Gaurav Kumar et al. 2019, 7:6
ISSN (Online): 2348-4098
ISSN (Print): 2395-4752

International Journal of Science, Engineering and Technology

An Open Access Journal

Survey of a Fatigue Life Prediction for Safe and Economical Machine Shaft

M.Tech. Scholar Gaurav Kumar Asst. Prof. Ranjeet Kumar Asst. Prof. R.S. Sikarwar

Department Of Mechanical Engineering,
Vaishnavi Institute of Technology
Bhopal, MP, India

Abstract

The Finite Element Method is essentially a product of electronic digital computer age. Though the approach shares many features common to the numerical approximations, it possesses some advantages with the special facilities offered by the high speed computers. In particular, the method can be systematically programmed to accommodate such complex and difficult problems as non homogeneous materials, non linear stress-strain behavior and complicated boundary conditions. It is difficult to accommodate these difficulties in the Least Square Method or Ritz Method etc. an advantage of Finite Element Method is the variety of levels at which we may develop an understanding of technique**.**

Keywords: wind turbine; finite element analysis; high-strength bolts; fatigue life.

I. 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 depend-able 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 con-temporary years the aid of finite element in engineering was enormously increased Key factors in finite element analysis (FEA) are numerical computations with the intention of estimate all parameters and boundaries agreed.

A good effort 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 keep 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. Fatigues 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.

II. LITERATURE SURVEY

Lyashuk et al. (2019) increased the durability of technological equipment elements, forecasting of the resource and diagnostics of failures of the technical system. The basic regularities are analyzed and causes of the failure of the extruder's working body shaft are determined. For torque values M = 40.74 - 64.37 Nm and of the extruder's working body shaft, the stress-strain state of the contact surfaces of the keyhole of the extruder's shaft is calculated by the method of three-dimensional finite element modelling. The maximum values of the stress intensity σint(max),which arise on the edge of the key groove, are calculated. It was established that an increase in the distance of the key groove to the fillet of the working body shaft by 2.0 mm leads to a decrease in the maximum stresses on the edge of the key groove by 15.72%. The results of the research allow optimizing the geometry of the shaft.

Kanget. al (2019) studied the ultimate strength of high-strength bolts in the connection between the hub and main shaft in a 2.5 MW wind turbine with pretension effects, two kinds of finite element models are presented: a solid bolt model and a simplified bolt model. Theoretical calculation was used to compare these models with the results of finite element methods. The ultimate strength analysis results showed that the simplified bolt model was the most efficient and useful in terms of computational time and memory usage. Based on this study, the simplified bolt model was used in the fatigue calculation considering multiple random fatigue loads, and the fatigue life of high-strength bolts was determined by combining the S–N curve of the bolt, based on the Palmgren–Miner Linear damage accumulation hypothesis.

Jadhav et al. (2019) Cracks are most common defects which can develop due to fatiguein structure and machine elements. Fatigue cracks are responsible for catastrophic failures. Cracks are the threat to an uninterrupted operation and performance of the modern day machines. The crack occurs in rotating shafts where detection of cracks is difficult due to inspection difficulties. The problem of rotating cracked shafts has been investigated for a long time. Two main features have been recognized: (1) the local flexibility introduced by the crack in relation to the affected shaft sections; (2) the opening–closing phenomenon of the crack during rotation called as breathing mechanism of the crack. Various researchers have studied the response of cracked shafts. The work on the diagnosis of crack has been mainly based on vibration signature. The changes in the vibration response in the form of changes of frequency composition have been found to be some of the important crack indicators. These vibration-based techniques have been applied to a variety of engineering structures, such as beams, trusses, rotors, etc. In this work, the influence of transverse crack in a shaft is analyzed. Vibration parameters are obtained from a simulation employing the Finite Element Method using modal analysis for different crack depths. The unique relation between the natural frequencies and modes shapes with crack depths is employed to relate failure patterns of different points of the shaft.

Afolabi et al (2018) 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 of72.0 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.

Cangi et al. (2018) In this study, tolerance bands between the drum shaft and its two bearings were examined to develop a relationship between the fatigue life of the shaft and the interaction tolerances. Optimization of tolerance bands was completed in consideration of the fatigue life of the shaft as the cost function. The following methodology is followed: multibody dynamic model of a washing machine was constructed and used to calculate dynamic loading on the components. Then, these forces were used in finite element analyses to calculate the stress field in critical components which was used for fatigue life predictions. The factors affecting the fatigue life were examined to find optimum tolerance grade for a given test condition. Numerical results were verified by experimental observations.

M. K. et al. (2018) investigated stresses and deformation for the manufacturing the drive shaft of PTO shaft of Field marshal mini tractor. The shaft is connected with gear and pinion. This is the major important component to be taken into account while designing. The stress concentration and deformation in the shaft in the different regions were analyzed. In this paper, the part files and assembly are done by using CREO software and the analyzing is done by using an ANSYS 15.0 workbench. The results obtained by the stress analysis is found to be good agreement with analytical stress value and modal analysis i.e., stresses and deformation presented are within the limit which helpful for remanufacturing analysis.

Celik et al. (2018) describes a finite element method (FEM) based deformation simulation procedure for a power take off (PTO) shaft in an agricultural tractor. The agricultural tractor is a mobile power source in agricultural fields. The Agricultural tractor transmits power to the working implement through several systems independently. Most especially, rotary elements used in agricultural machinery take the required power and movement from the tractor take off (PTO) shaft. During this operation, the PTO shaft experiences a high dynamic loading condition such as excessive instant (impact) loading. This may cause an undesired failure case for the PTO shaft.

In order to prevent such undesired failures, loading condition and stress distribution on the component should be described properly; however, an accurate description of the structural stress distribution on the shaft becomes an important problem. In this content, a case study was carried out on a failed PTO shaft, as described in this paper. The aim of this case study is to exhibit the stress distribution on the PTO shaft through finite element analysis under a torsional loading case which may be considered as the main cause of the failure. Visual outputs from the simulation results revealed a better understanding of the failure zone on the shaft. The maximum equivalent stress magnitude obtained from the simulation was 632.08 [MPa] (which was lower than the fracture point) on the shaft under maximum PTO torque, however, it was concluded that the main reason for the failure was excessive shock torsional loading. This work contributes to further research into usage of numerical method based deformation simulation studies for the transmission elements used in agricultural tractors/machinery.

Shankaregowda et al. (2018) Composites offer significant improvements over current available materials for automobile due to their excellent mechanical characteristics and relatively low density. Fatigue life of composite drive shaft with viscous interface under variable torsional loads is analyzed. The s-n curve is obtained explicitly by using the micromechanical model of composites. The study indicates that the life of the composite depends on several factors, including fiber volume fraction, the relative shear rigidity of fiber to matrix, and a dimensionless parameter composed of interface viscosity, fiber radius, vibration frequency and shear modulus of matrix. Computational approach is adopted for analyzing the fatigue life of composite drive shaft. S-N curve will be developed using a finite element analysis depends on the material directions of the layers. S-Glass fiber and epoxy composite material is considered for designing the drive shaft.

Mehta (2017) find out the fatigue life of axle which is used in race car. An axle is a nonrotating member, usually of circular cross-section for transmitting power. It is supported by hub. It is subjected to torsion, and bending in combination. The stress on the axle at a particular point varies with the irregularities of road by introducing fatigue. For finding out the fatigue life I am using Goodman method for analytical approach and FEA analysis done on ANSYS workbench software.

Yan et al. (2017) had done the parametric modeling of the torsion shaft with Solid works, and the model was transferred to the ANSYS Workbench through the data interface. A parameterized model of torsion shaft was developed based on Solid works and imported into ANSYS Workbench by data interface. At the same time, based on the ANSYS Workbench, the transient stress analysis and fatigue analysis of the parameterized model are carried out. On this basis, the influence of different structures on the fatigue life of the torsion shaft is analyzed, and the structure of the torsion shaft is improved. The optimization of structure of tracked vehicle components is explored, and the study results can widely extended to other engineering field.

III. FATIGUE LIFE ESTIMATION TECHNIQUES

The idea behind the selection of project, in today's structural design, the analysis of the crack within the structure is an important application if the damage tolerance and durability of structure and components are to be predicted. As parts of engineering design process engineers have to assess not only how well the design satisfies the performance requirements but also how durable product will be over its life cycle. Often cracks cannot be avoided in structures; however the fatigue life of the structure depends on the location and size of these cracks. In order to predict the fatigue for any component a fatigue life and crack growth study needs to be performed. Fatigue is a stochastic process. For this reason fatigue life estimation has become the important aspect for the replacement of worn out components.

IV. SHAFT

Shaft is a common and important machine element. It is a rotating member, in general, has a circular cross-section and is used to transmit power. The shaft may be hollow or solid. The shaft is supported on bearings and it rotates a set of gears or pulleys for the purpose of power transmission. The shaft is generally acted upon by bending moment, torsion and axial force. Design of shaft primarily involves in determining stresses at critical point in the shaft that is arising due to aforementioned loading. Other two similar forms of a shaft are axle and spindle. Axle is a non-rotating member used for supporting rotating wheels etc. and do not transmit any torque. Spindle is simply defined as a short shaft. However, design method remains the same for axle and spindle as that for a shaft.

V. CONCLUSION

The fatigue analysis will be applied to uniaxial and biaxial loading conditions for geometrically stepped shaft of diameter 30 mm and 20 mm with notch radius 5mm. The present work will study the numerically simulation analysis of the fatigue behaviour of stepped shaft made of Carbon steel SAE 1045_390_QT under a biaxial loading (torsion bending moment) and uniaxial loading (only bending moment case), (only torsion case). This work will include the complete analysis of stepped shaft by using ANSYS WORKBENCH 15.

REFERENCES

[1]. Ali NaeemOdaa and Yassr Y. Kahtan (2017), Numerical Simulation and Life Prediction of Stepped Shaft Under Biaxial Fatigue Loading, International Journal of Scientific & Engineering Research, 8(11), 301-309.

[2]. Ali NaeemOdaa, Yassr Y. Kahtan (2017), Numerical Simulation and Life Prediction of Stepped Shaft Under Biaxial Fatigue Loading, International Journal of Scientific & Engineering Research Volume 8, Issue 11,pp 1-17

[3]. B. Engel, Sara Salman Hassan Al-Maeeni (2017), Failure Analysis and Fatigue Life Estimation of a Shaft of a Rotary Draw Bending Machine, International Journal of Mechanical and Mechatronics Engineering, 11 (11), pp. 1785-1790.

[4]. DeepanMarudachalam M.G, K.Kanthavel, R.Krishnaraj (2011), Optimization of shaft design under fatigue loading using Goodman method, International Journal of Scientific & Engineering Research, Volume 2, Issue 8,pp 1-5.

[5]. G. Ragul, G. Kalivarathan, V. Jayakumar, Praveen Maruthur , Into Jacob, S. Naveen Kumar (2016), An Analytical Investigation on Design and Structural Analysis of Cam Shaft using Solid Works and Ansys in Automobiles, Indian Journal of Science and Technology, Vol 9(36) pp,1-9.

[6]. H. KursatCelik, Recep CINAR, Gokhan KUNT, Allan E.W. RENNIE, Mehmet UCAR,Ibrahim AKINCI (2018), Finite Element Analysis of a PTO Shaft Used in an Agricultural Tractor, Ergonomics International Journal, Volume 2 Issue 3, pp 1-6.

[7]. Hariom, Vijoy Kumar, Dr.Chandrababu D. (2016), A Review of Fundamental Shaft Failure Analysis, International Research Journal of Engineering and Technology, 3(10), 389-395.

[8]. Ji Kang, Haipeng Liu, Deyi Fu (2019), Fatigue Life and Strength Analysis of a Main Shaft-to-Hub Bolted Connection in a Wind Turbine, 12(1), 7-16.

[9]. M. Cangi, T. Dolar, C. Ersoy, Y. E. Aydogdu, A. I. Aydeniz, A. Mugan (2015), Optimization of Tolerance Grades of a Bearing and Shaft Assembly in a Washing Machine with Regard to Fatigue Life, International Journal of Aerospace

and Mechanical Engineering Vol:12, No:8,pp 791-797.

[10]. M. K. Khunti (2018), Stress Analysis Of Shaft Of Pto Drive Using Finite Element Analysis, International Journal of Advance Engineering and Research Development, Volume 5, Issue 04, pp 2535-2539.