivalent stress can be seen from the figure as 0.829753;
(2)The maximum solution of node equivalent stress is 0.348e + 11pa;
(3) The minimum solution for node equivalent stress is -0.556e + 9pa.

3.4. Equivalent stress global analysis

Select unities Menu > plotctrls > system > symmetric expansion > user specified expansion command. Type 10 in the "no. of repetition" text box.

Select local polar in type of expansion. In repeat pattern, select alternate symm. Type dy = 36 in DX, Dy, DZ increment. Click OK. Expanded results:

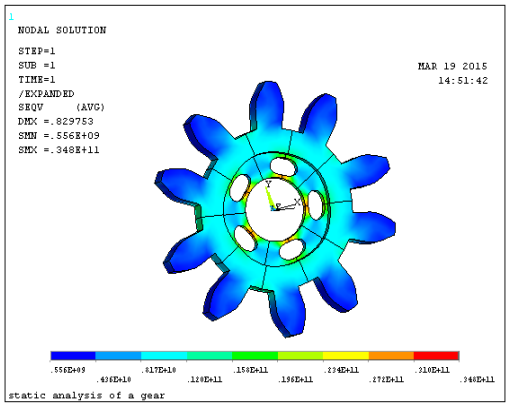


Fig. 14. Overall equivalent stress diagram

4. Conclusions

Based on ANSYS, analyze the mechanical characteristics of gears, establish a 3D model of the gears, use the stress analysis module, define boundaries and constraint conditions, Through analysis and solution, the deformation diagram and stress distribution diagram of the gear are obtained. Using the characteristic of periodic symmetry can simplify calculations for the target with this characteristic, which provides convenience for product design and improvement and saves cost.

Acknowledgments

Thank you to the reviewers of this paper, and to all the teachers and classmates who have provided assistance for the research on this topic.

References

[1] Yang Shuting. Modal analysis of motor support structure based on ANSYS. Internal combustion engine and accessories, 2019, (07): 72 -74.

[2] Zhao Haiming, Jiang Binbin, Li mi. Structural design of machine tool base based on topology optimization and multi-objective optimization. Mechanical design and research, 2018, 34 (04): 100-105.

[3] Vladimir I, Medvedev, Andrey E, et al. Mathematical Model and Algorithm for Contact Stress Analysis of Gears with Multi-Pair Contact. Mechanism and Machine Theory, 2015, 86:156-171.

[4] Tian Maojin, Huang Sheng. Application of American steel structure code in pile foundation design of high piled wharf. Port engineering technology, 2016, 53 (03): 52 -56.

[5] Sahoo P, Ghosh N. Finite elements contact analysis of fractal surfaces. Journal of Physics D Applied Physics, 2007, 40(14):42-45.

[6] Chatterjee B, Sahoo P. Finite Element based Parametric Study of Elastic-plastic Contact of Fractal Surfaces. Procedia Engineering, 2013, 64:552-561.

[7] Lin Senhai, Zhang Wenhui, Chen Rongchang, Jiang Jie, ye Xiaoping Prestress and modal analysis of high-speed rotating disc based on ANSYS Journal of Hubei Institute of engineering, 2016,36 (03): 84-86