University of Dundee

Design and finite element analysis of a fatigue life prediction for safe and economical machine shaft

Afolabi, Samuel O.; Oladapo, Bankole I.; Ijagbemi, Christianah O.; Adeoye, Adeyinka O.M.; Kayode, Joseph F.

Original Article

Design and finite element analysis of a fatigue life prediction for safe and economical machine shaft

Samuel O. Afolabi[a], Bankole I. Oladapo[a,*], Christianah O. Ijagbemi[b], Adeyinka O.M. Adeoye[a], Joseph F. Kayode[a]

[a] Department of Mechanical and Mechatronics, Engineering, Afe Babalola University, Ado-Ekiti, Nigeria
[b] Department of Mechanical Engineering, Federal University of Technology, AKure, Nigeria

ARTICLE INFO

ABSTRACT

In materials engineers, it is important to determine the cause of failure of a machine component, to prevent prospect occurrences and increase the performance of the component structure. In this study, the parameters of the fatigue life of machine shafts are investigated. An analysis of the nut cracking machine shaft was conceded for plastic deformations. The optimum safe and economical design of a machine shaft was proposed. The 3D model of a shaft was produced with Inventor® using absolute coordinate. The results of the commercial finite element analysis (FEA) and calculations are compared with results obtained earlier by other methods. The analysis of 30 mm shaft diameters under the maximum torque of 72.0 Nm shows a factor of safety of 10, while the 20 mm shaft diameter under the same torque gives a factor of safety of 2. This will provide designers guidelines to forecast the design on fatigue strength of a machine shaft.

1. Introduction

The durability of a machine structure can be defined as the skill of the structure in order to keep up its mechanical performance through its service life. Therefore, there are a close relationship between durability and safety. Structural failure is primarily due to static and fatigue lots. Hence, the machine can be analyzed according to uncommon types of loading, which is static and fatigue, in order to design safe and dependable structures. Many literatures, research on experimental or numerical studies related to machine based structures [1–3]. Element treatment is not limited to the engineering field, but furthermore, extends to health check and geospatial an application which is defined as a method of applying statistical analysis to data which has a geographical feature. Rapid advancement in a finite element is due to powerful computer processors and unremitting software development. In contemporary years the aid of finite element in engineering was enormously increased [3,4]. Key factors in finite element analysis (FEA) are numerical computations with the intention of estimate all parameters and boundaries agreed. A good effort

* Corresponding author.
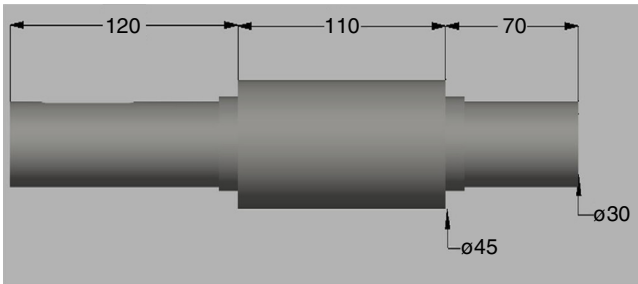E-mail: bankolyable01@gmail.com (B.I. Oladapo).

Fig. 1 – Design of shaft in 2D with the dimension in millimeter.

Table 1 – Physical and mechanical properties of shaft material.

Material	Steel
Mass density	7.85 g/cm^3
Mass	2.44247 kg
Area	37,098.1 mm^2
Volume	311,143 mm^3
Yield strength	207 MPa
Ultimate tensile strength	345 MPa
Young's modulus	210 GPa
Poisson's ratio	0.3 μl
Shear modulus	80.7692 GPa
Center of gravity	$x = 158.425$ mm
	$y = -0.0472625$ mm
	$z = 0$ mm

has been made to improve the productivity of kernel nut over the years. Engineers in their effort designed various machines to enhance the production. It is a growing field that keeps evolving to meet the pace at which science is growing. Engineers keep improving on current designs to bring about good productivity with methods of automating the verification of an acceptable freeform surface, using coordinate measuring machine (CMM). Computer-aided geometric design (CAGD) is used to analyze the surface for optimum continuity and assess the CMM data accuracy [5–7]. A broad understanding of many uncommon failure modes with the intention of existing in support of one logic agreement will be the answer for analyzing one failure of mechanical components. Spanning a large range of applications, such as a steam turbine engine, chatter turbine engines, plastic structures, apparatus, tools and furniture, compressors, personnel, equipment, pumps, turbine blades, rotating shafts heating tank, exhaust hoods rotors, turbine impellers, and support structures would solve the failure analysis for manufacturing equipment components. Fatigue is the gradual wear of a material as subjected to continual lots, normally separated into three stages: crack admittance, crack procreation (power law growth) and unstable, a rapid growth [2,5–7]. The experiences encountered during the development, modifications, and refinement of a finite element model of different machine component in automobile, aircraft and marine industries are vital [8,9]. This model was successfully used in the simulations of the full frontal, offset frontal, side, and oblique car-to-car impacts. The simulation results were validated with test data of actual vehicles. The validation indicates that the model is suitable for use as a crash partner for other vehicles. Computational tests of the model show that the model was computationally stable, reliable, and repeatable [4–6]. In 2001 research has presented a multi-probe measuring system integrated with a CMM, a structured light sensor, a trigger probe and a rotary table. Two types of scanning modes, which are a multi-view scanning mode and rotating scanning mode, have been used [7,10]. A simulation package was used [11,12]. In this research, the methodology of simulation and design has been employed to the shaft of a palm kernel cracking machine. This paper explains the generation of a 3D model of the shaft using an absolute coordinate technique. Stresses are analyzed at different loads and torque. Finally, the results of Autodesk Inventor for 30 mm diameter and 20 mm diameter were compared. Fig. 1 shows the geometric model which is generated by the AutoCAD software manually and then imported to Inventor software. The dynamic simulation of the system was loaded. Meshing and loading of property on software operation manually. Most finite element modeling is interaction with the user. The design result effectiveness depends on the experience of the designer in the design process [13–15]. The first prerequisite for an accurate and a precise design and model of a machine component is the knowledge of 3-D on stress-strain concentration factor. There is little literature on non-trivial geometries for boundary condition which explained and analyzed 3-D stress and strain field curved boundary condition and they are complex [16–18]. To our understanding, this is the first research on the effect of the working force on the shaft of a nut cracking machine of stress and strain factor to determine the factor of safety and the displacement under different loading. The different method done by many researchers [11–15], of finite element analysis was adopted for this same analysis. The main aim is to study the effect of different loading to the maximum load on the shaft of the machine and research on the bending moment and shear force of the shaft and know if there is the need to reduce the thickness of the shaft using the 3-D finite element method.

2.   Methodology

The shaft is a principal component part that transmits circular motion from the pulley derived from the motor to the hammer mill of the machine, which may lead to the cracking of the hammer mill and the nut. The shaft diameter of an existing palm kernel cracking machine was measured as 30 mm using an electronic digital caliper. The physical properties and mechanical properties of the shaft material applied area are shown in Table 1. The finite element computation was performed using SolidWorks and Inventor Standard. The finite-element meshes of these models were generated using eight-node-linear brick reduced integration elements (C3D8R). Using the above explanations, the finite element meshes of the ship structure have been created. The total number of elements and nodes are 59,467 and 528,576 respectively in the structural model. Fine meshes have been introduced near to the edges of bulb flats. Table 2 shows loading elements for the different models that were used in the current study after several refined meshes to ensure the conversion of FEA results.

Table 2 – Loads on the shaft.		
Quantities	Unit	Value
Mass	Mass	2.45 kg
Length	L	300.00 mm
Maximal bending stress	σ_B	153.25 MPa
Maximal shear stress	τ_S	29.55 MPa
Maximal tension stress	σ_T	0.00 MPa
Maximal torsional stress	τ	2.70 MPa
Maximal reduced stress	σ_{red}	158.93 MPa
Maximal deflection	f_{max}	178.92 μm
Angle of twist	φ	0.01°

Table 3 – Finite element analysis mesh settings.	
Parameters	Values
Avg. element size (fraction of model diameter)	0.1
Min. element size (fraction of avg. size)	0.2
Grading factor	1.5
Max. turn angle	60°
Create curved mesh elements	Yes

2.1. Finite element mathematical analysis of simulation

According to this research, the factor of the concentration of stress and strain is represented as k_σ and k_ε, respectively, and the mid-level stress and strain concentration factors of the plate are denoted with KRMP and KEMP, respectively, according to:

$$k_\sigma = \frac{\sigma_{NR}}{\sigma_{net}} \tag{1}$$

$$k_{RMP} = k_\sigma \text{ at } z = 0$$

$$k_\sigma = \frac{\varepsilon_{NR}}{\varepsilon_{net}} \tag{2}$$

$$k_{EMP} = k_\sigma \text{ at } z = 0$$

$$\sigma_{net} = \frac{\sigma_{yw}}{W - D} \tag{3}$$

$$\varepsilon_{net} = \frac{\sigma_y W}{E(W - D)} \tag{4}$$

In the Y direction of plate of the notch root, σ_{NR} and σ_{net} represent the longitudinal stress and strain; mean stress and strain are ε_{net} and ε_{net} for the net section respectively. The k is the stress and strain concentration factor.

2.2. Loading condition on the shaft

AutoCAD 3D modeling enables one to create drawings in solid view. Solid, surface and mesh objects were used functionally together, which produced a good 3D model of the shaft. A solid model is a volume that represents a 3D object and has properties such as mass, the center of gravity, and moments of inertia. Table 3 shows the finite element mesh settings for the shaft.
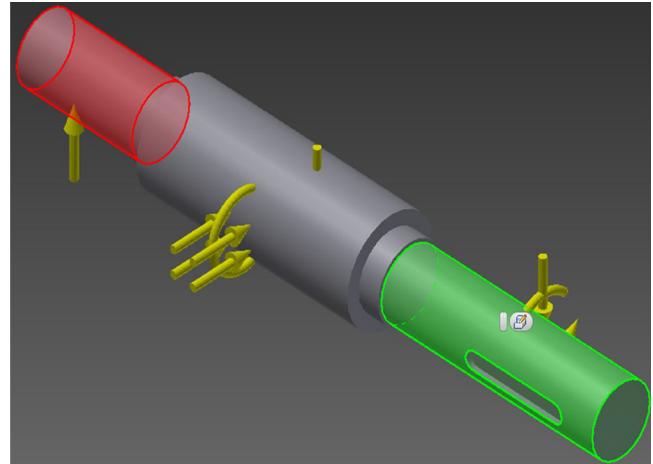

Fig. 2 – Constraints and loading conditions of shaft.

The load that existed on the shaft was studied in detail. The actual load that existed on the shaft of the machine was derived by the software after imputing the revolution and power of the electric motor into the software directly which are 1430 rev/min and 2.235 kW respectively. Fig. 2 shows the constraints and loading conditions of the shaft.

The minimum torque required to turn the shaft from the pulley was 41.0 Nm and the maximum torque applied is 58.3 Nm. The impulse force load of the palm kernel was calculated to be 12000 N/s minimum and 15,000 N/s maximum. This was calculated from the average mass of 25 kg of the palm kernel weight. The load distribution is as shown in Fig. 3, which describes the loading force and distribution of shear force on YZ plane, the bending moment and the deflection on the shaft. Table 4 describes the reaction force and moment on the constraints for the shaft.

In order to understand the various stresses that may occur within the shaft while in operation, stress analysis was carried out with the following assumptions. (1) Linear – stress is proportional to strain, according to Hooke's law. (2) All properties are independent. (3) Homogeneity properties will not change throughout the volume of the part. (4) Isotropic – material properties are identical in all directions. The stress analysis of the shaft under the mobile condition, the following loading condition was considered. When the shaft moves in a clockwise direction by the torque of the motor, the impulse force of the palm kernel is loaded in the opposite direction, the pressure exacted by the cracker is also applied and the force of gravity. The minimum moment 58.3 Nm, acting on the shaft, being derived from the use of the inventor and the maximum moment as 72.0 Nm, then the analysis was carried out at the stepping rate of 33 mm diameter and 30 mm diameter in order to study its behavior and reaction.

3. Results and discussion

Table 3 shows the mesh of the shaft which was generated by Inventor for the purpose of analysis under loading moment ranging from the minimum value to the maximum value.
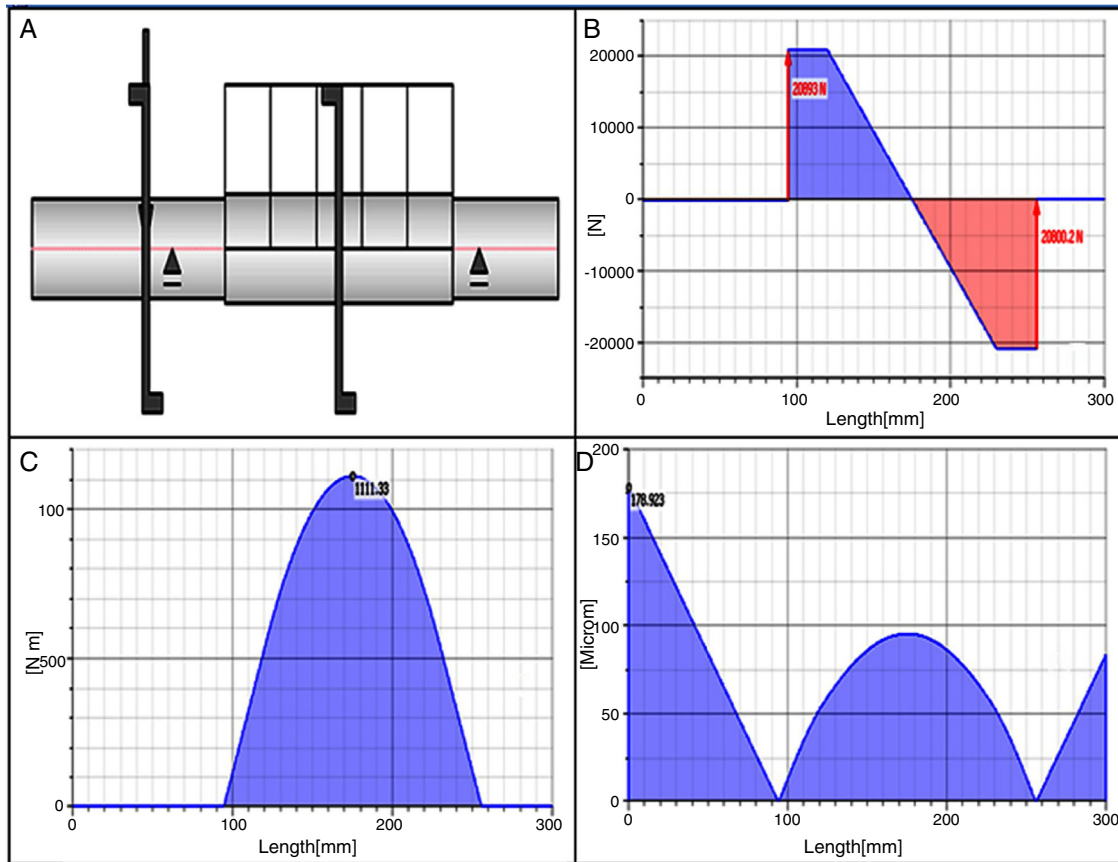
Fig. 3 – (A) Force loading and distribution on shaft. (B) Shear force, YZ plane. (C) Bending moment. (D) Deflection on the shaft.

Table 4 – Reaction force and moment on constraints.

Constraint name	Reaction force		Reaction moment	
	Magnitude (N)	Component (X, Y, Z) (N)	Magnitude (Nm)	Component (X, Y, Z) (Nm)
Pin constraint: 1	3636.67	217.179 0.0 −3630.18	20.0925	19.9366 −2.4983 0.0
Pin constraint: 2	3595.82	−217.27 0.0 −3589.25	36.0985	−36.0967 0.357705 0

After applying all the forces and moments in the evaluation, the maximum Von Mises stress, 1st principal stress, displacement and factor of safety (FOS) at the moment of 58,343.00 mm and 72,000 mm are 0.00000623362 MPa, 0.130798 MPa, 0.00182501 mm and 10.5296 µl respectively. As the minimum factor of safety is greater than 1, the design of the shaft of diameter 30 mm is safe at both maximum and minimum applied torque, while the minimum factor of safety is 10.5296 which mean the shaft can withstand ten times the load and torque applied, which is practically impossible to apply. Therefore, this is over designing and waste of material. Table 5 shows the results obtained and comparison between the two shafts. Fig. 4 shows the results analysis of 30 mm diameter of the shaft of the Von Mises stress at a torque of 72.0 Nm, 1st principal stress at a torque of 72.0 Nm, the displacement at torque at 72.0 Nm and factor of safety at the same torque.

Since the common shaft present in the palm kernel cracking machine of diameter 30.00 mm is too big for the work is performing there is a need for redesigning. This can be done by changing the material or by reducing the size of the shaft to suit the machine. But it is advisable to use the existing material (mild steel) in the market, which is cheap and easy to machine. Therefore, the reasonable method available is to reduce the size of the shaft. Fig. 5 describes results analysis of 30 mm diameter of the shaft of Von Mises stress at a torque of 72.0 Nm, 1st principal stress at a torque of 72.0 Nm, displacement at a torque of 72.0 Nm and the factor of safety at the same torque.

When all the forces and moment have been applied to the system, the maximum Von Mises stress, 1st principal stress, displacement and factor of safety (FOS) at a moment of 58.3 Nm and 72.0 Nm are 0.0000342006 MPa, −0.0913924 MPa, 0.0072593 mm, and 2.02199 µl respectively. As the minimum

Table 5 – Shaft analysis and comparison between diameter 20 mm and 30 mm at maximum torque of 72.0 Nm.

| Name	Diameter 20 mm		Diameter 30 mm	
	Minimum	Maximum	Minimum	Maximum
Volume	112,211 mm^3		311,131 mm^3	
Mass	0.881979 kg		2.44238 kg	
Von Mises stress	3.420×10^5 MPa	102.374 MPa	8.361×10^5 MPa	19.655 MPa
1st principal stress	-0.0914 MPa	65.10 MPa	-0.13214 MPa	11.165 MPa
3rd principal stress	-64.529 MPa	9.84×10^2 MPa	-11.5303 MPa	1.12×10^2 MPa
Displacement	0 mm	7.26×10^3 mm	0 mm	1.83×10^3 mm
Safety factor	2.02199 µl	15 µl	10.5314 µl	15 µl
Stress XX	-5.366 MPa	3.547 MPa	-2.06252 MPa	1.094 MPa
Stress YY	-17.150 MPa	17.434 MPa	-8.28839 MPa	6.590 MPa
Stress ZZ	-17.125 MPa	20.53 MPa	-7.99338 MPa	6.420 MPa
Equivalent strain	1.347×10^{10}	3.956×10^4	3.499×10^{10}	8.112×10^5
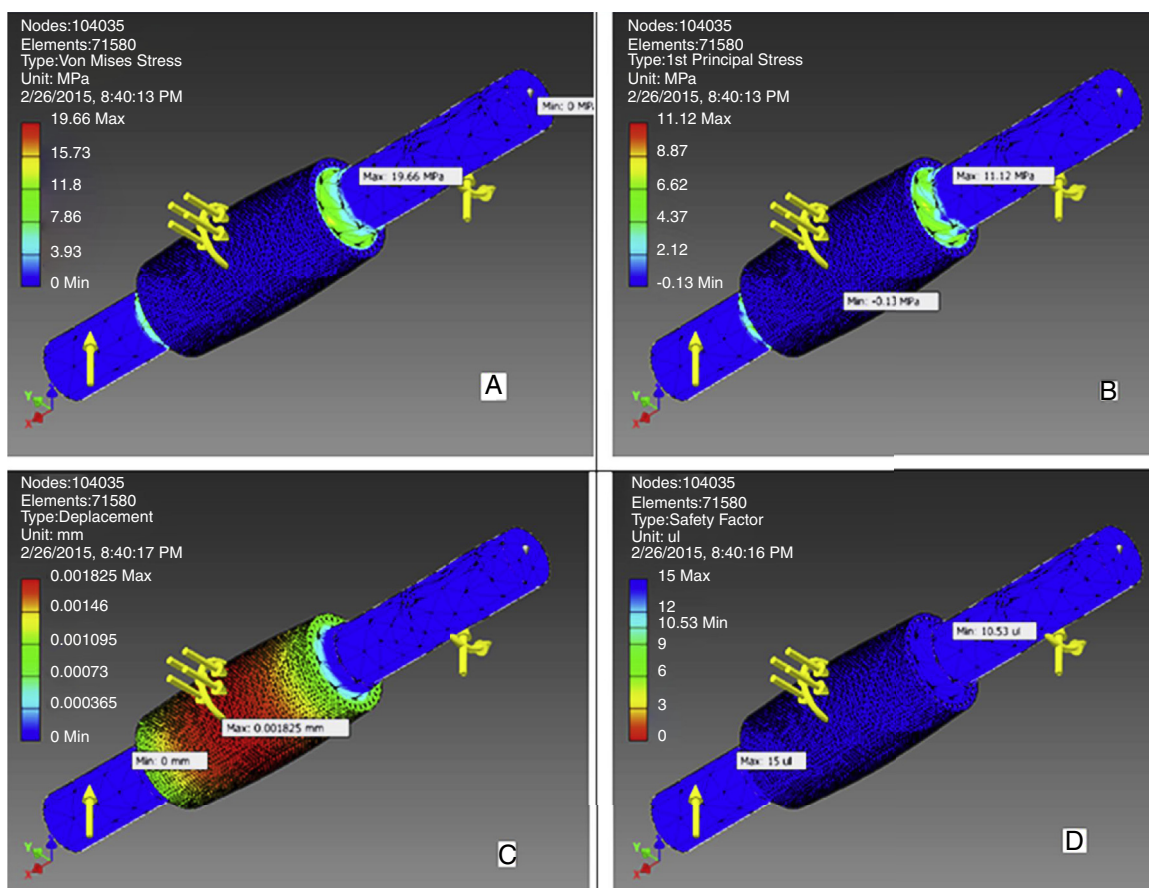
Fig. 4 – Results analysis of 30 mm diameter of shaft: (A) Von Mises stress at torque of 72,000 Nm, (B) 1st principal stress at torque of 72,000 Nm, (C) displacement at torque of 72,000 Nm, (D) factor of safety at torque of 72000 Nm.

factor of safety is greater than 1, the design of the shaft of diameter 30 mm is safe at both maximum and minimum applied torque. But the minimum factor of safety is 2.02199 µl which means the shaft can withstand at least two times of the load and torque apply which is suitable for a design of a palm kernel cracking machine of 2.443 kW. Fig. 6 shows the analysis results at different torque loading from minimum 58.3Nm loading off to the maximum axial loading of 72.0 Nm.

4. Conclusions

The 3D design and analysis of the shaft of 30 mm diameter palm kernel-cracking machine have been analyzed and simulated, the following conclusions are made:

1. From Fig. 6, the analysis shows that at maximum applied torque of 72.0 Nm, the factor of safety for 30 mm diameter
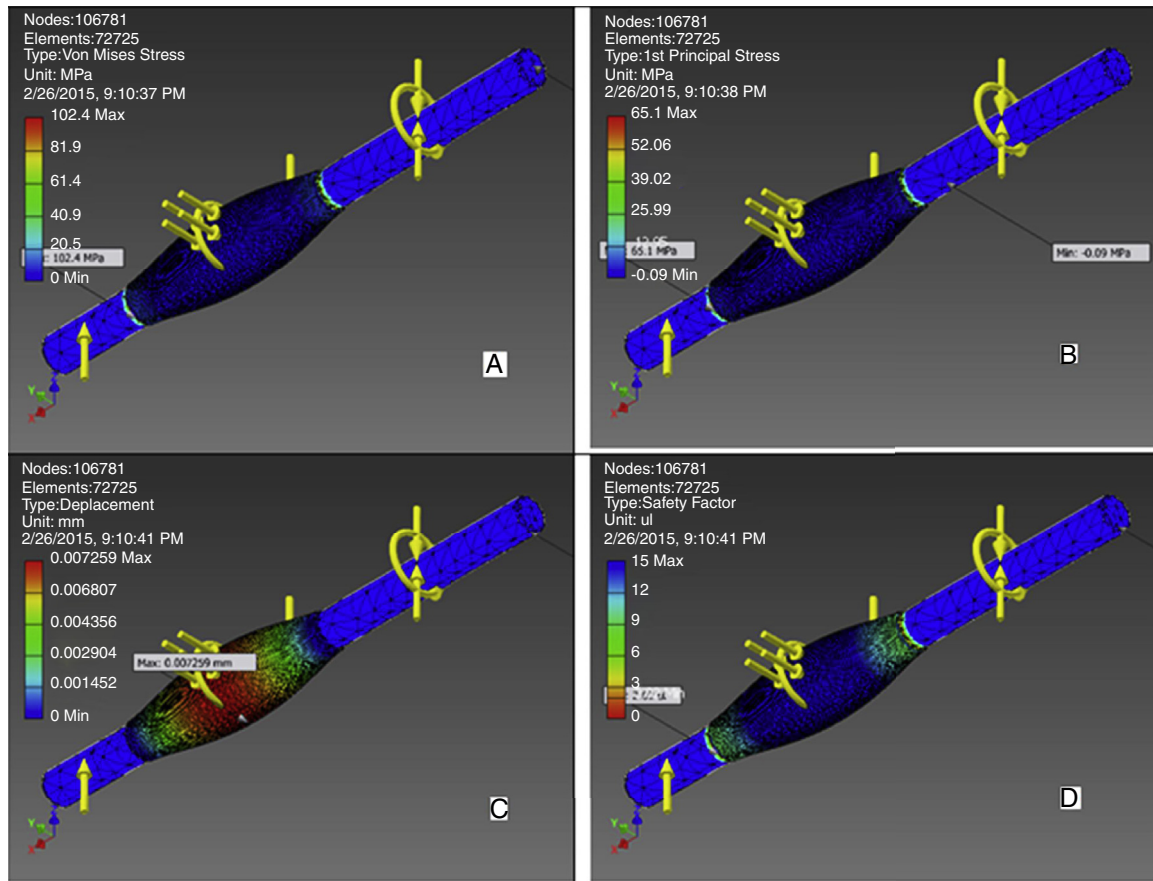
Fig. 5 – Results analysis of 30 mm diameter of shaft: (A) Von Mises stress at torque of 72,000 Nm, (B) 1st principal stress at torque of 72,000 Nm, (C) displacement at torque of 72,000 Nm, (D) factor of safety at torque of 72,000 Nm.
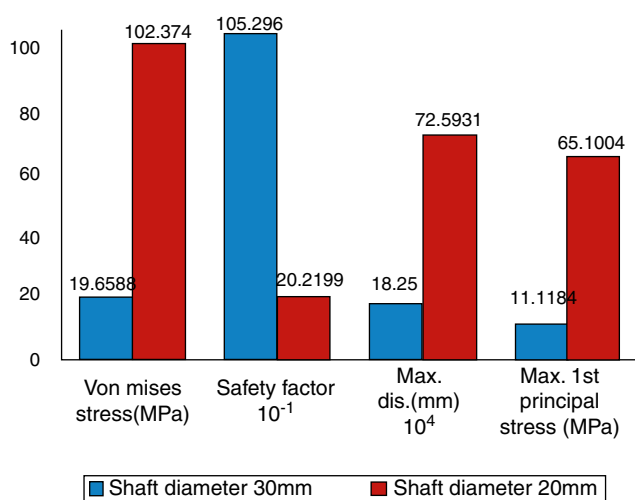

Fig. 6 – Comparison between 30 mm diameter and 20 mm diameter.

2. The result for a factor of safety for 20 mm diameter is 2.02199 µl, which indicates the shaft can carry two times the load and the Von Mises stress 102.4 MPa under the ultimate yield stress of 156 MPa for mild steel material, which complies with VMS theory.
3. The working conditions of a machine shaft structure were investigated. The FEA was carried out taking into account linear constitutive relations of a plastic deformation and residual stresses to determine the stress and strain states, Von Mises stress distribution of the machine shaft structure under static loads.
4. The shaft diameter of 20 mm is appropriate for manufacturing of the machine and also safe and economical.

Conflicts of interest

The authors declare no conflicts of interest.

is 10.5296 µl. This indicates the shaft can carry about ten times the torque and load which is practically impossible showing over design of the component and that of 20 mm diameter is 2.02199 µl, which shows an optimized design.

REFERENCES

[1] Ertas VA, Ahmet FY. Finite element simulation of a mercantile vessel shipboard under working conditions. Procedia Eng 2014;69:1001–7.

[2] Xie Z, Wang J, Zhang Q. Complete 3D measurement in reverse engineering using a multiprobe system. Int J Mach Tools Manuf 2005;45:1474–86.

[3] Swadesh KS, Vinay K, Paresi PR, Gupta AK. Finite element simulation of ironing process under warm conditions. J Mater Res Technol 2014;3(1):71–8.

[4] Al-Bahkali EA, Abbas AT. Failure analysis of vise jaw holders for hacksaw machine. J King Saud Univ Eng Sci 2016, http://dx.doi.org/10.1016/j.jksues.2015.12.007.

[5] Balendran VR, Sivayoganathan K. Accuracy analysis of 3D data collection and free-form modeling methods. J Mater Process Technol 2003;133:26–33.

[6] Syed MH, Swadesh KS, Amit KG. Experimental and numerical investigation of formability for austenitic stainless steel 316 at elevated temperatures. J Mater Res Technol 2014;3(1):17–24.

[7] Ertas AE, Sonmez FO. A parametric study on fatigue strength of spot-weld joints. Fatigue Fract Eng Mater Struct 2008;31:766–76.

[8] Ertas AH, Sonmez FO. Design optimization of spot-welded plates for maximum fatigue life. Finite Elem Anal Des 2011;47:413–23.

[9] Qu NS, Fang XL, Li W, Zeng YB, Zhu D. Wire electrochemical machining with axial electrolyte flushing for titanium alloy. Chin J Aeronaut 2013;26(1):224–9.

[10] Ijagbemi CO, Oladapo BI, Campbell HM, Ijagbemi CO. Design and simulation of fatigue analysis for a vehicle suspension system (VSS) and its effect on global warming. Procedia Eng 2016;159:124–32.

[11] Oladapo BI, Balogun AV, Oke AO, Esoso AA. Design and finite element analysis on car seat height screw adjuster using autodesk inventor. IJSRES 2015;2(8):82–6.

[12] Jaiganesh V, Manivannan S. Numerical analysis and simulation of nylon composite propeller for aircraft. Procedia Eng 2014;97:1079–88.

[13] Xuejin S, Yunfei L, Lei C, Xiaoyang C. Numerical simulation of sliding wear for self-lubricating spherical plain bearings. J Mater Res Technol 2012;1(1):8–12.

[14] Wang D, Zhu Z, Wang H, Zhu D. Convex shaping process simulation during counter rotating electrochemical machining by using the finite element method. Chin J Aeronaut 2015.

[15] Suresh K, Srinivasa PR. Experimental and numerical studies on formability of extra-deep drawing steel in incremental sheet metal forming. J Mater Res Technol 2014;3(2):158–71.

[16] Oladapo BI, Aban S, Azeez MT, Afolabi SO. Computer aided drafting and construction of standard drafting table for college of engineering studio in Afe Babalola university. IJSER 2015;6(8):614–20.

[17] Ashraf RM, Mohie SS, Janet MS. Prediction of the behavior of reinforced concrete deep beams with web openings using the finite element method. Alex Eng J 2014;53(2):329–39.

[18] Mostafa M, Scott N, Yasser EI. Stress analysis of laminated glass with different interlayer materials. Alex Eng J 2012;51:61–7.