Effect of the Number of Teeth on the Structural Integrity of Gear Shaft base on Finite Element Method

Zhiyuan Li

Department of Mechanical Engineering, University College London, London, United Kingdom

Abstract. Gear shaft is an indispensable component in many dynamic mechanical systems including automotive drive systems, turbine systems, and industrial manufacturing machines. Precise control of the structural integrity of gear shaft with different number of teeth is crucial to the safety and reliability of its application in mechanical systems. In this paper, finite element method (FEM) is adopted to instigative the effect of changing the number of teeth on the strain and deformation of a series of spur gear shafts. Firstly, a mesh type dependence simulation is conducted to determine the appropriate mesh type for finite element analysis (FEA) of gear shaft. Secondly, a mesh size sensitivity test is performed, which determines the suitable mesh size. After the precise FEM is determined, strain and deformation simulations are carried out based on a gear shaft series with gear module 1.5 and number of teeth ranging from 10 to 13. The simulation results demonstrated that increasing the teeth number increases the maximum strain of the gear shafts while decreases the maximum deformation. The results are coherent with the change in gear shaft geometry with the change in teeth number.

Keywords: Gear shaft; Finite Element Method; Simulation; Failure.

1. Introduction

Research on mechanical component's failure has a long tradition. Failure analysis allows prediction of disastrous events due to the mechanical failure of a mechanical component and shed light on the design modification required to avoid such failure. In gear shaft transmission systems, there exists many causes for gear failure, such as improper design, poor geometry, and unsuited material [1]. While statistics have shown that gear failure accounts for approximately 80% of transmission machinery and 10% of rotary machinery failure, stressing the importance of the reliability of gear in its various applied systems [2]. When manufactured, gear shafts are designed to withstand a certain applied load limit. Yet a sudden change of operating condition such as assembling error, overload, or material defects can easily exceed the designed operating limits. It has been reported that gear teeth may be subject to failure from tip yielding or tooth fillet crack due to stress concentration [3]. After the yield strength of the material has been exceeded, the plastic deformation of the tooth may lead to distortion in the gear pitch and tooth thickness which compromises performance of systems. In the worst-case scenario, large deformation can cause gear misalignments and slippage, implying complete failure of the dynamic system. In this paper, failure due to microscopic manufacturing crack is not accounted as the FEM assumes an ideally made gear shaft; rather a careful investigation on the deformation behavior in a gear shaft series with different number of teeth is conducted.

For decades, one of the most popular ideas in such failure analysis in both scientific literature and industrial product development is FEM. This approach is influential in both fields as it offers a computational solution to solid mechanics problems without the need of actual product prototyping and testing [4]. Furthermore, when precise model is created, FEM can provide reliable results. Göksenli and Eryüred employed FEM to investigate the failure of an elevator shaft due to fatigue crack [5]. The FEA results showed that the stress concentration at a notch in the shaft is inversely proportional to the radius of curvature of the fillet in the notch, which corresponds to empirical stress concentrator theory. This allows the conclusion that the failure is due to a faulty design. Moreover, FEA is adopted by Jing and Meng to evaluate the stress concentration on a failed gear shaft connected to an extruder [6], where the region of highest equivalent stress in the FEA model correctly coincides with the point of crack initiation in the actual shaft. Vukelik et al. investigated the crack propagation

due to a crane gear misalignment and found high correspondence between maximum equivalent stress and pressure obtained from experiment and numerical FE model [7].

Therefore, this paper analysed the loading of a spur gear shaft and investigated whether the number of teeth affects the structural performance. The overall goal of this work was to establish a suitable meshing method for FEA of gear shafts and shed light on gear shaft design faults.

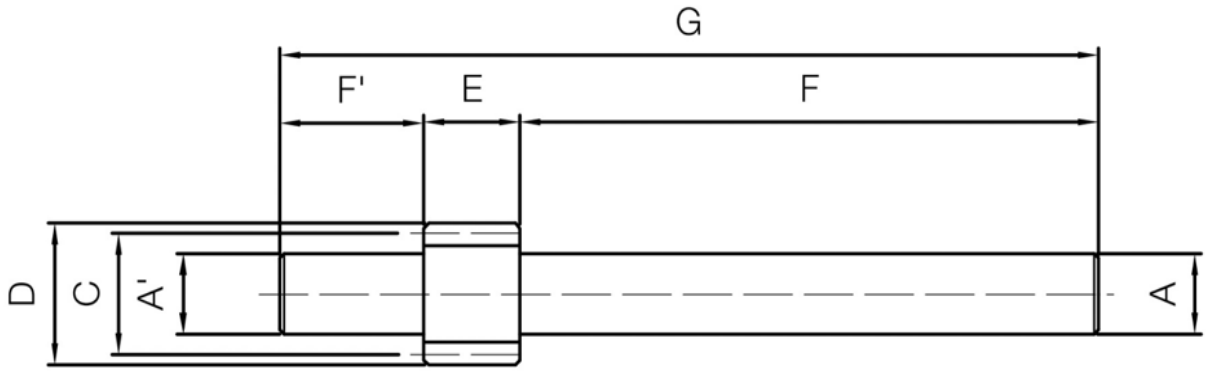
The rest of this paper is organized as follows. Section 2 describes the FEA model used to simulate the deformation of a gear shaft. Section 3 presents the simulated strain and deformation results. Finally, conclusions are drawn in Section 4.

2. Method

The gear shafts investigated are provided by KHK® with product ID SSS1.5-10, SSS1.5-11, SSS1.5-12, and SSS1.5-13. The first and second number in the product ID indicates the gear module and number of teeth. Shaft SSS1.5-10 is shown in figure 1 below for illustration. The gear shafts are made from JIS S45C low carbon steel (0.45% carbon content). All gear shafts have a fixed total length of 140mm and gear face width of 15mm. The pitch diameter, outer gear diameter, and shaft diameter varies depending on the number of teeth. The dimensions for the module 1.5 series gear shafts are specified in figure 2.



Figure 1. SSS1.5-10 (module 1.5, 10 teeth) spur gear shaft (top) and a stem pinion shaft (bottom).



Catalog Number	Module	No. of teeth	Profile shift coefficient	Shape	Shaft diameter (L) A'	Shaft length (L) F'	Pitch dia. C	Outside dia. D	Face width E	Shaft diameter (R) A	Shaft length (R) F	Total Length G
SSS1.5-10	***m*1.5**	10	+0.5	SB	12.2	25	15	19.35	15	12.2	100	140
SSS1.5-11		11	+0.5		13.7		16.5	20.85		13.7		
SSS1.5-12		12	0		13.7		18	21		13.7		
SSS1.5-13		13	0		15.2		19.5	22.5		15.2		

Figure 2. Technical drawing and table displaying the gear shaft dimensions.

2.1. Theory of Finite Element Method

Developed in the 1950s as a numerical computation method, FEM became widely used in engineering modelling. In solid mechanics applications, FEM discretizes a continuous object into a finite set of elements. The displacement and loading condition of each element can then be calculated to define the condition of the continuous object.

The main computation steps of FEM are as followed:

2.1.1 Discretization of Continuous Object

The continuous object can be discretized into 3D elements with either tetrahedron, hexagonal, prism, or other shapes. The elements are interconnected by finites number of nodes at element boundary which contains the local structural condition; the entire domain containing the elements and the nodes is known as the mesh [8].

2.1.2 Elemental Analysis

Analysis of meshed elements is based on nodal displacement and nodal force equilibrium equations, which can be written in matrix form as:

$$\{F^e\} = [K^e]\{u^e\} \quad (1)$$

Where $\{F^e\}$ is the element nodal load vector; $[K^e]$ is the element stiffness matrix; $\{u^e\}$ is the element nodal displacement vector (unknow).

2.1.3 Global Analysis

Global analysis of the continuous object is based on the collective elemental analysis. At region constraint by the applied boundary conditions, the nodal displacement is computed based on its linear relationship with nodal loading. Then, using elemental force equilibrium and the compatibility of strain, the surrounding nodal displacement can be found, leading to the displacement of the entire object. The global force equilibrium condition can be written in matrix form as:

$$\{F\} = [K]\{U\} \quad (2)$$

Where $\{F\}$ is the global nodal load matrix; $[K]$ is the global stiffness matrix; $\{U\}$ is the global nodal displacement matrix (unknown).

To find the unknown$\{U\}$, the stiffness matrix inverse $[K]^{-1}$ is computed such that:

$$\{U\} = [K]^{-1}\{F\} \quad (3)$$

The 3D models of the gear shafts are downloaded from the KHK® website and imported into ANSYS workbench. From online material property table, S45C steel has density 7850kg/m^3, young's modulus 2.05×10^{11}N/m^2, and Poisson ratio 0.29.

The model uses a constant static load of 100N applied at the tip of one tooth tangential to the gear, for all four gears. The gear shaft is fixed by applying a fix support on both end faces of the shaft as shown in figure 3.

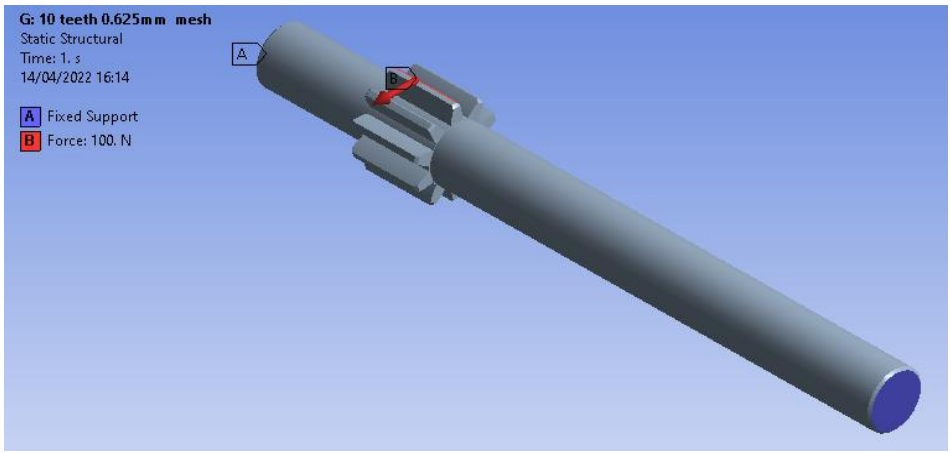
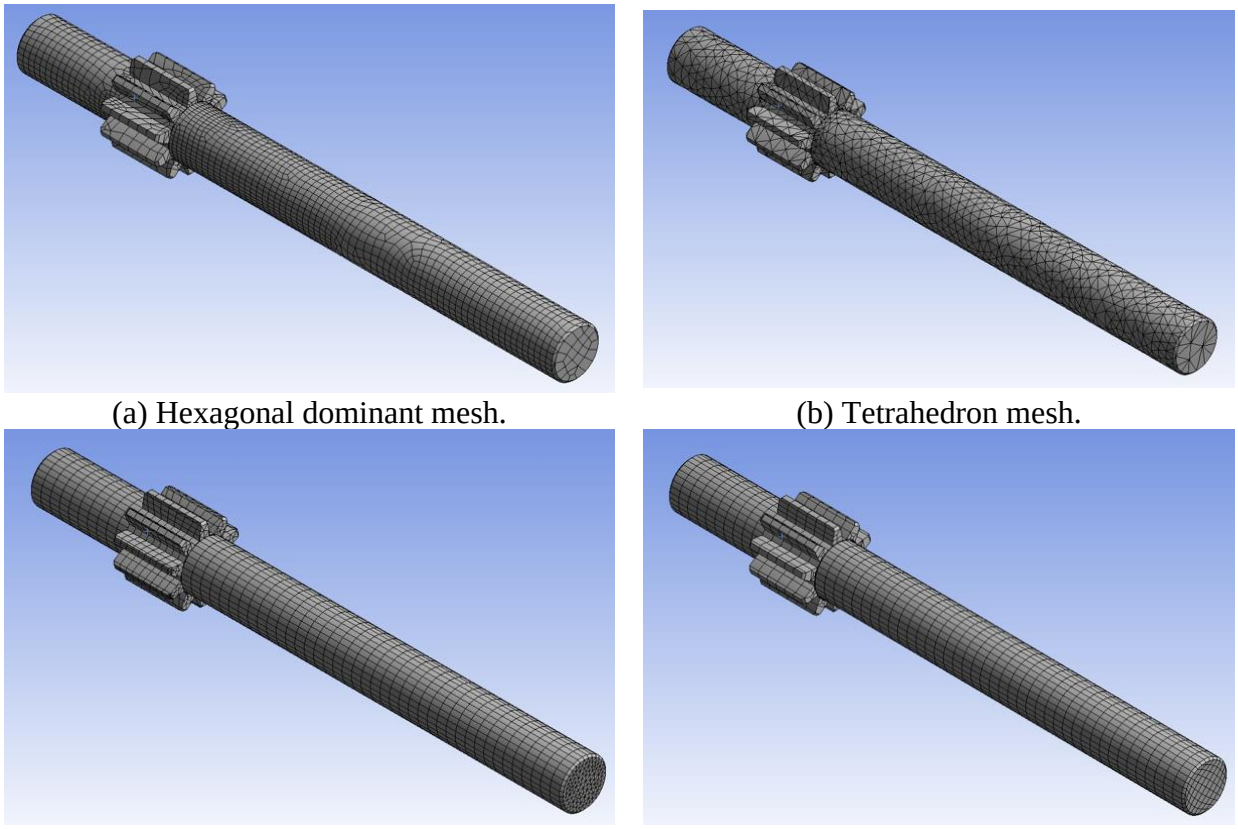


Figure 3. SSS1.5-10 gear shaft FEA boundary condition.

3. Results and Discussions

3.1. Mesh Sensitivity Analysis

A mesh type test using mesh size of 2.5mm is first conducted on the SSS1.5-10 gear shaft to determine the most accurate mesh type out of the four appropriate mesh types available on ANSYS: multizone prism, multizone hexagonal, hexagonal dominant, and tetrahedrons. Figure 4 shows the meshed gear shaft using the four mesh types.



(a) Hexagonal dominant mesh. (b) Tetrahedron mesh.

(c) Multizone prism mesh. (d) Multizone hexagonal mesh.

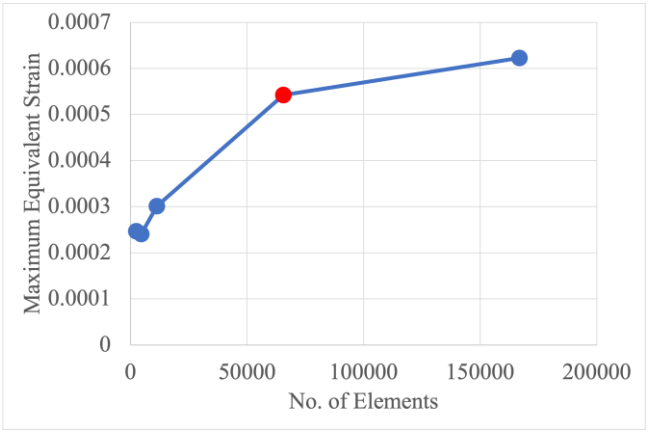
Figure 4. Different mesh types of SSS1.5-10 gear shaft based on mesh size of 2.5mm.

The maximum strain and deformation of the gear shaft based on the four mesh types are averaged and displayed in figure 1. The multizone hexagonal is the only mesh type that has higher than average maximum strain and deformation, indicating high sensitivity to boundary condition input. Hence, it is selected as the most accurate mesh type.

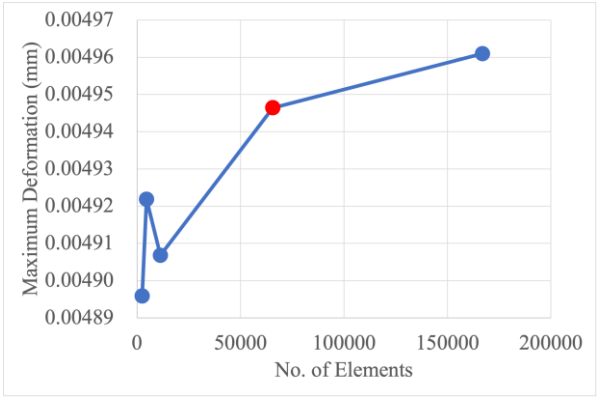
Table 1. Table showing the maximum strain, deformation, and the average values of both.

Mesh Size 2.5mm	Multizone Prism	Multizone Hexagonal	Hexagonal Dominant	Tetrahedron	Average
Max. Strain	0.0002599	0.0002409	0.00020341	0.00013182	0.00020902
Max. Deformation (mm)	0.004839	0.004922	0.004904	0.004779	0.004861

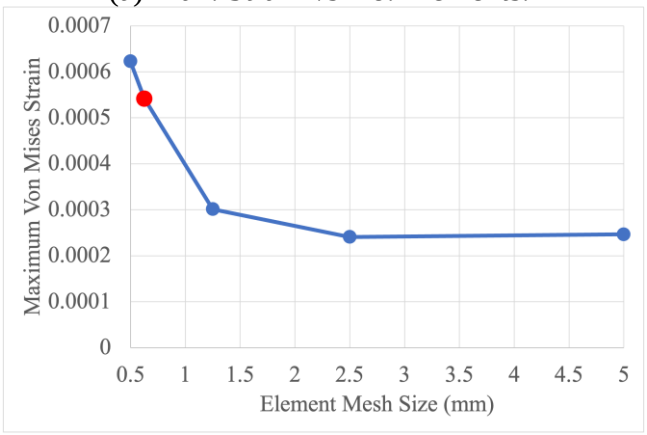
A mesh size convergence test is then performed based on the multizone hexagonal mesh using mesh sizes of 0.5mm, 0.625mm, 1.25mm, 2.5mm, and 5mm. The results are displayed in figure 5 below.



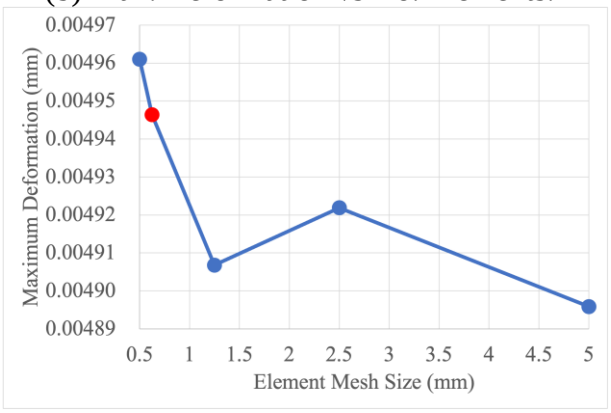
(a) Max. Strain vs No. Elements.



(b) Max. Deformation vs No. Elements.



(c) Max. Strain vs Mesh Size.



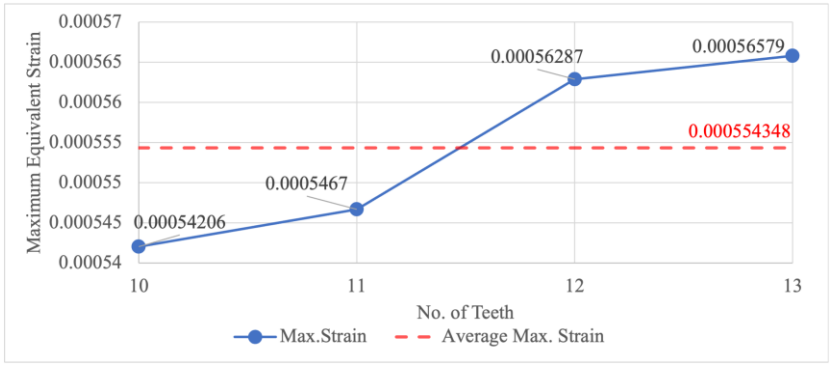
(d) Max. Deformation vs Mesh Size.

Figure 5. Mesh convergence test based on maximum deformation and equivalent strain plotted against mesh size and the number of elements. The markers shaded in red indicate the mesh size of 0.625mm, which is chosen for further simulations.

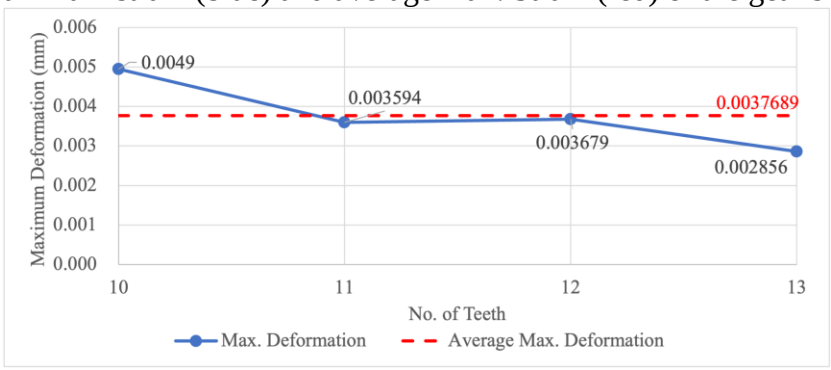
Mesh convergence study shows that both maximum equivalent strain and maximum deformation converges as mesh size reduces. Mesh size of 0.625mm corresponding to 65592 mesh elements are chosen for further simulations due to its accuracy relative to the 0.5mm mesh yet significantly lower element number and computational time.

3.2. Strain and deformation analysis

Based on multizone hexagonal mesh of size 0.625mm, strain and deformation for all gear shafts are simulated. Figure 6 presents the maximum strain and deformation for all four gear shafts plotted against the number of teeth. It can be noted that under a constant applied load, the increase of gear teeth number increases the maximum equivalent strain while decreases the maximum deformation. However, the overall variation in strain and deformation is small. The total strain and deformation result for SSS1.5-10 is illustrated in figure 7 as an example. The maximum strain and deformation both occur at the edge of the gear teeth where load is applied.

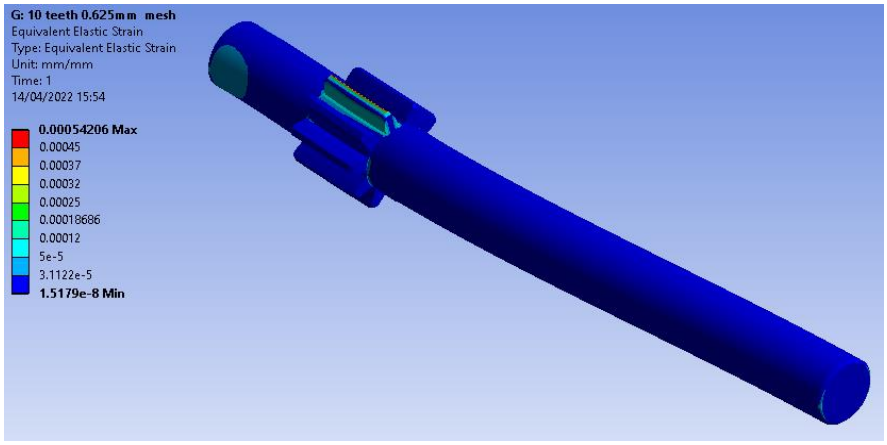


(a)Maximum strain (blue) and average max. Strain (red) of the gear shafts.

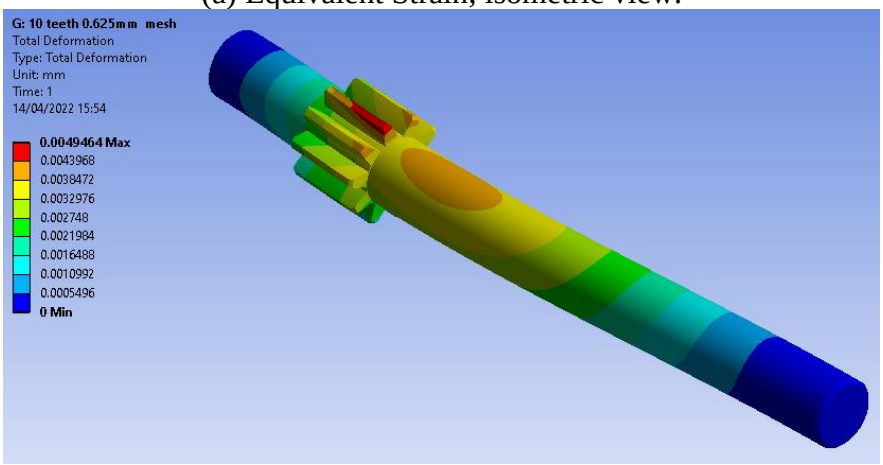


(b) Maximum deformation (blue) and average max. deformation (red) of the gear shafts.

Figure 6. Maximum strain and deformation against the number of teeth of the gear shaft.



(a) Equivalent Strain; isometric view.

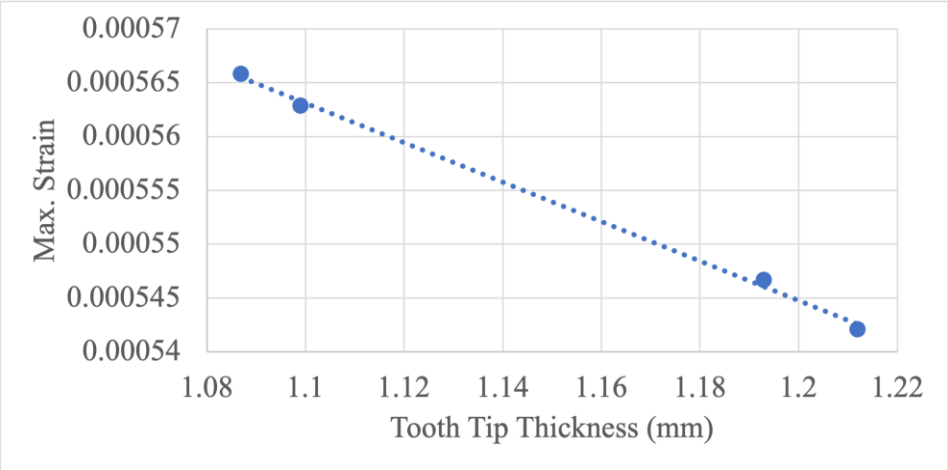


(b) Deformation; isometric view.

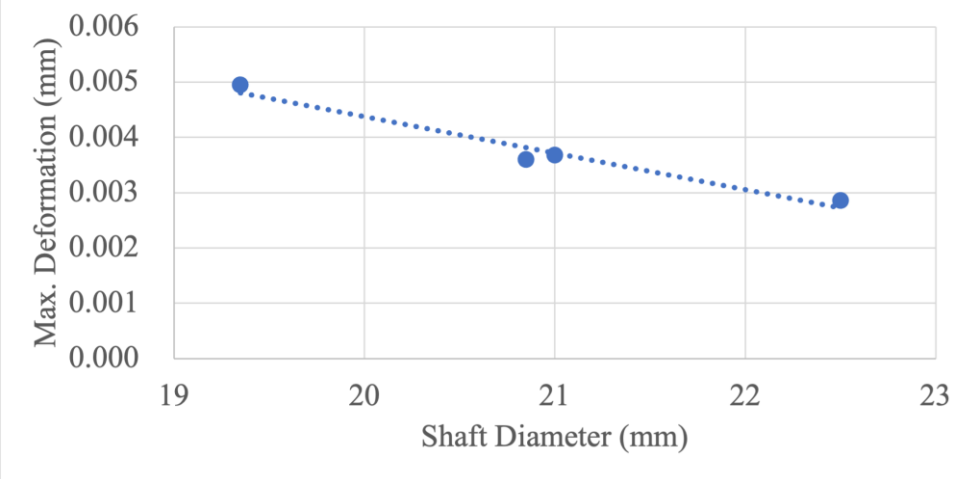
Figure 7. Total deformation and strain simulation results of SSS1.5-10.
Gear shaft deformation is shown in 1000x magnification relative to real scale.

The opposite trend of maximum strain and deformation variation with teeth number can be explained via the change in geometry of the gear shaft as teeth number increases: the shaft diameter increases while the tooth thickness at the tip where the load is applied decreases. Increase in shaft diameter will increase the second moment of area of the shaft, hence enhance its resistance to the transverse bending load that causes deformation. Although the overall deformation decreases with increasing shaft diameter, local deformation at the tip become larger due to thinner tooth thickness. As a result, maximum strain is greater for gear shaft with greater teeth number.

The maximum strain is plotted against tooth thickness in figure 8(a) and the maximum deformation is plotted against shaft diameter in figure 8(b). A line of best fit is shown for both plots. Strong linear correlation confirms previous explanation on strain and deformation trends.



(a) Maximum strain vs. tooth thickness of gear shafts.



(b)Maximum deformation vs. shaft diameter.

Figure 8. Plots showing the relationships between maximum strain and tooth tip thickness and between maximum deformation and shaft diameter.

4. Conclusions

This paper developed a static structural simulation model based on finite element method. The structural integrity of SSS1.5 series gear shafts is investigated. The conclusions can be summarized as follows:

(1) Multizone hexagonal mesh with element size of 0.625mm is the most suitable meshing model for this gear shaft series based on its sensitivity of strain and deformation simulations.

(2) Increasing the number of gear teeth shows a positive correlation with the maximum strain and a negative correlation with the maximum deformation, both of which are due to geometrical changes following increase of teeth number.

Based on the current work, future research should be devoted to investigate the effect of dynamic loading response of the gear shafts to fully characterize the effect of teeth number on structural integrity.

Reference

[1] W. Feng, Z. Feng, and L. Mao, "Failure analysis of a secondary driving helical gear in transmission of electric vehicle," Engineering Failure Analysis, vol. 117, p. 104934, Nov. 2020, doi: 10.1016/j.engfailanal.2020.104934.

[2] R. Ma, Y. Chen, and Q. Cao, "Research on dynamics and fault mechanism of spur gear pair with spalling defect," Journal of Sound and Vibration, vol. 331, no. 9, pp. 2097–2109, Apr. 2012, doi: 10.1016/j.jsv.2011.12.010.

[3] W. Yu, C. K. Mechefske, and M. Timusk, "Effects of tooth plastic inclination deformation due to spatial cracks on the dynamic features of a gear system," Nonlinear Dynamics, vol. 87, no. 4, pp. 2643–2659, Nov. 2016, doi: 10.1007/s11071-016-3218-y.

[4] Z. Bi, "Chapter 8 - Applications—Solid Mechanics Problems," ScienceDirect, Jan. 01, 2018. https://www.sciencedirect.com/science/article/pii/B978012809952000008X

[5] A. Göksenli and I. B. Eryürek, "Failure analysis of an elevator drive shaft," Engineering Failure Analysis, vol. 16, no. 4, pp. 1011–1019, Jun. 2009, doi: 10.1016/j.engfailanal.2008.05.014.

[6] J. JianPing and M. Guang, "Investigation on the failure of the gear shaft connected to extruder," Engineering Failure Analysis, vol. 15, no. 4, pp. 420–429, Jun. 2008, doi: 10.1016/j.engfailanal.2007.01.010.

[7] G. Vukelic, D. Pastorcic, G. Vizentin, and Z. Bozic, "Failure Investigation of a Crane Gear Damage," Engineering Failure Analysis, p. 104613, May 2020, doi: 10.1016/j.engfailanal.2020.104613.

[8] A. E. Tekkaya and C. Soyarslan, "Finite Element Method," CIRP Encyclopedia of Production Engineering, pp. 508–514, 2014, doi: 10.1007/978-3-642-20617-7_16699.