MATEC Web of Conferences 26, 03001 (2015)
DOI: 10.1051/matecconf/20152603001

Spur Gears Static and Dynamic Meshing Simulation and Tooth Stress Calculation

Ali Jammal[1,a], Hui Wang[1,b], Yiming Rong[1, 2,c]

[1]Beijing Key Lab of Precision/Ultra-precision Manufacturing Equipments and Control, Department of Mechanical Engineering, Tsinghua University, Beijing, 100084,China

[2]Department of Mechanical Engineering, Worcester Polytechnic Institute, Worcester, 01609, USA

Abstract. Gear meshing is a complicated process, and is subjected to the simulation process in the following paper. A flexible quasi-static and dynamic finite element analysis (FEA) models were built, to calculate contact principal and shear stresses. Full sized 3D spur gears are simulated under different boundary conditions. The first model, was a quasi-static analysis, where torque was used as input; and the second model, which was transient dynamic analysis, where rotational speed was used as input. The static analysis showed high stress concentration at the tooth contact point and under the contacting surface. The dynamic analysis provided the highest stress value at the different stages of gear engagement points along the line of action. Analytical and simulation result were in agreement in general, and the use of the new simulation model was discussed.

1 Introduction

Gears are the main component in rotating machinery, they are mainly used in different kind of gearboxes, which are employed mainly in automobile industry, also in wind energy for supplying wind turbines gearboxes. Gears are a critical part, which will greatly affect the functionality of the system. The meshing process of gears is complex, and is difficult to be described analytically, the dynamic loading will affect the stress distribution especially on the point of contact and the tooth root. FEA is employed to monitor dynamic loading, tooth profile and meshing parameter are optimized and then put under testing using the finite element analysis, the main aim is to reduce the stresses, leading to reduced fatigue subjected to the teeth and a better overall system reliability. A full gear 3D model is employed, in order to provide a more reliable results than a partial gear or one tooth model. In this work, 3D gears models were built, a pinion and a gear, having respectively 25 and 31 teeth, the module was equal to 8, and the tooth thickness was equal to 2 cm. the center distance was equal to 22.4 cm, and an initial clearance was given. Furthermore, hardened steel gear material model is used for the static and dynamic FEA analysis.

Gear contact stress is a main cause of fatigue and eventual failure, mainly the contact area will be subjected to a high stress concentration, which will cause fatigue, and the tooth will break under a stress lower than it could endure when put into service. Tooth geometry and design parameter will affect the stress distribution and concentration. The parameters sensitivity will affect mainly noise and vibrations, and contact patterns. When the gears are given a high clearance value more vibrations will surface, however if the clearance or backlash are not regulated properly, and their values are small, the gravity will affect the gear meshing, especially for gears in a wind turbine, because of the big size, and the tooth will come in contact in two points, causing higher contact stress. The tooth stress distribution is an area of interest, because, providing the ability of a fast and reliable calculation, tooth profile modification and parameter optimization will be enabled, hence reducing maximum stress on selected areas.

In Wang's [1] study, FEA result was obtained by using adaptive re-mesh with contacts using quad and brick elements. The elastic strains of the gear-shaft system were investigated in the high order end. The result showed the potential for using strains to monitor or control the transmission system. Fernandez et al. [2] presented an advanced model for the analysis of contact forces and deformations for spur gear meshing. A numerical example was presented where the quasi-static behavior of a single stage spur gear transmission is discussed. Wang and Howard [3] analyzed torsional stiffness of a pair of involute spur gears in mesh using finite element method, adaptive meshing was employed within a finite element program to reveal the detailed behavior alteration over region from single to double tooth contact zones and vice versa. Vijaya et al. [4] used a lumped-parameter mathematical and a finite element models, with different mesh modeling assumptions to analyze the nonlinear dynamics of planetary gears. The dynamics of planetary gears show nonlinear contact

[a] Corresponding author: jammal.ali@live.com; [b] wanghuisx@tsinghua.edu.cn; [c] yrong@mail.tsinghua.edu.cn

behavior. Responses from the dynamic analysis using analytical and finite element models were compared. The result showed that the analytical and finite element model's responses were in agreement for different planetary gear configurations. Topakci et al. [5] used parametric design software to build a 3D transmission gear train model, and simulated the structural stress distribution. The simulation analysis were static and linear, using Cosmosworks finite element software.

The followed approach, when trying to calculate the stress distribution, is accomplished mostly by employing quasi static analysis as shown in literature review above, because of the complexity of the calculation, as the gears are statically put into contact then the analysis is executed, further the angular positions are changed manually before each simulation. The following approach is followed in this paper, however a step forward has been achieved, where a transient analysis is executed, providing continuous and realistic results when compared to the real gear model.

2 Contact stress formulation

Having curved surfaces, when gears are in action, stresses will develop on the contact areas, due to sliding and rolling actions, since the bodies are pressed together while having a line contact. Hertezian stress theory is applied in the following case, where this stress is localized around the contact area, will cause elastic and in some cases plastic deformation, depending on the material properties and loading conditions. At the following contact area, micro cracks will appear, which eventually will become full sized cracks, as well as pits and flanks on the surface on the gears tooth. In order to quantify the occurring stress, the bodies coming to contact having each a radius of curvature, are considered as two special case of contacting cylinders, the stress results presented are known as Hertezian stress [6].

The two teeth considered as cylinders are presented in Fig.1, having the diameters d_1 and d_2, are pressed together as a result of the applied force F. The circular area had a radius equal to a. After specifying the material properties, consisting of the elastic constants Young's Modulus and the Poisson's Ratio E_1, v_1 and E_2, v_2, the radius a can be calculated as follows:

$$a = \sqrt{\frac{2F}{\pi l}\frac{(1-v_1^2)/E_1+(1-v_2^2)/E_2}{1/d_1+1/d_2}}. \quad (1)$$

The resulting pressure due to pressing is hemispherical in each cylinder, as the highest pressure calculated will be at the contact area and is:

$$p_{max} = \frac{2F}{\pi a l}. \quad (2)$$

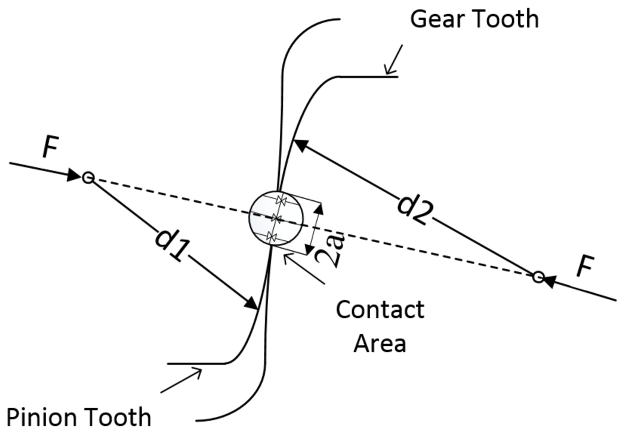

Figure 1. Hertz theory schematic applied to two gear teeth in contact

The maximum stress occur along the Y-axis, the principal stresses are presented bellow as follows:

$$\sigma_x = -P_{max}\left[\frac{1+2\frac{y^2}{a^2}}{\sqrt{1+\frac{y^2}{a^2}}} - 2\left|\frac{y}{a}\right|\right] \quad . \quad (3)$$

$$\sigma_y = \frac{-p_{max}}{\sqrt{1+\frac{y^2}{a^2}}} \quad . \quad (4)$$

$$\sigma_z = -2\nu P_{max}\left[\sqrt{1+\frac{y^2}{a^2}} - \left|\frac{y}{a}\right|\right] \quad . \quad (5)$$

Since the stress state off the y-axis is the highest, the other stress x and z-axis are only considered to calculate shear stress with respect to the depth value a (see Eq.1) as shown in the formulation below:

$$\tau_{max} = \frac{\sigma_z - \sigma_y}{2}\ (0\ to\ 0.436a)\ and\ \tau_{max} = \frac{\sigma_x - \sigma_y}{2}(> 0.436a) \quad . \quad (6)$$

3 Results

For static analysis, an input torque equal to 23000 N.mm was applied to the pinion, and using Eq.3, Eq.4, Eq.5 and Eq.6 the contact stress at and under the contacting surface of the gear tooth was calculated using numerical computation package MATLAB. The set of results presented below (Fig.2) are the maximum principal stress along the y direction as well as the maximum shearing stress. The maximum principle stress was equal to 300 MPa at the contact point and decreased to almost 50 MPa for a depth equal to 2.5 mm under the tooth contacting surface. As for shear stress, and at the contact area, was equal to 60 MPa and increased to hit is maximum value equal to 90 MPa, then the value decreased to reach 20 MPa at a depth equal to 2.5 mm under the contacting surface.

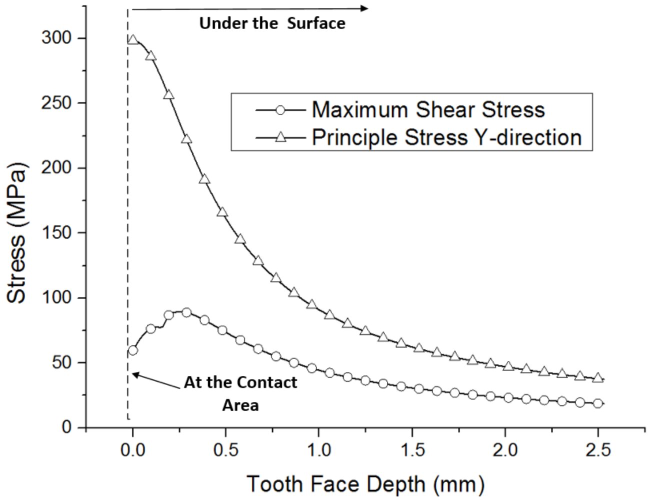

Figure 2. Tooth principle and shear stresses at the contact point and under the surface

The fact that the shear stress was higher under the contact surface explains the formation of micro cracks under the surface due to plastic deformation, and eventually the cracks will develop to full sized macro cracks under cyclic loading, leading to tooth breakage and failure. Under the same loading conditions, the finite element model was simulated using FEA software ANSYS, the input torque applied on the pinion was identical to the numerical calculation and equal to 23000 N.mm.

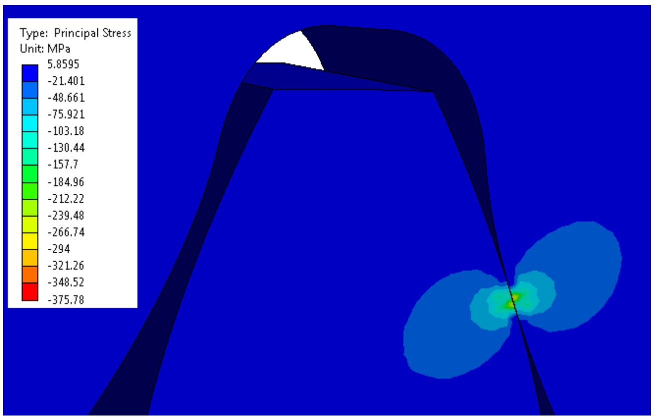

Figure 3. Tooth principle stress at the contact point and under the surface

The results presented are the principal and shearing stresses shown Fig.3 and Fig.4, then the results are plotted along the numerical calculation, in order to verify the simulation model.

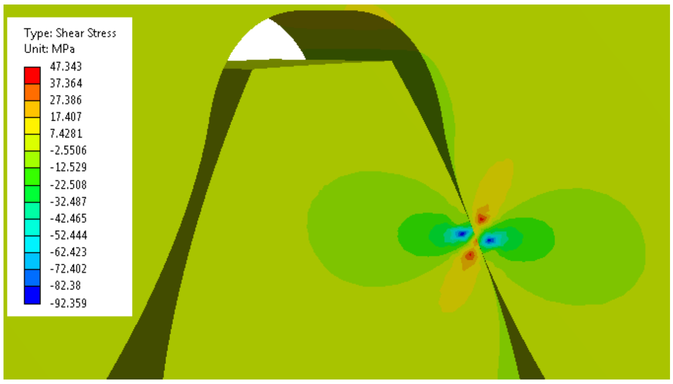

Figure 4. Tooth shear stress at the contact point and under the surface

The data comparison between analytical and simulation results are plotted bellow, the stress variation at the contact point and below the surface followed the same trend for both shear and principles stresses. It can be seen for Fig.5, the shear stress simulation results are slightly different at the contact area from the analytical calculation, where the maximum shear stress was equal to 60 MPa from simulation, and 42 MPa from analytical calculation, similarly for the maximum shear stress under the surface was different and equal to 85 MPa for simulation and 74 MPa for analytical calculation. However both results showed that the maximum shear stress occurred below the contact area, then the values converged identically. The difference was probably due to the accuracy of the model, as the analytical calculation step was reduced to match the simulation model step, and the element size is reduced to 0.5 mm, however for smaller element, as shown in Fig.2, MATLAB analytical calculation for shearing stress showed more accuracy when the element size was smaller. As for principle stresses initially there was a little difference between simulation and analytical models, as the contact principle stress was equal to 350 MPa for simulation and 300 MPa for analytical calculation, then the results showed more coherence as the stress faded under the contacting surface.

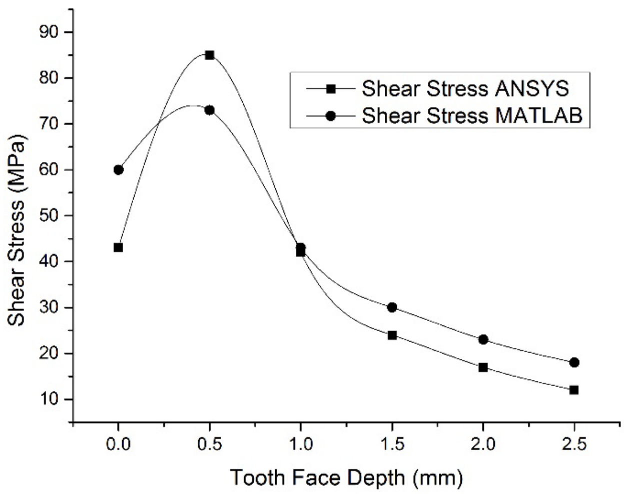

Figure 5. Shear stress at the contact area and below the Surface

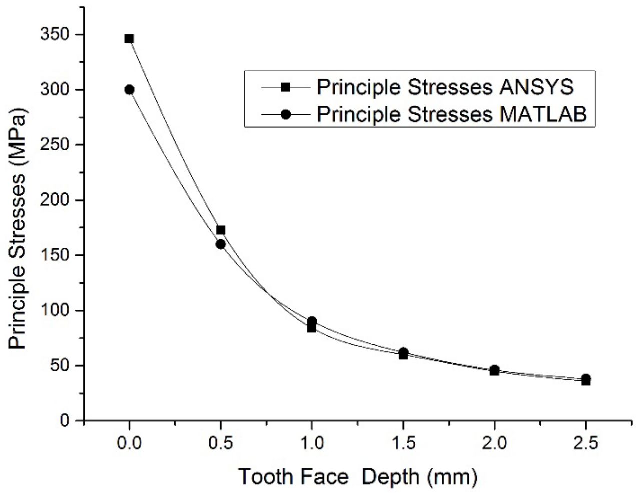

Figure 6. Principle stress at the contact area and below the surface

As for dynamic analysis, the input torque is replaced by an input rotational speed and the gears couple were free to rotate, which allowed to calculate the contact stress along the line of action, the rotational input speed was equal to 2rd/s (radians per second), and the contact surface was set to be frictionless, the set of results are shown below and it presented the value of contact stress between the tooth engagement and disengagement points.

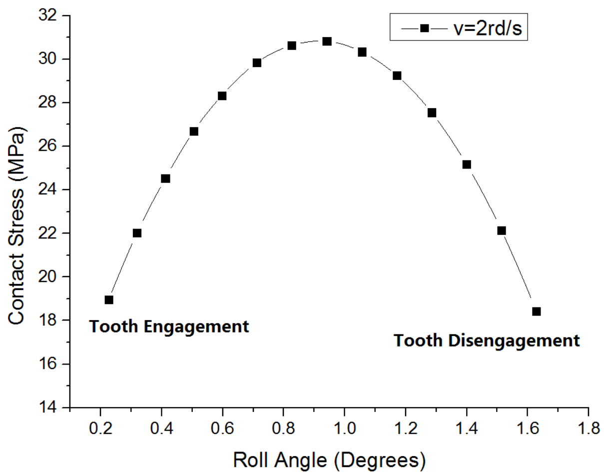

Figure.7. Contact stress along the line of action

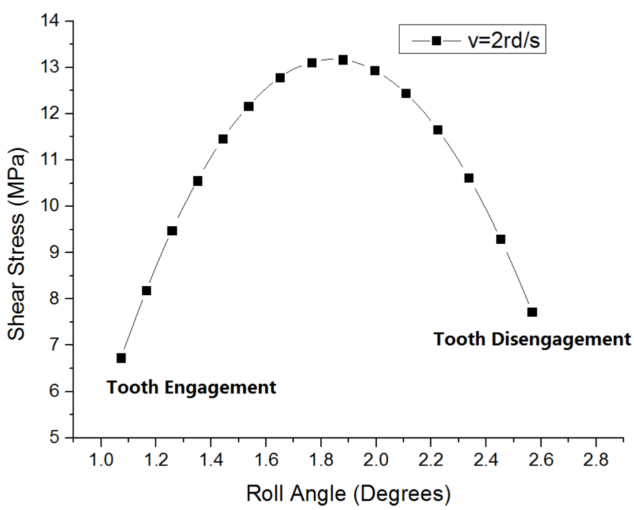

Figure 8. Shear along the line of action

Fig.7 and Fig.8 show the variation of contact and shear stresses along the line of action, both results showed that initially, and at the engagement point the stress was equal to the half of the maximum stress, which take place at the midway of the line of action, this is due to the fact that initially two teeth pairs shared the input load, and midway through the line of action, one pair of teeth carried the full load, then the load is shared again by two teeth pairs before the disengagement point. The results calculated through dynamic model were continuous and smooth, and showed that using current configuration will provide stresses results at different gears meshing points along the line of action, and this model can serve to include more factors, into gears simulations as friction and speed effect on gear tooth stresses.

4 Conclusion

In the following paper, and analytical and FEA model were set, to calculate gear tooth stresses, the finite element model included static and dynamic boundary conditions. Principal and shear stresses at the tooth contact area and under the surface were calculated. And both analytical and simulation results were compared and in general showed good agreement. Further the dynamic model allowed to calculate stresses results along the line of action. The shear stress results showed that the maximum stress value was located under the contact surface and not at the contact point, which indicates why micro cracks appear initially under the surface and propagate to the surface under cyclic loading. The difference between static simulation result and analytical calculation was probably due to the accuracy of the chosen element, and finer elements would have allowed to get more accurate result, however simulation time would have been extended. At the end the dynamic model allowed to calculate continuous stress results, which allowed to interpret stress at every point of teeth engagement, the following model can be used as a base to include more factors into the initial model as friction, in order to interpret its effect on shearing stress. Which can help to redesign gear tooth, while reducing shearing stresses, in order to avoid crack formation and other forms of fatigue as pitting and spalling.

Acknowledgements

This research is partially sponsored by National Natural Science Foundation of China (Grant 51205221).

References

1. Wang J., Numerical and experimental analysis of spur gears in mesh, Curtin University, Perth. 2003.
2. A. F. Rincon, F. Viadero, M. Iglesias, et al., A model for the study of meshing stiffness in spur gear transmissions, Mechanism and Machine Theory, 2003, 62, 50-58.

3. Wang J., I. Howard, The torsional stiffness of involute spur gears, Journal of Mechanical Engineering Science, 2004, 218, 131-142.
4. V. K. Ambarisha, R. G. Parker, Nonlinear dynamics of planetary gears using analytical and finite element models, Journal of Sound and Vibration, 2007, 302, 577-595.
5. M. Topakci, H. K. Celik, D. Yilmaz, et al., Stress analysis on transmission gears of rotary tiller using finite element method, Akdeniz universitesi Ziraat Fakultesi Dergisi, 2008, 21(2), 155-160
6. R.G. Budynas, J.K. Nisbett, Shigley's Mechanical Engineering Design, eighth ed. McGraw-Hill, New York, 2006.