IMPROVING BENDING LOAD CAPACITY OF SPUR GEARS WITH INCREASING ROOT RADIUS

B.Sc.[1] Kalay O.C., M.Sc.[1] Doğan O., Ass. Prof. Karpat F., PhD.[1]
Department of Mechanical Engineering – Uludağ University, Turkey[1]

onurcankalay@uludag.edu.tr

Abstract: *Gears are one of the most important machine elements in the industry. They are using many areas such as; automotive, energy, aviation, etc. Gears are exposed to higher loads day by day due to the increase in power and speed on the machines. Therefore, the stress values which are occur on the gear root is also increase. These stresses cause to damage on the teeth root. Thus the stress values have to be decrease to design optimum gear body. In this study, the effects of root radius on the gear bending stress are evaluated by using finite element method. At first, gears with standard root radius is investigated both DIN 3990 and finite element method. After the validation of finite element model, the root radius of the gear is taken as parameter. Gears with different root radiuses are analyzed by using finite element method. It is seen that with the increasement of gear tooth radiuses, the root stress is considerably decreased. With increasement of root radiuses the maximum principal stress reduced nearly 20%.*

Keywords: SPUR GEARS, BENDING STRESS, LOAD CAPACITY, FINITE ELEMENT METHOD

1. Introduction

Nowadays gears have a widespread using areas such as automotive, energy, aviation etc. In these areas the demanding power and speed values increase so; gears are exposed to higher loads day by day. Consequently with increasing power and speed on machines, gear designs need to keep up with today's requirements. During the operation of gears each tooth is affected from static and dynamic loads. In lower rotation speeds, in proportion to transmitted torque, static loads are exposed on each tooth. With increasing of rotation speeds, dynamic loads begin to be effective. For longer fatigue life, quiet and safety operation of gears, dynamic loads need to be reduced. Bending stresses that occur at tooth root are one of the preventive factor for optimum gear design. Awareness of these stresses, especially in design stage, is also important for taking precautions to gear damage and improvement load carrying capacity [1]. Due to developments in engineering technologies gear's durability and load carrying capacity also can be modified with different ways such as change of root radius for reducing these stresses.

Widely using of gears in industry brings along various numerical and experimental methods. Numerical methods that enable to investigate effects of many parameters satisfactorily in small time periods are become dominated for gear damage analyses [1]. In this context many studies have been made in the literature; Ural et al. studied on the stress at the profile of tooth in cylindrical spur gears with ANSYS computer code and developed a model with 3 DOF using 3D solid 186 element. But the study contains only constant values of pressure angle number of teeth, modulus etc. [2]. Fetvaci and Imrak created a FEM model of a spur gear for examining tooth root stress. A macro code that developed by Fetvaci has used for applying tooth forces to model respectively in some points of engagement cycle. After their model's analysis with ANSYS, they compared the results with literature [1]. Karpat et al. investigated gears with asymmetric teeth. They developed a model which used for the modeling of tooth via finite element model and examined the root bending stress [3]. Fetvaci and Imrak proposed a new finite element gear mesh pattern which is suitable for investigations of bending stress analysis of spur gears during the engagement cycle [4]. Hasl et al. studied on a calculation method for tooth root stress of plastic gears and represented the results of calculated contact ratios and tooth root stresses with graphics [5]. Fetvaci and Imrak gave information about things to pay attention for modeling involute spur gears. Investigated stress accumulation that triggers the fatigue crack in tooth root due to loading conditions. A novel finite element method consisting of quadratic elements in the second order is also developed. The results of tension stresses in number of teeth between 20 and 40 are compared with values in the literature [6]. Wei analyzed the characteristics of involute spur gears by using finite element method. The contact stresses were examined using two-dimensional (2D) FEM models and the bending stresses in tooth root were examined using a three-dimensional (3D) FEM model [7]. Rajaprabakaran and Ashokraj studied on finite element model of spur gear with a segment of three teeth for investigating stress concentration [8].

There are many methods for investigating root bending stresses such as the finite element method which is one of the popular method but it is more reasonable that support the FEM with experimental and analytical methods for obtaining results with a better accuracy. In this context; Shaker, in his master thesis, studied on optimization of tooth – root profile for maximum load carrying capacity for spur and bevel gears and developed a novel approach to design tooth – root profile of spur and bevel gears for meeting industry's demand. Therefore, he used FEA and experimental method together [9]. Lisle et al. compared root bending stress according to ISO (The International Organisation of Standardisation) 6336:2006, AGMA (American Gear Manufacturers Association) 2101 – D04, ANSYS finite element analysis and strain gauge techniques. They compared the root bending stresses values of ISO-AGMA-FEA and FEA-strain gauge with graphics and evaluated the results [10].

In this study the effects of root radius on the gear bending stress is investigated by using finite element method. At first for the gears with standard root radius is investigated both finite element method and DIN 3990 standard. After the validation, gears which have various root radiuses are designed and analyzed. Features of gear model and mesh properties are given with tables. After creating FEA model the effect of tooth radius values on bending stress is investigated and the results are discussed.

2. *Material and Method*

There are various methods have been developed for the calculation of gear bending stress. These methods can be classified as analytical methods, numerical methods and experimental methods. ISO 6336 and DIN 3990 are the most well-known methods and standards in the literature. These standards are quite similar to each other and calculations are made according to the following assumptions.

- The critical section of the tooth is the thickness of the tangent to the root of the tooth, making an angle of 30 ° starting from the symmetry of the tooth
- Compression stress which is caused by the radial component of normal force on the gear can be neglected.
- The tooth load is considered to influenced by the addendum circle in DIN 3990 / Method C and ISO 6336 / TC 60 Method C.

According to DIN 3990, the maximum tooth bending stress in the spur gears is calculated according to Eq.(1).

$$\sigma_{F0} = \frac{F_t}{b\,.\,m_n} Y_F\,.\,Y_s\,.\,Y_\varepsilon\,.\,Y_\beta \qquad [1]$$

Where F_t is the tangential force, b is tooth width, m_n is normal module;

Y_F is form factor;

$$Y_F = \frac{6\,.\,(h_{F\alpha}/m_n)}{(S_{Fn}/m_n)^2} \frac{\cos\alpha_{F\alpha n}}{\cos\alpha_n} \qquad [2]$$

Y_S is stress correction factor;

$$Y_S = \left(1{,}2 + 1{,}3\,.\frac{S_{Fn}}{h_{F\alpha}}\right)\left(\frac{S_{Fn}}{2\,.\,\rho_f}\right)^{1/[1{,}21+2{,}3(h_{F\alpha}/S_{Fn})]} \qquad [3]$$

Y_ε load sharing factor;

$$Y_\varepsilon = 0{,}25 + \frac{0{,}75}{\varepsilon_\alpha} \qquad [4]$$

Y_β is helix factor, in this study the gear type is defined as spur gear thus the Y_β can be considered as ‘’1’’ also the finite element analyses is conducted by using single tooth model so the load sharing factor is also defined as, ‘’1’’.

In this study, the effects of root radius on the gear bending stress are evaluated by using finite element method (FEM). Also for validation, DIN 3990 standards have taken into account. After the validation of finite element model, the root radius of the gear is taken as parameter. Gears with different root radiuses are analyzed by using finite element method. ANSYS static structural module is selected for the finite element model creation.

Table 1. *Features of FEA gear models*

Material	Steel
Young Module (N/mm^2)	210000
Poisson Ratio	0.3

Gear material is defined as standard steel whose properties are given in Table.1. The gear material is defined as an isotropic due to the low stress levels.

The gear geometry is defined as a three tooth model. The whole gear is not preferred because of the calculation time. The mesh structure is consisted of 44353 hexahedral elements and 179875 nodes. The mesh properties are given in Table 2.

Table 2. *Mesh properties*

Mesh Structure	Hexahedral
Number of Element	44353
Number of Nodes	179875
Mesh Type	Solid187

The boundary conditions and the mesh structure of the finite element model is given in Fig 1. The model is fixed on both sides and at the bottom. The static load is applied on the tip of the gear.

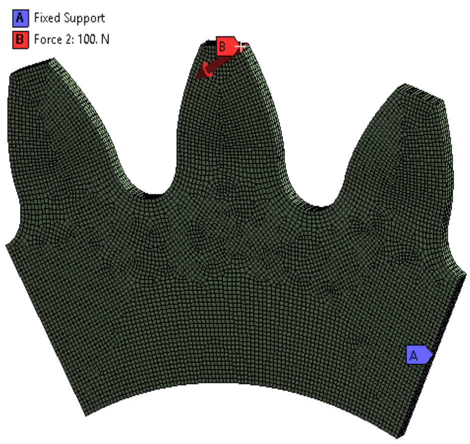


Fig 1. *Finite element model*

The pressure angle takes different values on the addendum circle and pitch circle. The standard pressure angle for the spur gear is 20°. The pressure angle is determined according to the following equation on the addendum circle.

$$r_0 * \cos\alpha\, r_{(i)} * \cos\alpha_{(i)} \qquad [5]$$

where r_0: radius of pitch circle, α=20° pressure angle on the pitch circle, $r_{(a)}$: addendum circle on the gear, $\alpha_{(a)}$: pressure angle on addendum circle.

Different root radius values are selected as a design parameter to reduce gear bending stress as ρ = 0,1*m, 0,2*m, 0,3*m, 0,375*m (standard) and 0,47*m respectively. The gears, which are used in this study, properties are given in Table 3.

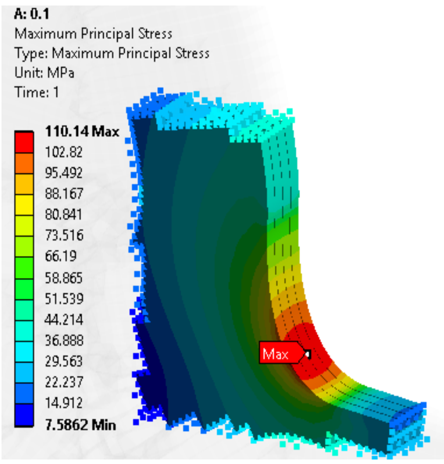
Table 3. *Gear properties*

Module (mm)	**4**
Tooth number	20
Pressure angle (°)	20
Face width (mm)	1
Normal force (N)	100
Root radius	0.1*m , 0.2*m, 0.3*m, 0.375*m 0.47*m

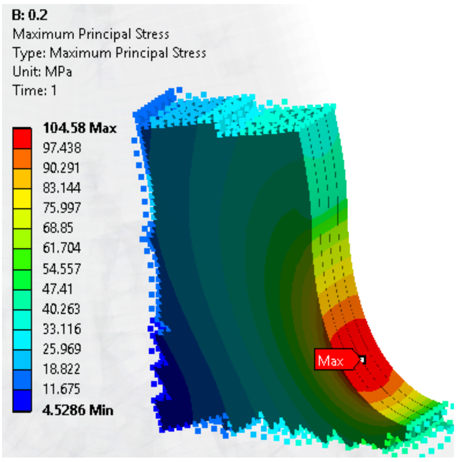
3. Results and Discussion

The maximum principle stress results are evaluated after the finite element analysis calculation. The maximum stress is seen at the root region of the tooth as in in Fig 2. At first gear which has standard root radius is investigated by using both finite element method and DIN 3990. According to the DIN 3990 form factor is defined as Y_F =2.87, stress correction is defined as Y_S= 1.55, Y_ε and Y_β can be taken 1. Thus, the maximum stress is defined as 95,004 MPa by using DIN 3990. The finite element model result can be seen in Fig 2 at 94.141 MPa. The differences between two methods can be negligible so, the finite element model is validated for the standard spur gears.

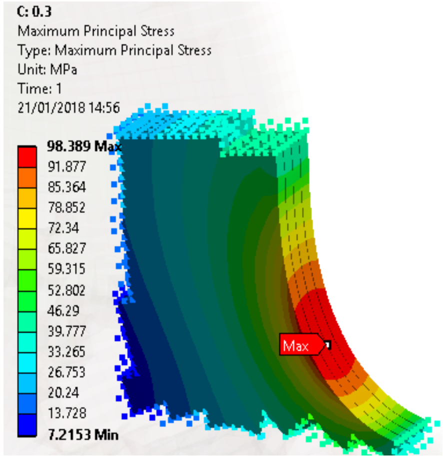
After the verification process of the finite element model the root radiuses of the spur gears are varied from 0.1*m to 0.47*m. The results can be also seen in Fig 2.



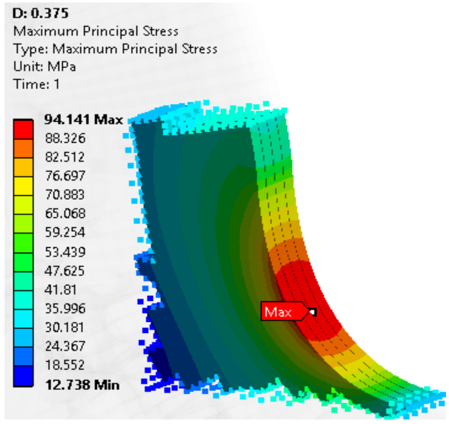
ρ=0,1*m



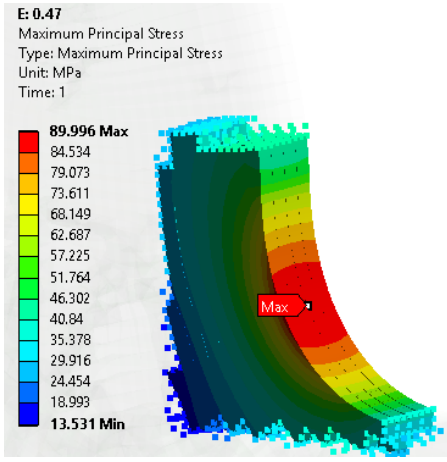
ρ=0,2*m



ρ=0,3*m



ρ=0,37*m (standard)



ρ=0,47*m

Fig 2. *Results of FEA, maximum principal stresses for different root radiuses*

The maximum stress is seen when the root radius is the minimum values at 0.1*m. The maximum stress is 110 MPa, the minimum stress is occurred, when the root radius is maximum value at 0.47*m. The minimum stress value is 89 MPa for the maximum root radius.

The effect of root radius on the maximum bending stress is seen in Fig. 3. It is clearly seen that, when the root radius increase the maximum bending stress which is occurred in the root region is considerably decreased. Nearly 20% of stress decreasement is achieved only the increasement of the root radius.

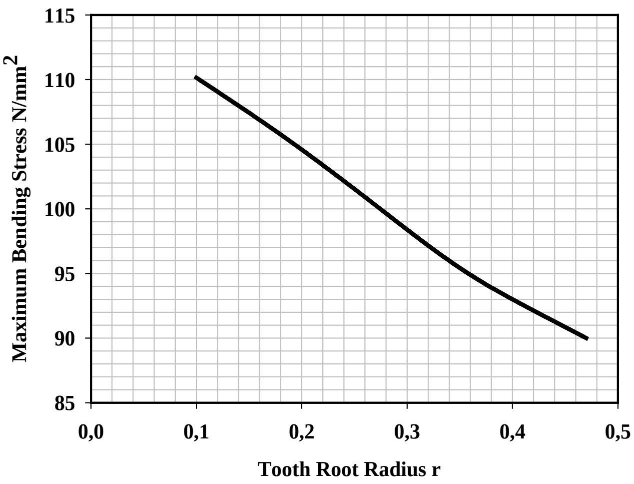


Fig 3. *Effect of tooth root radius on bending stress*

4. Conclusion

Most of gears give failure due to the high bending stress. Therefore, in the design phase of the gears, the bending stress should ne minimize. Increasing, gear root radius is a powerful way to decrease gear bending stress. To see the effect of gear root radius on the gear bending stress finite element model is created. Firstly gear with standard radius is analyzed and validated by using DIN 3990 analytically. After that the root radius values varied form ρ = 0,1*m, 0,2*m, 0,3*m, 0,375*m (standard) to 0,47*m respectively and maximum bending stresses are calculated by using finite element method. With increasement of root radiuses the maximum principal stress reduced nearly 20%.

5. References

[1] Fetvacı M.C., İmrak C.E., Dişdibi Gerilmelerinin Analizi için Düz Dişli Çarkların Sonlu Eleman Modellenmesi, Gazi Üniversitesi Mühendislik ve Mimarlık Fakültesi Dergisi, 2004, Cilt 19, No 2, P. 199-203.

[2] Ural F., Özsoy M., Uçar V., Düz Dişlilerde Dişdibi Gerilmelerinin İncelenmesi, SAU Fen Bilimleri Enstitüsü Dergisi, 2003, Cilt 7, Sayı 2, P. 100-103.

[3] Karpat F., Çavdar K., Babalik F.C., Asimetrik Evolvent Düz Dişlilerin Bilgisayar Destekli Analizi, Uludağ Üniversitesi Mühendislik-Mimarlık Fakültesi Dergisi, 2004, Cilt 9, Sayı 1, P. 123-131.

[4] Fetvacı M.C., İmrak C.E., Düz Dişlilerde Kavrama Çevriminde Gerilmelerin Değişiminin Sonlu Eleman Metodu ile İncelenmesi, Mühendis ve Makine, 2005, Cilt 46, Sayı 545, P. 19-25.

[5] Hasl C., Liu H., Oster P., Tobie T., Stahl K., Method for Calculating the Tooth Root Stress of Plastic Spur Gears Meshing with Steel Gears Under Consideration of Deflection-Induced Load Sharing, Mechanism and Machine Theory, 2017, 111, P. 152-163.

[6] Fetvacı M.C., İmrak C.E., Düz Dişli Çarkların Sonlu Elemanlar Metodu ile Modellenmesi, Mühendislik ve Fen Bilimleri Dergisi, 2004, P. 46-54.

[7] Wei Z.,Stresses and Deformations in Involute Spur Gears By Finite Element Method, Master Thesis, University of Saskatchewan, 2004.

[8] Rajabrabakaran V., Ashokraj R., Spur Gear Tooth Stress Analysis and Stress Reduction, Journal of Mechanical and Civil Engineering, P. 38-48.

[9] Shaker M., Optimization of Tooth-root Profile for Maximum Load-carrying Capacity: Spur and Bevel Gears, Master Thesis, McGill University, 2014.

[10] Lisle T.J., Shaw B.A., Frazer R.C., External Spur Gear Root Bending Stress: A Comparison of ISO 6336:2006, AGMA 2101-D04, ANSYS Finite Element Analysis and Strain Gauge Techniques, Mechanism and Machine Theory, 2017, 111, P. 1-9.