

ERROR IDENTIFICATION AND COMPARISON WITH AGMA STANDARD IN GEARS USING FINITE ELEMENT ANALYSIS

Dr. N. Nandakumar¹, T. Allwin Raja²

¹Associate Professor, Department of mechanical Engineering, Government College of Technology, Coimbatore, Tamilnadu, India.

²PG Scholar, Department of Engineering Design, Government College of Technology Coimbatore, Tamilnadu, India

Abstract - The objectives of this thesis are to use a algebraical style to progress theoretic replicas of the behavior of helical gears and rack and pinion in mesh, to help to predict the effect of gear contact stresses and bending stress error. The core attention of the existing study as established to govern proper representations of contact rudiments, to analyze bending stresses and contact stress of helical gear and rack and pinion respectively by ANSYS and equate the outcomes with AGMA theoretic standards.

Key Words: Helical gear, rack and pinion, contact stress analysis, bending stress analysis

1. INTRODUCTION

Gearing is one of the most critical components in a mechanical power transmission system, and in most industrial rotating machinery. It is possible that gears will predominate as the most effective means of transmitting power in future machines due to their high degree of reliability and compactness. In addition, the rapid shift in the industry from heavy industries such as shipbuilding to industries such as automobile manufacture and office automation tools will necessitate a refined application of gear technology. The increasing demand for quiet power transmission in machines, vehicles, elevators and generators, has created a growing demand for a more precise analysis of the characteristics of gear systems. In the automobile industry, the largest manufacturer of gears, higher reliability and lighter weight gears are necessary as lighter automobiles continue to be in demand. In addition, the success in engine noise reduction promotes the production of quieter gear pairs for further noise reduction. Noise reduction in gear pairs is especially critical in the rapidly growing field of office-automation equipment as the office environment is adversely affected by noise, and machines are playing widening role in that environment.

The finite element method is very often used to analyze the stress state of an elastic body with complicated geometry, such as a gear. In this thesis, first, the finite element models and solution methods needed for the accurate calculation of two dimensional Rack and pinion contact stresses and Helical gear bending stresses were determined. Then, the contact and bending stresses calculated using ANSYS 19.2 were equated to the outcomes gained from present approaches. The purpose of this thesis is to develop a model

to learning and forecast the typical counting the connection stresses and the Bending stress of gears in mesh by the ANSYS 19.2 Tool package grounded on mathematical system. The aim is to find the amount of stress error in the gears, and thereby reduce the amount of failure rate generated.

2. CONTACT STRESS

2.1. IMPORTANCE OF CONTACT STRESS

Despite the importance of contact in the mechanics of solids and its engineering applications, contact effects are rarely seriously taken into account in conventional engineering analysis, because of the extreme complexity involved. Mechanical problems involving contacts are inherently nonlinear. Usually the loading causes significant changes in stiffness, which results in a structure that is nonlinear. Nonlinear structural behavior arises for a number of reasons, which can be reduced to three main categories (1) Geometric Nonlinearities, (2) Material Nonlinearities, (3) Change in Status Nonlinearities, Through the quick change of computational mechanism, but, excessive advancement has been finished in geometric study of the tricky. Using the finite element way, several connection glitches, oscillating after moderately humble ones to quite difficult ones, can be resolved with high precision. The Finite Element Method can be restrained the desired technique to treat interaction problems, since of its confirmed achievement in giving a extensive variety of engineering delinquent in zones of solid mechanics, fluid flow, heat transfer, and for electromagnetic field and coupled field problems.

2.2. CONTACT PROBLEM OF RACK AND PINION

The fig 1 shows Rack and pinion are hard-pressed counter to each other. This prototype was constructed grounded on the Hertz contact stress theoretic delinquent. The ranges were premeditated beginning from the pitch diameters of the pinion and gear. The contact stress of this prototype should characterize the contact stress among two gears. In the input file, first, the geometry of Rack and pinion must be described. Then the geometry areas were meshed. In contact areas a fine mesh was built. The boundary conditions were applied in this model. The loads also were applied. In each step there are a lot of sub-steps. In each sub-step the number of equilibrium iterations was set.

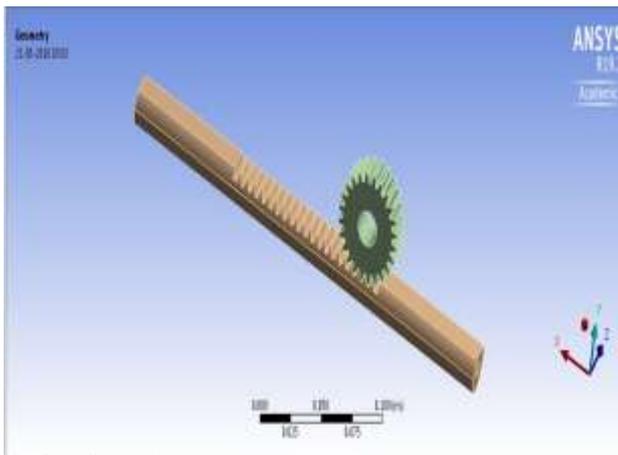


Fig.1. RACK AND PINION

the large computer memory space that is necessary. In the technique for producing a FEM typical for bending stress analyses, the calculations secondhand to produce the gear tooth shape curve. When meshing the teeth in ANSYS, if "SMART SIZE" is used the number of elements near the roots of the teeth are automatically much greater than in other places. It also indicates that only one tooth is enough for the bending stress analysis for the 3-D model.

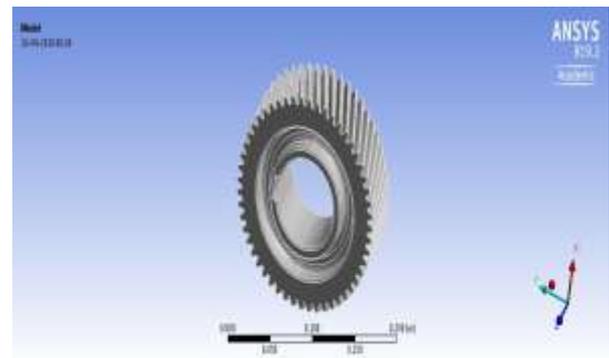


FIG.3.HELICAL GEAR

2.3 . CONTACT STRESS ANALYSIS

The fig 2 shows the usual connection stress laterally on the interaction parts. The similar process is ended for dissimilar materials and the outcomes are equated with AGMA ordinary outcome and discourse around the stress difference.

3.1. BENDING STRESS ANALYSIS

The Figure 4 displays how to mesh the 3D model and how to place happening the load on the sample. Here are internal side nodes on the individually side of apiece part. So a great number of degrees of free movement in this 3D model take a lengthier period to enclose running.It similarly specifies that only one tooth is sufficient for the bending stress study for the 3-D model. The investigation displays one tooth FEM prototype and display how abundant Von Mises stress is on the base of tooth when the number of teeth is 37 for the gear.

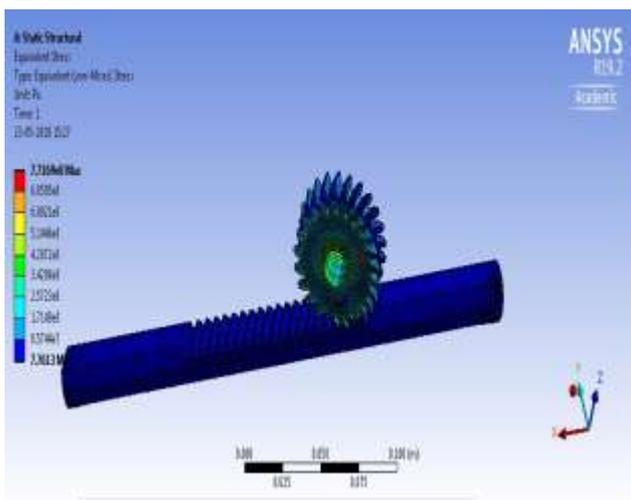


FIG.2.CONTACT STRESS ANALYSIS

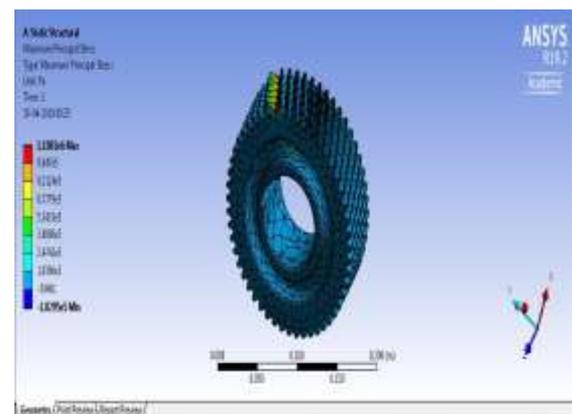


FIG.4.BENDING STRESS ANALYSIS

3. BENDING STRESS

When one investigates actual gears in service, the conditions of the surface and bending failure are two of the most important features to be considered. The finite element method is very often used to analyze the stress states of elastic bodies with complicated geometries, such as gears. There are published papers, which have calculated the elastic stress distributions in gears. In these works, various calculation methods for the analysis of elastic contact problems have been presented. The finite element method for two-dimensional analysis is used very often. It is essential to use a three-dimensional analysis if gear pairs are under partial and non uniform contact. However, in the three-dimensional calculation, a problem is created due to

In this unit the tooth base stresses and the tooth deflection of one tooth of helical gear is designed using an ANSYS model.

4. AGMA ANALYSIS

In this section, a comparison of the tooth root stresses obtained in the three dimensional model and in the two

dimensional model using ANSYS with the results given by the standards of the AGMA is carried out. is recommended by the AGMA and the other coefficients, such as the dynamic factor, are set at 1.2.

$$\sigma_c = Z_E \sqrt{W^t K_o K_v K_s \frac{K_H Z_R}{d_{w1} b Z_1}}$$

$$\sigma_F = F_t \frac{Y_F Y_\epsilon Y_\beta}{m_n b} \left(\frac{K_v K_o}{K_L K_{FX}} \right) S_F$$

Where $\sigma(c)$ and $\sigma(f)$ are contact and bending stress respectively.

5. BENDING STRESS COMPARISON

The Von Mises stresses on the root of tooth were carried out in order to know if they match the results from ANSYS.

The above calculations of the Von Mises stresses on the root of tooth were carried out in order to know if they match the results from ANSYS In this Table 1 the maximum values of the tooth root stress obtained by the ANSYS method.

TABLE.1.BENDING STRESS ANALYSIS

NUMBER OF TEETH	AGMA (MPa)	ANSYS (MPa)	% ERROR
28	650.43	671.24	3.2
30	670.32	690.43	3
34	732.65	757.12	3.34
37	745.59	771.69	3.5
40	765.76	797.92	4.2
42	788.77	820.32	4

5.2. CONTACT STRESS COMPARISON

TABLE.2.CONTACT STRESS ANALYSIS

NUMBER OF TEETH	AGMA (MPa)	ANSYS (MPa)	% ERROR
33	85.050	84.429	0.74
35	91.129	91.770	0.69
37	115.126	110.78	3.97
40	116.86	113.79	2.63
42	128.86	124.80	2.93
44	141.97	132.15	7.43

These changes are supposed to be produced by aspects such as the mesh design and the limited circumstances on the finite element analysis, and the supposed location of the dangerous unit in the values are shown in Table 2.

6. CONCLUSION

It was exposed that an FEA prototype could be helped to analyze the interaction among two physiques exactly by confirmation of contact stresses among two cylinders in contact and comparison with the Hertzian equations. Active approaches to approximation the tooth interaction stress using a 3D interaction stress prototype and to guess the root bending stresses via 3D FEA prototype are anticipated. The expansion of a innovative mathematical way for FEA showing of the full gear figure which can alternate in mesh together with the contact problem is offered. The motionless stress fault was too attained after running the replicas in ANSYS.

REFERENCES

- [1] Norton, R. L., "Machine Design: An Integrated Approach", New Jersey: Prentice-Hall Inc.
- [2] Hamrock, B. J., Jacobson, S. R., "Fundamentals of Machine Elements".
- [3] Buckingham, E., 1949, "Analytical Mechanics of Gears", McGraw-Hill, New York.
- [4] Coy, J. J., Chao, C. H. S., 1982, "A method of selecting grid size to account for Hertz deformation in finite element analysis of spur gears", Trans. ASME, J. Mech. Design 104 759-766.
- [5] Gatcombe, E.K., Prowell, R.W., 1960, "Rocket motor gear tooth analysis", (Hertzian contact stresses and times) Trans. ASMA, J. Engng Industry.
- [6] Umezawa, K., 1988, "Recent Trends in Gearing Technology", JSME International Journal Series III Vol.31, No. 2, pp 357-362.
- [7] Klenz, S. R., 1999, "Finite Element Analyses of A Spur Gear Set", M.Sc. Thesis, Dept. of Mechanical Engineering, University of Saskatchewan.
- [8] Smith, J. O. Liu, C. K., "Stresses Due to Tangential and Normal Loads on an Elastic Solid with Applications to Some Contact Stress Problems", *Journal of Applied Mechanics*
- [9] Wang, J., 2003, "Survey of Nonlinear Vibration of Gear Transmission Systems" *Appl Mech Rev* vol 56, No 3.
- [10] Chong, T. H., Bar, I., 2001, "Multiobjective optimal Design of Cylindrical Gear Pairs for the Reduction of Gear Size

and Meshing Vibration”, JSME International Journal, Vol. 44, No. 1, pp 291-292

- [11] B.Bloomfield, Designing facegears, Mach.Design19(1947) 129134.
- [12] G.F. Heath, R.B. BosslerJr., Advanced rotorcraft transmission (ART) program final report, NASA Contractor Report 191057, January 1993.
- [13] G.F.Heath,R.R.Filler,J.Tan,Developmentoffacegear technologyforindustrialandaerospacepowertransmission,NASA/CR2002211320,May2002.
- [14] R.R.Filler, G.F.Heath, S.C.Slaughter, D.G.Lewicki, Torque Splitting By a Concentric Face Gear Transmission, the American Helicopter Society 58th Annual Forum, Montreal, 2002June1113.
- [15] G.F. Heath, S.C. Slaughter, M.T.Morris, J.Fetty, D.G. Lewicki, D.J. Fisher, Face Gear Development under the Rotor craft Drive System for the 21st Century Program, the American Helicopter Society 65th Annual Forum, 2009 May 27-29.