The Eurasia Proceedings of Science, Technology,
Engineering & Mathematics (EPSTEM)
ISSN: 2602-3199

The Eurasia Proceedings of Science, Technology, Engineering & Mathematics (EPSTEM), 2019

Volume 7, Pages 315-320

IConTES 2019: International Conference on Technology, Engineering and Science

A Correlation Study of an FEA Method Developed for Heavy Duty Driveshaft Applications

Muzaffer KASABA
Tirsan Kardan A.Ş

Sedat TARAKCI
Tirsan Kardan A.Ş

Efe ISIK
Tirsan Kardan A.Ş

Onur AKKAS
Tirsan Kardan A.Ş

Abstract: The driveshafts are responsible for transmitting power and rotational movement from the engine to the rear axles. In order to fulfill high torque transmission requirements, driveshaft components are generally produced from steel by hot forging process. For steel components, reduction of the weight is limited by geometrical alterations due to boundary conditions, functionality requirements etc. For this reason, verification method of design iterations must be conducted accurately. Although verification on physical component is much reliable, regarding high material, die, tooling and machining costs an alternative verification method is needed. Generally finite element analysis (FEA) is preferred as design verification method. The major phenomenon is to build the simulation as similar as possible with real conditions in order to have accurate results. In this study, an FEA method was developed for design verification of heavy duty driveshafts. The results from FEA analysis of heavy duty driveshaft were compared with the results from the bench tests and a correlation study was conducted to reveal the consistency of developed FEA method.

Keywords: Finite element method, Data acquisition, Stress, Strain

Introduction

Today, the expectations in the automotive industry are increasing day by day. Accordingly, the lightweight driveshaft design and implementation in drivetrain systems is one of the biggest expectations. Increasing ecological concerns have led automobile manufacturers around the world to seek new solutions. Developing lightweight but high-strength components for ground vehicles that will led to less fuel consumption without compromising safety and comfort is one of the major goals of automobile manufacturers.

For steel components, reduction of the weight is limited by geometrical alterations due to boundary conditions. Therefore, reduction of the weight should be carried out very carefully. Verification of the developed lightweight designs is the most important step of the studies. Although design verification by experimental methods is the best solution for validating the design, it has some disadvantages like test costs, long testing durations and labor expenses. For these reasons, there is a need to create a finite element model which can be used in place of the experimental method and simulate the actual state of the driveshaft in the most appropriate way.

Within the scope of the study, the main technical characteristics of the relevant cardan shafts (yield point, ultimate torsional strength, torsional stiffness value) were determined. Then, finite element models were established, and non-linear static analyzes were performed according to the determined mechanical properties.

After the non-linear FE analysis, stress distributions on the heavy duty driveshaft were revealed. In parallel, data acquisition studies were conducted. By performing static torsion test, the strain data from the each sub-components of the heavy duty driveshafts were collected and the regions where the stress lines and points were concentrated revealed. Finally, finite element analyzes results and the test results were compared and the results were presented with percentage deviations.

Finite Element Method

The finite element method (FEM), or finite element analysis (FEA), is a computational technique used to obtain approximate solutions of boundary value problems in engineering. Boundary value problems are also called field problems. The field is the domain of interest and most often represents a physical structure. (Kasaba, 2017)

The field variables are the dependent variables of interest governed by the differential equation. The boundary conditions are the specified values of the field variables (or related variables such as derivatives) on the boundaries of the field. For simplicity, at this point, we assume a two-dimensional case with a single field variable φ (x, y) to be determined at every point P(x, y) such that a known governing equation (or equations) is satisfied exactly at every such point. (Kasaba, 2017)

A General procedure of finite element analysis is as follows:

Preprocessing

Define the geometric domain of the problem.
Define the element type(s) to be used.
Define the material properties of the elements.
Define the geometric properties of the elements (length, area, and the etc.)
Define the element connectivities (mesh the model).
Define the physical constraints (boundary conditions). Define the loadings.

Solution

Computes the unknown values of the primary field variable(s)
Computed values are then used by back substitution to compute additional, derived variables, such as reaction forces, element stresses, and heat flow.

Post processing

Postprocessor software contains sophisticated routines used for sorting, printing, and plotting selected results from a finite element solution.

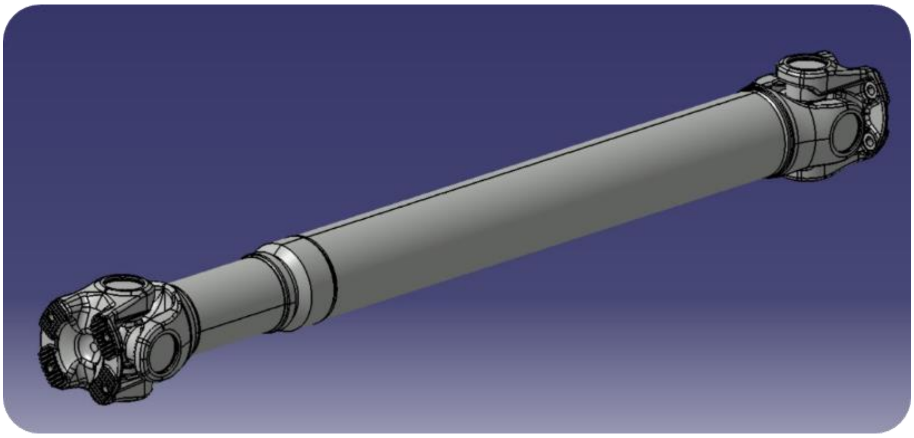


Figure 1. CAD Model of the Driveshaft

The driveshaft was modeled by using Catia V5 2014 software as shown in Figure 1. After the solid model was obtained, a finite element model was generated by using pre-processing steps as shown in Figure 2, and a non-linear finite element analysis was performed by considering the vehicle interface restrictions and contact definitions. The boundary conditions are given in Table 1.

However, in the design stage, the additional costs such as equipment, prototype and personnel required for the implementation of the experimental method, as well as time loss occur. For these reasons, there is a need to create a finite element model which can be used in place of the experimental method and simulate the actual state of the driveshaft in the most appropriate way. In this study, a finite element analysis method has been developed to meet these expectations.

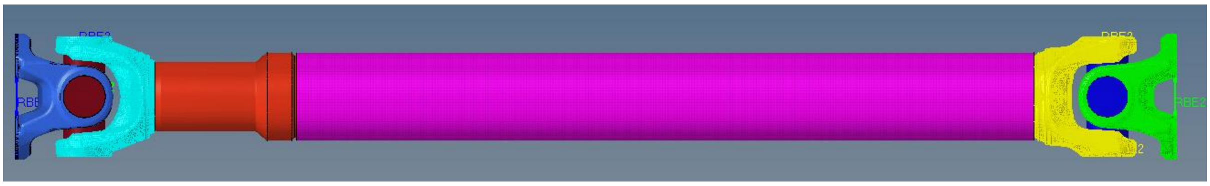


Figure 2. Finite Element Model

Table 1. Boundary conditions

Fixed B.C	**Moment**
B.C 1,2,3,4,5,6 (fixed) RBE2s are bounded to master node	B.C 2,3,5,6 (fixed) RBE2s are bounded to master node
	Moment is bounded to master node (RBE2)

Data Acquisition Study

In order to specify the strain levels on specific areas of driveshaft sub-components at different levels of torsional loads, the driveshaft test sample was installed on a static torsion test bench which is depicted in Figure 3. Before the installation of the test sample onto the test bench, the test sample was instrumented by following steps:
1. Data collection regions were determined.
2. Surface was prepared with handle sandpaper.
3. Adhesive was applied to the surface which was cleaned with alcohol and degreaser and rosette type strain-gauge was bonded.
4. Terminals were placed on the paper tapes.
6. The quarter bridge connection was generated by soldering the strain gauges wires onto the terminals.
7. Connections checked with ohmmeter (120 Ω).
8. Strain-gauge was covered with liquid silicon for protection.

Static torsion test was conducted at different levels of torque loads and at each torque load strain data of specific areas on driveshaft were recorded for stress calculations.

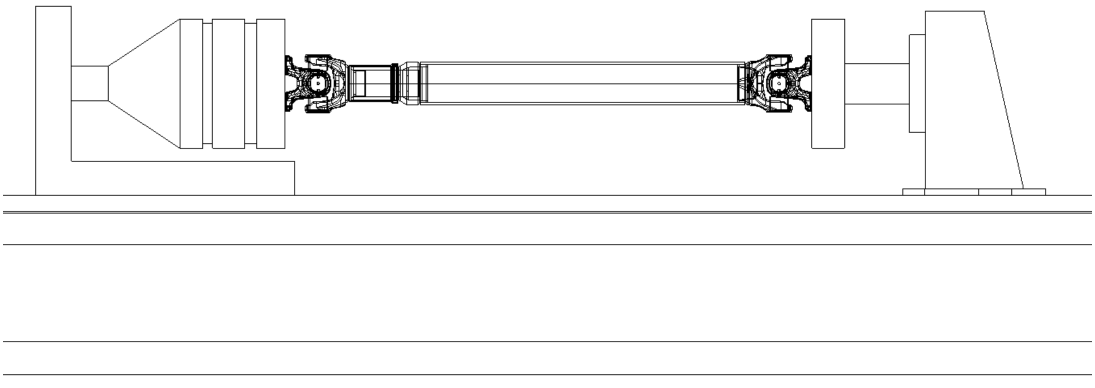


Figure 3. Static torsion test system

Results

Within the scope of this study, it was aimed to generate stress map of heavy-duty driveshaft by measuring strain at specific points / areas (Figure 4) and compare the results with the stress values obtained from finite element analyzes. After installation of test sample onto the test bench, LMS Bridge calibration was done and then static torsion test were performed in clock-wise and counter clock-wise directions with 500 Nm load steps up to 20 kNm torsional load. During static torsion test, strain data was collected using 21 different channels at the specified torque levels from the driveshaft sub-components. By using strain data, equivalent stress (Von Mises) was calculated. The strain values obtained from the driveshaft sub-components are presented in Figure 5.

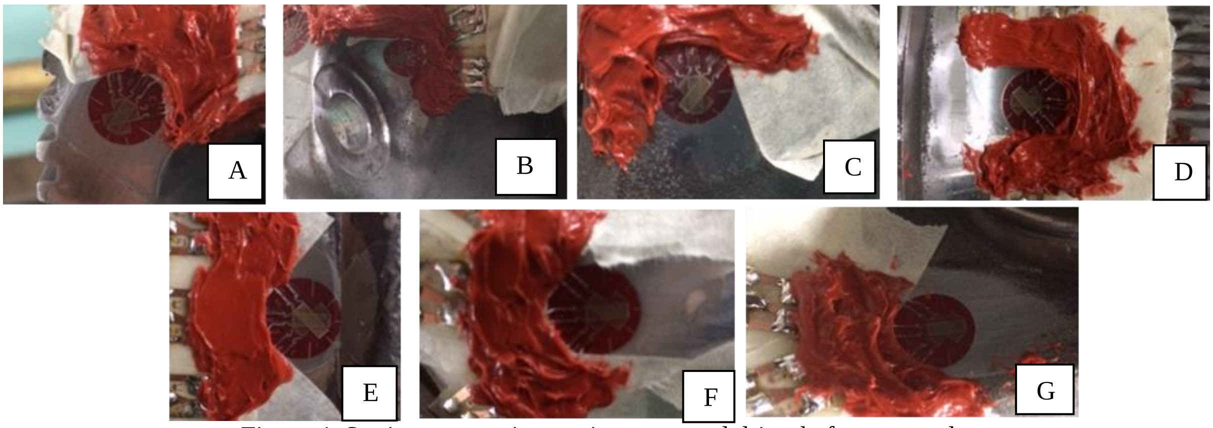


Figure 4. Strain gauge points on instrumented driveshaft test sample

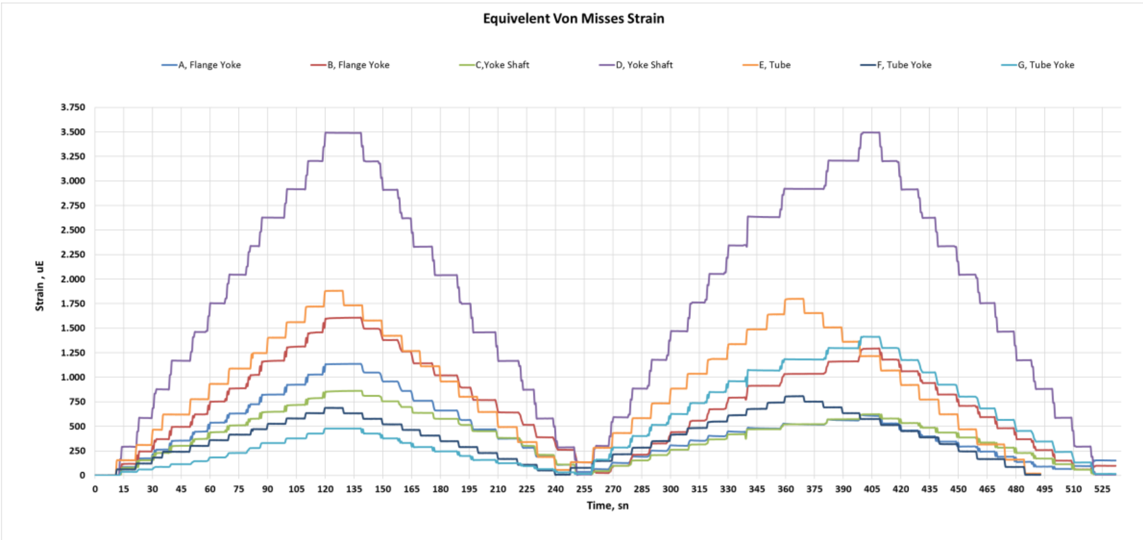
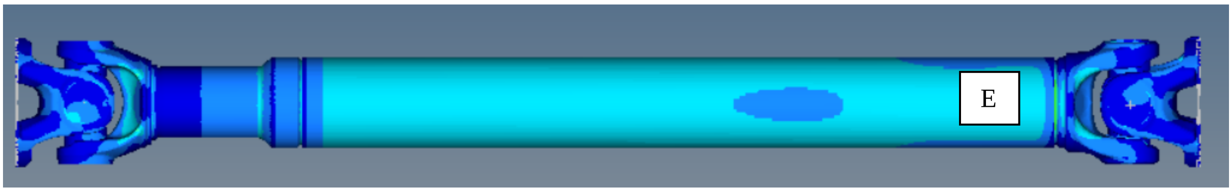


Figure 5. Data acquisition strain results

Finite element models are constructed in accordance with the testing conditions. All driveshafts were analyzed under 20 kNm torque load. The FEA results (Figure 6) are compared with data acquisition results and the comparison is given in Table 3.

When the percentage deviation values were examined in Table 3, it was determined that the difference between the test results and the finite element analysis results did not exceed 10%.



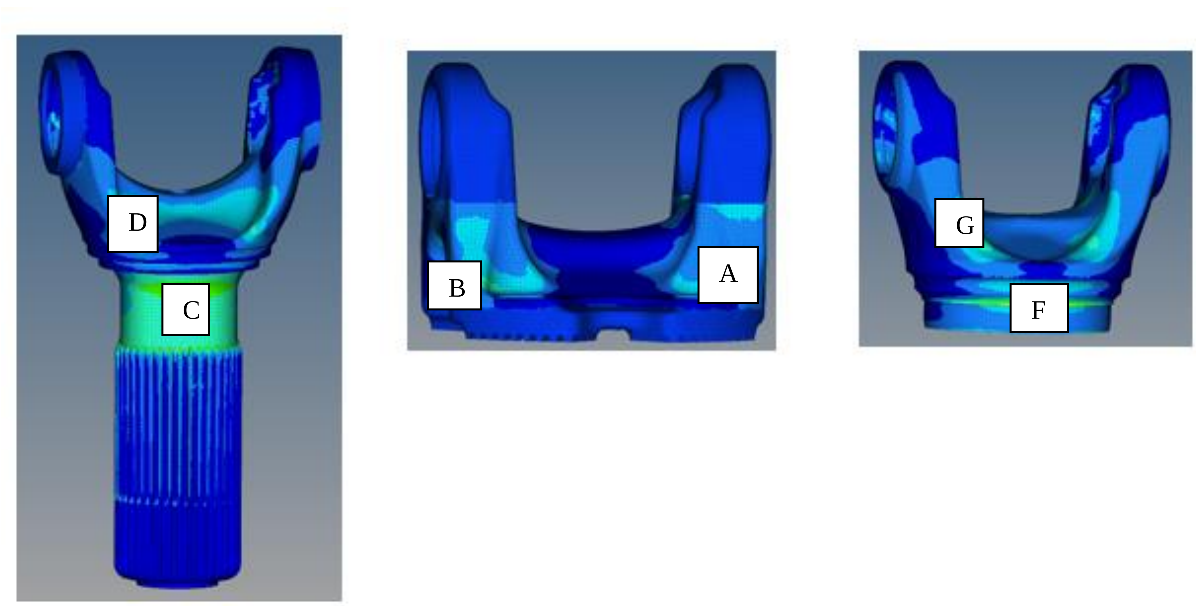


Figure 6. Finite element analysis results

Table 3. Static torsion test & fea results comparison table

	A	B	C	D	E	F	G
% Deviation	9,78	6,29	10,32	3,74	7,67	8,16	9,15

Conclusion

As a result, it has been shown that proposed non-linear finite element model can be used safely without the need for a driveshaft test sample in the design stages with a maximum allowable deviation of 10 %. In this way, production and test costs will be reduced which will create a competitive advantage within the market.

Recommendations

This study is conducted for heavy duty driveshafts only, therefore our study can be extended to all driveshaft product families in the future work.

Acknowledgements or Notes

We would like to thank Tirsan Kardan R & D Center and Test Center for their support in the studies.

References

Kit, H., Khaj, M., & Mykhaskiv, V. (1996). Analysis of dynamic stress concentration in an infinite body with parallel penny-shaped cracks by BIEM. Engineering Fracture Mechanics, 55(2), 191–207.

Noda, N. (1995). Stress concentration factors for round and flat test specimens with notches. International Journal of Fatigue, 17(3), 163–178.

Bhattacharya, S., & Kumar, A. (1995). Modelling of rotational factor in notched bend specimen under general and local yield situation. Theoretical and Applied Fracture Mechanics, 24(1), 33–46.

Davies, D., & Jenkins, S. (2012). Influence of stress and environment on the fatigue strength and failure characteristics of case carburised low alloy steels for aerospace applications. International Journal of Fatigue, 44, 234–244.

Shen, W., & Choo, Y. S. (2012). Stress intensity factor for a tubular T-joint with grouted chord. Engineering Structures, 35, 37–47.

Cheng, B., Qian, Q., & Zhao, X.-L. (2015). Stress concentration factors and fatigue behavior of square bird-beak SHS T-joints under out-of-plane bending. Engineering Structures, 99, 677–684

She, C., & Guo, W. (2007). Three-dimensional stress concentrations at elliptic holes in elastic isotropic plates subjected to tensile stress. International Journal of Fatigue, 29(2), 330–335.

Ndiaye, A., Hariri, S., Pluvinage, G., & Azari, Z. (2007). Stress concentration factor analysis for notched welded tubular T-joints. International Journal of Fatigue, 29(8), 1554–1570.

Yu, P., Guo, W., She, C., & Zhao, J. (2008). The influence of Poisson's ratio on thickness-dependent stress concentration at elliptic holes in elastic plates. International Journal of Fatigue, 30(1), 165–171.

Zheng, M. (1997). Analysis of the stress concentration factor for a shallow notch by the slip-line field method. International Journal of Fatigue, 19(3), 191–194.

Fielder, R., Millwater, H., Montoya, A., & Golden, P. (2019). Efficient estimate of residual stress variance using complex variable finite element methods. International Journal of Pressure Vessels and Piping, 173, 101–113.

Paroissien, E., Lachaud, F., Silva, L. F. D., & Seddiki, S. (2019). A comparison between macro-element and finite element solutions for the stress analysis of functionally graded single-lap joints. Composite Structures, 215, 331–350.

Hu, H., Zou, Z., Jiang, Y., Wang, X., & Yi, K. (2019). Finite element simulation and experimental study of residual stress testing using nnlinear ultrasonic surface wave technique. Applied Acoustics, 154, 11–17.

Kasaba, M. (2017). Numerical Analyses Of Structural Acoustic Systems Based On Monte Carlo Simulations And Modal Impedances. Graduate School of Natural and Applied Sciences of Dokuz Eylül University

Author Information

Muzaffer Kasaba
Tirsan Kardan A.Ş
Organize Sanayi Bölgesi 3. Kısım M.Kemal Bulvarı No:15
Manisa, 45030 Türkiye
Contact E-mail: *m.kasaba@tirsankardan.com.tr*

Sedat Tarakci
Tirsan Kardan A.Ş
Organize Sanayi Bölgesi 3. Kısım M.Kemal Bulvarı No:15
Manisa, 45030 Türkiye

Efe Isik
Tirsan Kardan A.Ş
Organize Sanayi Bölgesi 3. Kısım M.Kemal Bulvarı No:15
Manisa, 45030 Türkiye

Onur Akkas
Tirsan Kardan A.Ş
Organize Sanayi Bölgesi 3. Kısım M.Kemal Bulvarı No:15
Manisa, 45030 Türkiye