Strength based Design Analysis of A Damaged Engine Mounting Bracket Designed for A Commercial Electric Vehicle

H. Kursat CELIK[1]*, Hakan ERSOY[2], Ayla DOĞAN[2], Gokhan ERAVCI[1], Allan E.W. RENNIE[3], Ibrahim AKINCI[1]

[1,*] *Dept. of Agricultural Machinery and Technology Engineering, Akdeniz University, Antalya, Turkey*
[2] *Dept. of Mechanical Engineering, Akdeniz University, Antalya, Turkey*
[3] *Engineering Dept., Lancaster University, Lancaster, United Kingdom*

**Corresponding author*	*: Dr H. Kursat CELIK*
e-mail	*: hkcelik@akdeniz.edu.tr*
Tel	*: +90 242 310 65 70*
Fax	*: +90 242 310 24 79*
Address	*: Dept. of Agricultural Machinery and Technology Engineering, Akdeniz University, 07070, Antalya, Turkey*

Abstract

This study describes a strength-based design analysis protocol by means of finite element analysis (FEA) for a damaged engine mounting bracket in a converted electric vehicle. The mounting bracket considered in the study is a product specifically designed and manufactured for a converted electric vehicle and failed during conventional operation of the vehicle. Thus, design improvement/revision (re-design) on the bracket geometry was investigated. In this context, in order to prevent such undesired failures, strength-based design features such as deformation behaviour and stress distribution under projected loads on the bracket should be properly described, however, an accurate description of these features of the bracket may become a complex experimental problem to be solved by designers. This study described redesign of the strength-based design features of the engine mounting bracket through finite element analysis under torsional loading generated by the electric engine that was determined to be the reason for the failure and thus the motivation to realise a safer design. Visual and numerical results obtained from the simulation revealed a clear understanding of the failure behaviour of the bracket and therefore enabled an informed approach to the re-design stage. The initial FEA of the part design mapped the damage regions on the part geometry and indicated the stress magnitudes that were in excess of the material's stress limits. The comparison of the failure plots and numerical data obtained from this initial FEA and physically damaged part was consistent. This concluded that the FEA satisfactorily exhibited the deformation behaviour and the main reason for the failure was insufficient geometry thickness and notch effect against predefined loading conditions. Therefore, the main design improvement was realised on these geometric features. Subsequently, the final FEA highlighted that the re-design would enable safe operation. This work contributes to further research into usage of numerical method based deformation simulation studies for the mounting elements used in customised electric vehicles.

Keywords: Electric Vehicle, Engine Mounting Bracket, Product Design, Failure Analysis, Finite Element Analysis

[1,]Huseyin Kursat CELIK, PhD, Assoc. Prof.*	*hkcelik@akdeniz.edu.tr*	*ORCID: 0000-0001-8154-6993*
[2]Hakan ERSOY, PhD, Assoc. Prof.	*hakanersoy@akdeniz.edu.tr*	*ORCID: 0000-0001-8770-6150*
[2]Ayla DOĞAN, PhD, Assoc. Prof.	*ayladogan@akdeniz.edu.tr*	*ORCID: 0000-0002-9060-3053*
[1]Gokhan ERAVCI, MSc Candidate	*gkhneravc@gmail.com*	*ORCID: 0000-0001-7920-4767*
[3]Allan E.W. RENNIE, PhD, Prof.	*a.rennie@lancaster.ac.uk*	*ORCID:0000-0003-4568-316X*
[1]Ibrahim AKINCI, PhD, Prof.	*iakinci@akdeniz.edu.tr*	*ORCID:0000-0002-0057-0930*

Word Counts	: Approx. 2900 (Except references)
Number of Figures	: 5
Number of Tables	: 0

Introduction

Growth in the world population is increasing environmental and transportation problems as a consequence of the movement of people in their daily routines. The vehicles used for such transportation mostly have internal combustion engines. Consequently, every new internal combustion vehicle that goes into use increases the volume of harmful gases released into the environment (Kerem, 2014). Additionally, it is a well-known issue that oil reserves are not finite and this leads to the search for alternatives to transportation vehicles with internal combustion engines. Thus, there is an increased demand for alternative energy-based vehicles such as the electric car that can provide up to 80% energy saving compared to conventional vehicles equipped with internal combustion engines. In fact, the interest for electric vehicles has been clear since the 1960s due to their silent operation capabilities in addition to environmentally-friendly features (Zeraoulia et al., 2006). Therefore, parallel to the development in machine design and manufacturing technology, the number of electric energy-based vehicles has been increasing and the use of hybrid and electric vehicles has been rapidly spreading in the market (Wada, 2009). Additionally, the conversion of existing internal combustion vehicles to electric vehicles has gained importance in recent years. As an alternative to purchasing a new electric vehicle, it is now possible for private individuals or organisations to build an electric vehicle with ease - and at low cost - with reliable, off-the-shelf electric vehicle components (Keoun, 1995). In up-to-date technology, electric motors designed for high-capacity torque and speed, which are reduced in weight and can be easily assembled or replaced, are available. These electric engines stimulate the vehicle drive system, either with the gearbox or directly connected to the differential systems. Most especially in converting operations, customised part design procedures related to engine replacement and confirmation of the safe design features has become an important issue as experiencing part failure is a high possibility during these initial customisation procedures. In a converted electric vehicle, the engine is the main power source resting on the mounting brackets which are custom designed and connected to the chassis of the car. Vibrations, engine torque, materials and fatigue features of an engine bracket has been continuously a concern which may lead to structural failure if the resulting vibrations and stresses are severe and excessive (Ghorpade et al., 2013). In this context, it is clear from the consumer perspective that durability and reliability of a product are the most important issues. Reliability is the measure of unanticipated interruptions or unexpected failures during customer use (Subbiah et al., 2011).

Failure analysis of damaged components in mechanical systems has been a necessary procedure since the first mechanical systems were designed. Engineers have adapted the experimental, analytical and numerical methods in order to realise the analysis procedures for damaged component re-design, however, in the case of larger dimensional size and complexity of the engineering problem, it was understood that analytical and experimental methods might uncover significant problematic aspects. context such, numerical methods-based advanced computer aided engineering applications may provide the information that can be employed in the design improvement activities. Most especially, finite element method (FEM) based engineering analysis (FEA) is a useful technique and very popular in solving complex design/engineering problems. Therefore, FEA has been widely used in various engineering fields, most especially for structural strength investigations of mechanical components. Structural stress analysis realised in order to map the stress distribution on a structural component is one of the most important aspects of these types of strength investigations conducted (Pardhi & Khamankar, 2014). The analyses made in order to prevent such undesired failures, strength-based design features such as deformation behaviour and stress distribution under projected loads on the engine assembly components should be described properly, however, an accurate description of these features of the components (such as engine mounting bracket) become a complex experimental problem to be solved by designers.

In this paper, strength-based design analysis of a failed engine mounting bracket designed for a commercial electric vehicle was introduced. A FEM based deformation simulation procedure for the failed bracket has been discussed in detail. Visual outputs from the simulation results are presented in order to provide a better understanding of the failure tracks on the bracket and subsequent re-design operations. Suggestions for preventing similar failures on the bracket have also been discussed.

Application Algorithm

Within the scope of the study, an application algorithm has been developed and the algorithm steps have been followed in a sequence. This sequence starts with a decision for strength-based design analysis and should end with an improved re-designed product. This algorithmic approach helps the user in the standardisation of a computer aided design and engineering technique-based design analysis study for similar requirements. Fundamentally, the algorithm can be divided into three core operational sections to be fulfilled: Model digitalisation (CAD modelling) (1), FEA set up and internal verification (2), and re-design operation and confirmation (3). The application algorithm is shown in Figure 1.

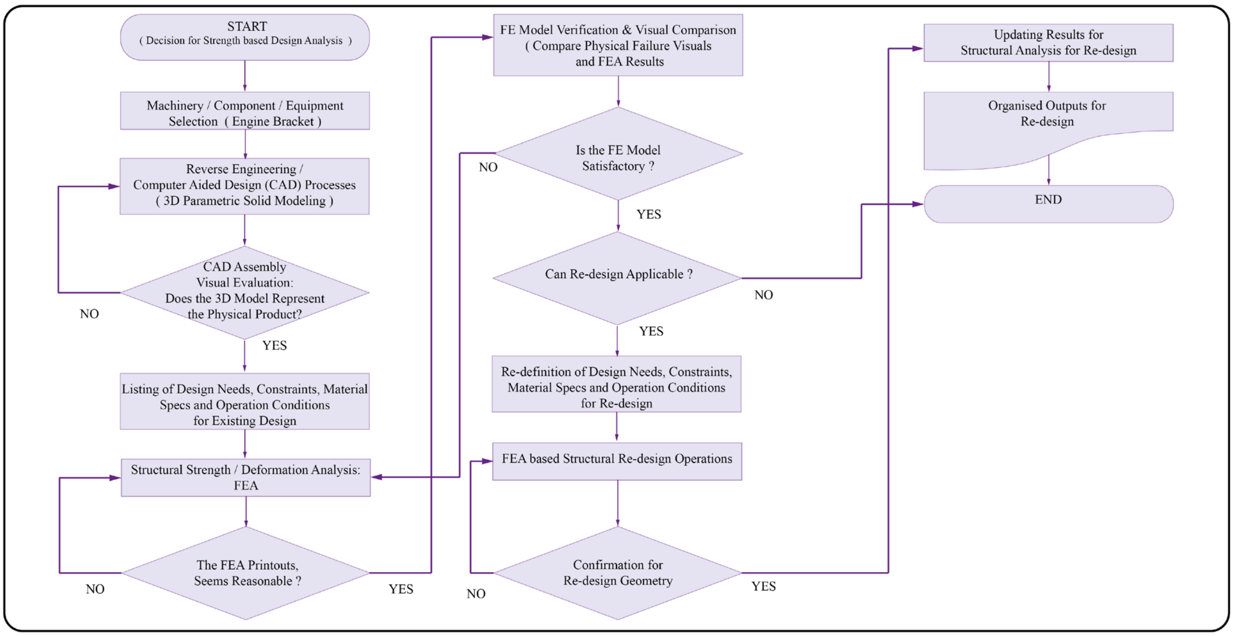


Figure 1. Application algorithm

Failed Engine Mounting Bracket

In this study, a failed engine mounting bracket specifically designed for a commercial electric vehicle has been considered. The bracket experienced failure during conventional operation of the vehicle following an operational period of 50 000 km driving. The same failure problem was reported for eight other identical vehicles. At the initial inspections, cracks and crack follower fracture signs were clear on the bracket. Initial observations of the bracket led us to consider an excessive torsional loading case, and/or design faults such as insufficient part thickness and notch radius in the model geometry; however, it was not possible to predict the exact deformation behaviour that caused the failure at the initial visual investigation. In order to clarify the loading effect and structural stress distribution at the failure zones on the bracket, a FEA was utilised with due consideration of critical material properties such as yield and fracture stress points of the bracket. Some technical dimensions of the bracket geometry and visuals taken from the failed bracket are shown in Figure 2.

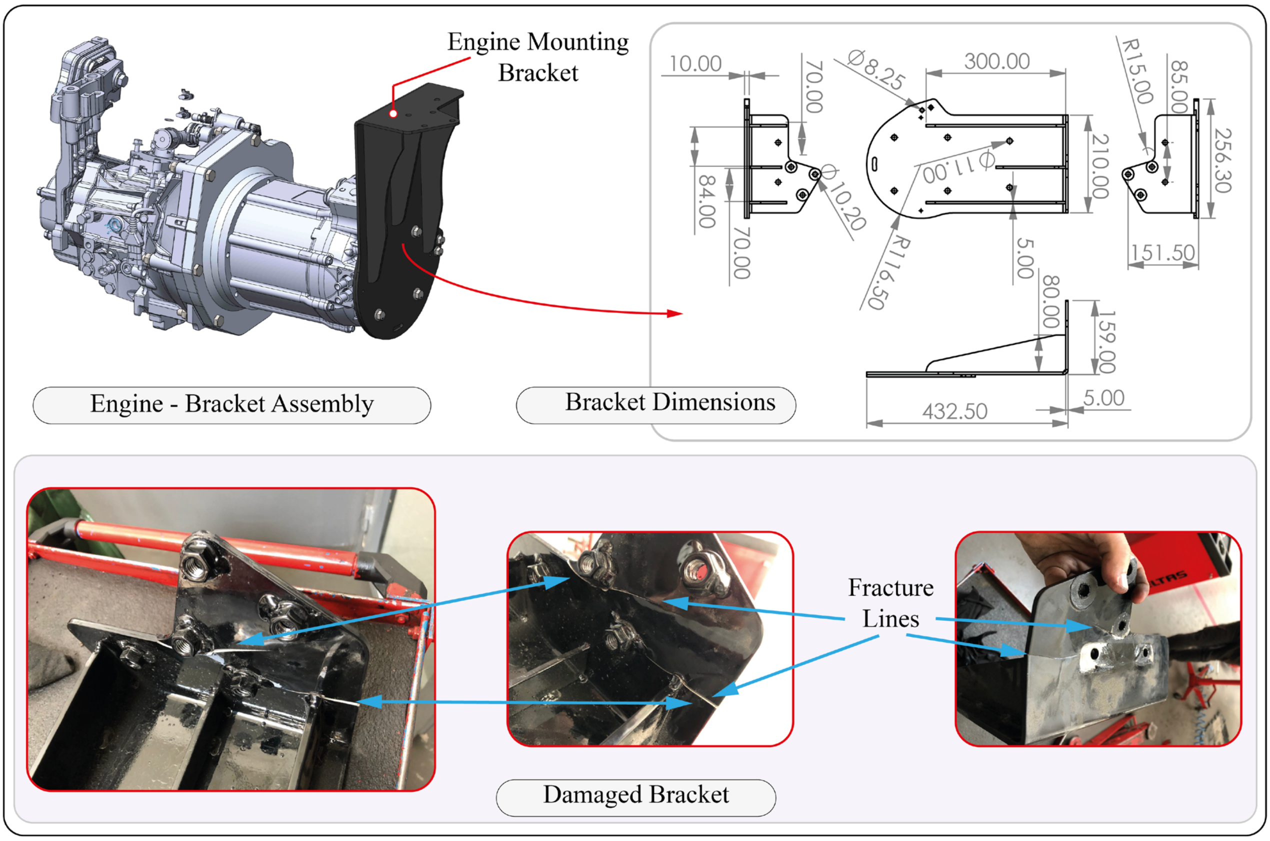


Figure 2. Technical dimensions and failure images of the engine mounting bracket

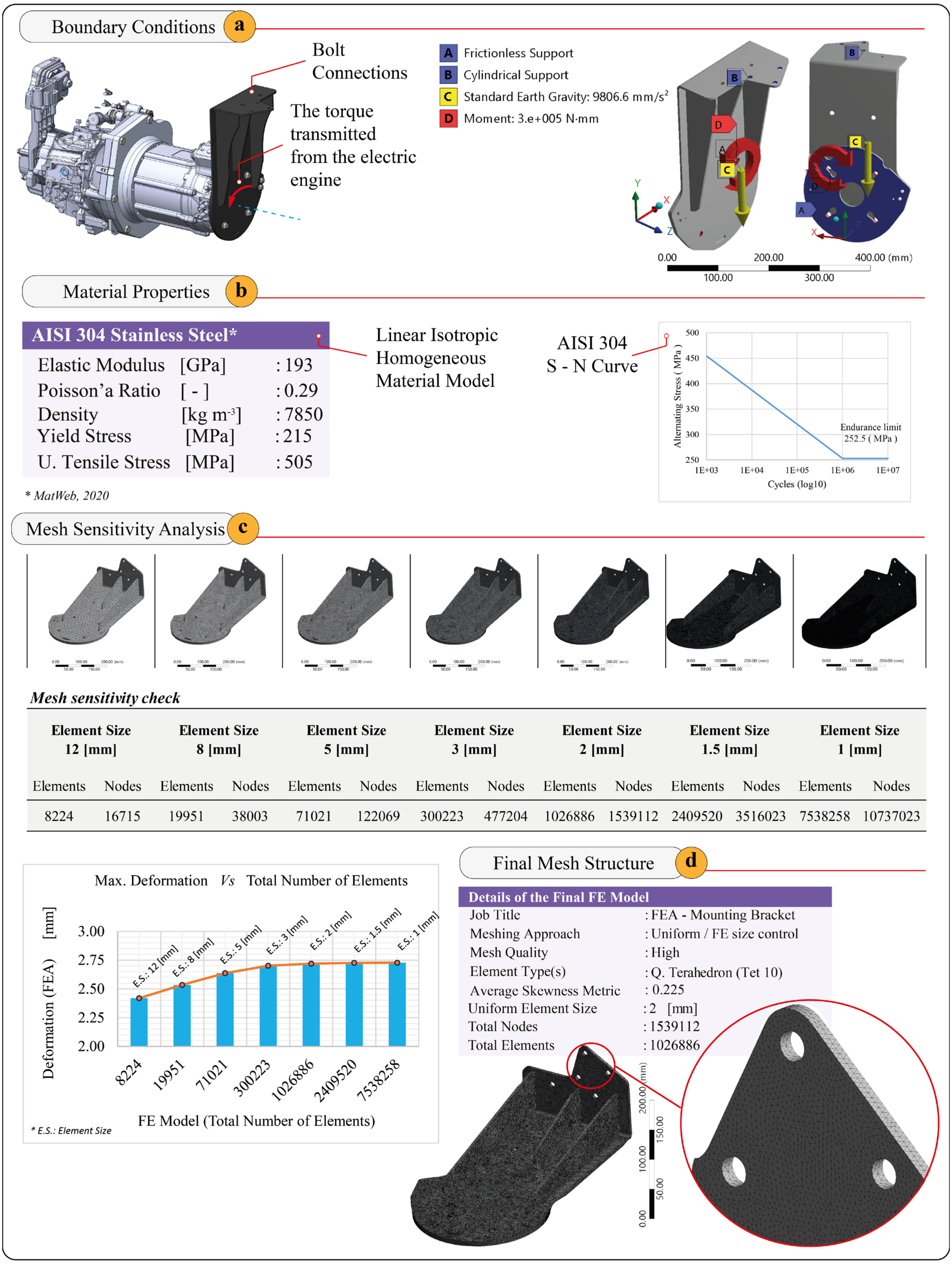
FEA of the Initial Design

The main loading source for the bracket was the torque produced by the electric engine, thus, the FEA was set up to simulate deformation behaviour and equivalent stress distribution on the bracket under torsional loading. The mass of the electric motor was 48 kg and the torque magnitude produced by the engine and transmitted to the wheels via the gearbox was 300 N·m at 3500 min^{-1}. In consideration of these initial parameters, the maximum torque of 300 N·m applied to the bracket was obtained from the engine catalogue data. Losses in torque magnitudes during transmission was ignored in order to evaluate maximum loading condition.

At the first stage, a reverse engineering approach-based 3D parametric solid modelling operation was carried out for existing design of the bracket. Solid modelling procedures were carried out using SolidWorks 3D parametric solid modelling software. The FEA simulations were carried out via ANSYS Workbench commercial FEA code. The material assigned in the production of the mounting bracket was AISI 304 stainless steel. Isotropic homogeneous elastic material model and linear static loading assumptions were considered in the FEA setup (MatWeb, 2020). The Finite Element (FE) model (mesh structure) of the bracket was created using the meshing functions of the ANSYS Workbench (ANSYS Product Doc., 2017).

In order to obtain an accurate result from a FEA, assigning an appropriate element size for creation of the Finite Element (FE) Model (mesh structure) is a critical decision, since the FE models are created from CAD representations and the fact is that the FEA mesh structure indicates an approximate geometry. In this context, creating accurate FE models most especially for complex CAD geometries with high NURBS (Non-Uniform Rational B-Splines) may become a discretisation problem in the simulation. Recent research such as 'Isogeometric Analysis' may promise an effective solution to this problem and simplify a specific design analysis algorithm (Hughes et al., 2005; Nguyen et al., 2015), however, here, the geometry of the engine bracket does not have complex geometric features (relatively) and the mesh density can be considered as a significant metric in order to control accuracy of the FE model in this study. Ideally, smaller elements can efficiently represent the solid model in a FEA, however, if the element size is too small, a larger amount of computer time may be required to solve the problem (Celik H.K. et al., 2018). In this regard, a sensitivity check was utilised in determination of the appropriate element size mesh for the bracket geometry. In the initial FEA set up, the loading scenario of the bracket was run with various element sizes (from coarse to fine) and the correlation between element size and deformation results obtained from the simulation were evaluated. The mesh sensitivity evaluation indicated that the minimum element size was 2 mm in order to sufficiently represent the model geometry of the bracket with an acceptable computation time. Additionally, skewness metric of the FE model with the element size of 2 mm was checked. The average skewness values of 0.225 was calculated which was an indication of an excellent mesh quality for the FE model (ANSYS Product Doc., 2019; Brys et al., 2004). Finally, a uniform element sizing strategy was utilised in creating the FE model (mesh structure) of the bracket. The boundary conditions, material properties, mesh sensitivity analysis, details of the final mesh structure, and how they are utilised in the FEA set up of the bracket loading scenario are shown in Figure 3.

In a failure analysis activity, carrying out investigations on the material damage mechanism of the damaged components under consideration of numerical methods and fracture mechanics is an important issue. As such, utilising a 2D and 3D crack evolution algorithm and describing a specific failure model may play a critical role in order to understand and describe the failure mechanism (Areias et al., 2016, 2018; Ganjiani & Homayounfard, 2021). However, such complicated failure models in geometry and material deformation definitions was not considered in this study. Crack occurrence and propagation issues on a damaged zone of the bracket were also not discussed. The main failure occurrence assumption considered in this study is based on von Mises failure stress criterion that highlights the failure when the damage parameter (absolute stress magnitude) reaches a critical value (yield stress point), hence the failure evaluations were conducted through von Mises failure stress criterion. A damage model for predicting ductile fracture can be utilised for detailed material fracture analysis of the bracket in future works.



Element Size 12 [mm]		Element Size 8 [mm]		Element Size 5 [mm]		Element Size 3 [mm]		Element Size 2 [mm]		Element Size 1.5 [mm]		Element Size 1 [mm]	
Elements	Nodes	Elements	Nodes	Elements	Nodes	Elements	Nodes	Elements	Nodes	Elements	Nodes	Elements	Nodes
8224	16715	19951	38003	71021	122069	300223	477204	1026886	1539112	2409520	3516023	7538258	10737023

Figure 3. Boundary conditions (a), material properties (b), mesh sensitivity analysis (c), details of the final mesh structure (d)

Afterward the completion of the pre-processing procedures, the FEA simulation was run and the results were recorded. The simulation outputs showed that the maximum equivalent stress (Von-Mises) on the bracket was 1139.8 MPa and the maximum deformation under engine torsion was 2.858 mm. These results revealed that the equivalent stress magnitude exceeds the material yield point (215 MPa) which was an indication of plastic deformation failure under defined boundary conditions (maximum torsional loading). The minimum safety factor was calculated as 0.188 which also signifies failure. The stress magnitude was also higher than the materials ultimate tensile stress point (505 MPa). This may be interpreted as the cause of fractures at the maximum stress zones on the bracket as the ultimate tensile stress point can be assumed to be the fracture failure limit in this static study. Moreover, the simulation printouts and physical damage pictures were compared and it was seen that the fracture and plastic deformation lines on the part were compatible with the simulation outputs (Figure 4). Here, another important point is cyclic loading conditions which may cause crack initiation and result in fatigue failure. Related literature highlights the empirical calculation of endurance limit in crack occurrence for steel materials which is 50% of the ultimate tensile strength (Bannantine, Julie A; Comer, Jess J; Handrock, 1990). In this study, this endurance value was calculated as 252.5 MPa which exceeds the material yield point (215 MPa). Therefore, the critical failure limit in this study was assigned as the material yield point and fatigue evaluation was not included.

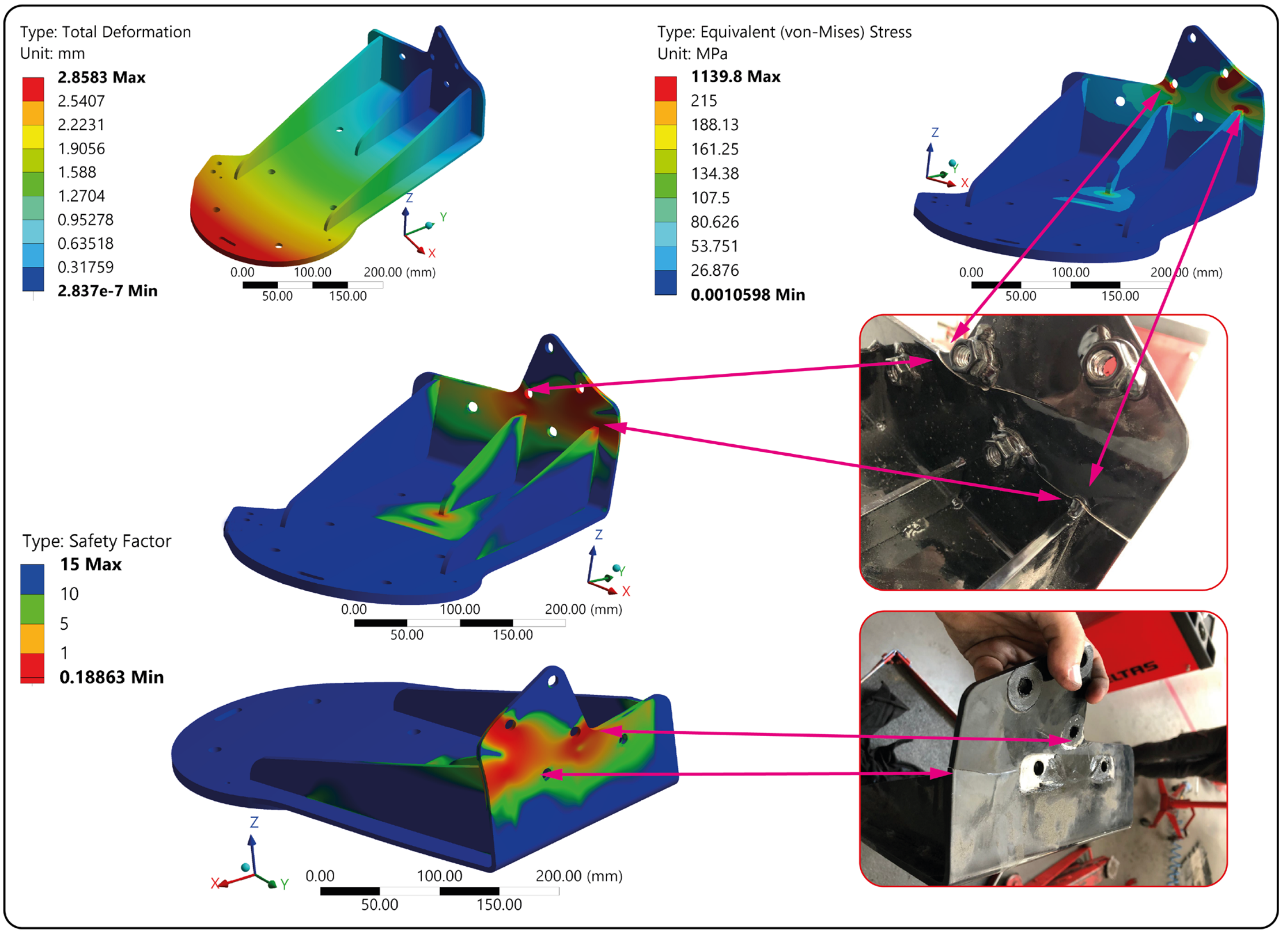


Figure 4. FEA outputs for the initial design and visual comparison

Re-Design and Corresponding FEA Validation

The results obtained from the initial design analysis on the damaged bracket led to the belief that there is insufficient part thickness at the bolt connection side (thickness is 5 mm) and notch radius where stress concentration is collected. Pre-defined design needs limited choice to using the same material (AISI 304). Therefore, re-design procedures focused on the improvement of the insufficient part thickness and avoiding the notch effect where the stress concentration is dense at the part geometry. In this regard, several re-designs were studied and then one of them was decided upon as the new design geometry by considering assembly and manufacturing ease features. After this decision-making process, the new design was validated by means of FEA. The FEA setup protocol for the re-design model had identical definitions with the FEA setup created for the initial design analysis.

Maximum equivalent stress of 169.91 MPa and maximum deformation of 0.283 mm on the re-design bracket was calculated in the re-design FEA solution respectively. These results revealed that the equivalent stress magnitude is lower that the material yield point (215 MPa). This was the indication from the elastic region of the bracket's material under defined boundary conditions. The deformation value was approximately 10 times lower than the initial design which may provide a more stable functionality of the bracket. The minimum safety factor was calculated as 1.265 which also signifies safe operation of the bracket. These results led to approval of the re-designed features.

Numerical method-based analysis techniques such as FEA provide approximate solutions, therefore, errors are inevitable. The errors may be methodical and numerical which are related to the mathematical model (e1), the mathematical discontinuity (e2) and the numerical solution processes (e3) (Narasaiah, 2008; Pancoast, 2009; Salmi, 2008). Additionally, user-based errors in setup and interpretation of the FEA results should also be kept under consideration, however, FEA is a very useful analysis tool in order to simulate real-life loading conditions.

Specific to this study, FEA results successfully exhibited the failure zone on the bracket, however, the exact fracture occurrence was not represented in the simulation, since the static and linear solving approach with linear elastic material model used in the FEA wouldn't reveal the part fracture failures on the bracket model considered in this study, however, this may be studied through a deeper analysis that would be considered in any future work focusing on dynamic conditions, and a nonlinear material model with fracture mechanics approach. Additionally, the likelihood of fatigue of part features should be kept under consideration.

As the result of the analyses conducted on the bracket, the major reasons for the failure were confirmed as being associated with the insufficient part thickness at the critical mating connection side and design faults on the bracket geometry which were eliminated through the re-design operation. Improved re-design dimensions and FE model and the FEA outputs for the re-design analysis are given in Figure 5. This would help to obtain more accurate part fracture results in a FEA for similar failure analysis studies.

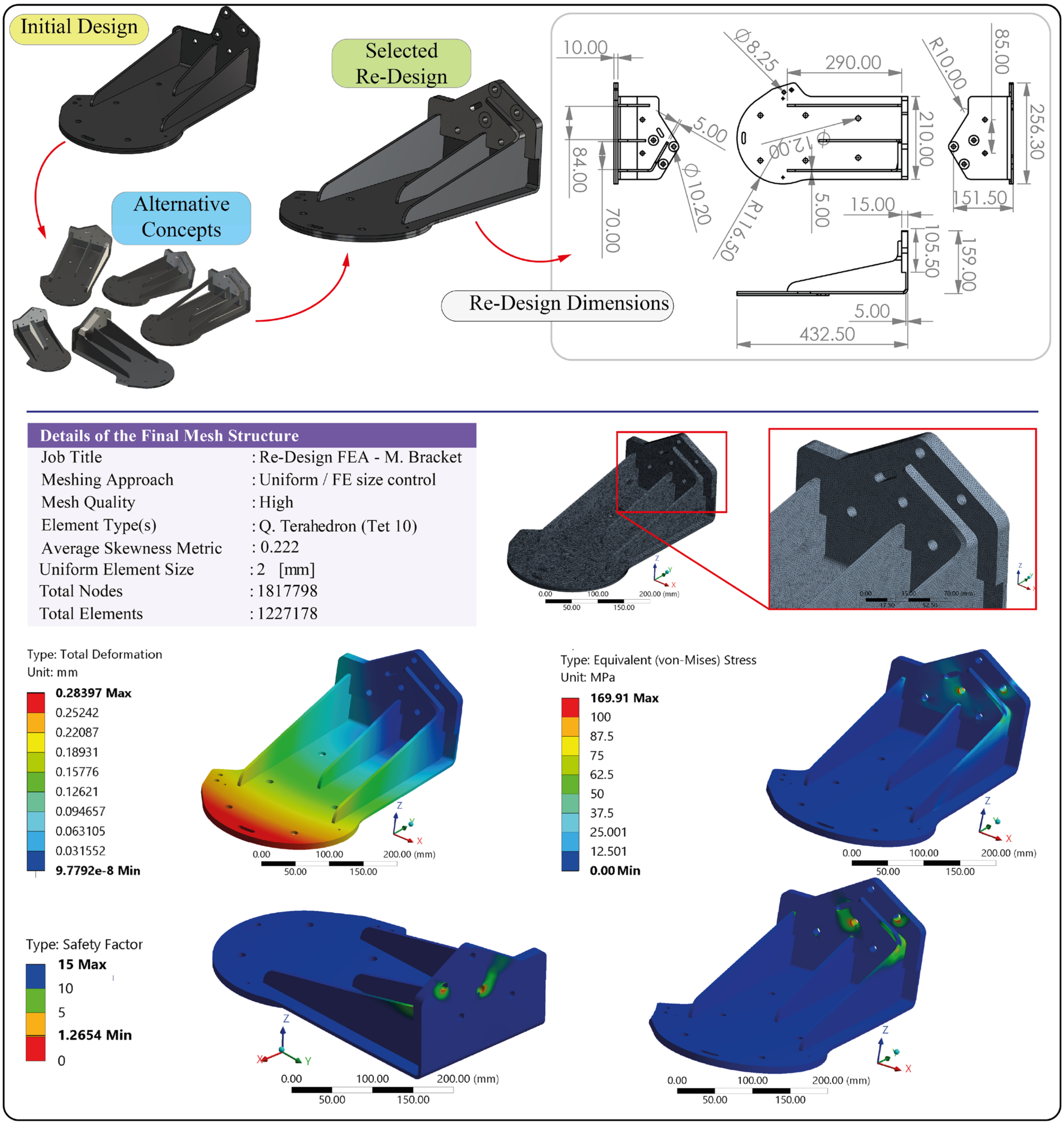


Figure 5. Re-Design and related FEA outputs

Conclusion

FEM-based design analysis of a failed engine mounting bracket specifically designed for a converted electric vehicle was discussed in detail in this paper. The results of the analyses conducted on the bracket, revealed that the major reasons for the failure are related to the insufficient part thickness and design faults on the existing bracket geometry. These reasons for failure were eliminated through re-design operation. Failures are frequently referred to as seriously affecting the production costs and durable design reputation of the components used in a converted electric vehicle. To avoid recurrence of the type of failure described in this study, it is recommended that the vehicle should be used within its approved operational conditions and loading limits which are an important failure prevention parameter. To design an engine mounting bracket with high service durability, mechanical characteristics such as yield strength, fatigue strength, and fracture toughness are important properties that should be considered during material selection and appropriate geometry at the design stage. Appropriate fillet radius and part thickness should be considered for the geometry design and the existing sharp features should be avoided. Component assembly should be undertaken with care and in accordance with the tolerances in order to provide durable loading transmissions.

This study provides a well-described numerical method-based simulation strategy for informing further research on complicated stress and deformation analyses of part deformation analysis through advanced computer aided engineering applications.

Acknowledgement

This research was supported financially by The Scientific Research Projects Coordination Unit of Akdeniz University (Antalya-Turkey). The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

References

ANSYS Product Doc. (2017). *Resolved Issues and Limitations: Release notes: Meshing, Release 18.2. ANSYS Inc.* (Issue August). ANSYS Inc., USA.

ANSYS Product Doc. (2019). ANSYS Meshing User's Guide: Skewness (Release 2019 R2). In *ANSYS Inc., USA*. https://ansyshelp.ansys.com/account/secured?returnurl=/Views/Secured/corp/v191/wb2_help/wb2_help.html

Areias, P., Msekh, M. A., & Rabczuk, T. (2016). Damage and fracture algorithm using the screened Poisson equation and local remeshing. *Engineering Fracture Mechanics*, *158*, 116–143. https://doi.org/10.1016/j.engfracmech.2015.10.042

Areias, P., Reinoso, J., Camanho, P. P., César de Sá, J., & Rabczuk, T. (2018). Effective 2D and 3D crack propagation with local mesh refinement and the screened Poisson equation. *Engineering Fracture Mechanics*, *189*, 339–360. https://doi.org/10.1016/j.engfracmech.2017.11.017

Bannantine, Julie A; Comer, Jess J; Handrock, J. L. (1990). *Fundamentals of metal fatigue analysis*. Prentice Hall.

Brys, G., Hubert, M., & Struyf, A. (2004). A robust measure of skewness. *Journal of Computational and Graphical Statistics*, *13*(4), 996–1017. https://doi.org/10.1198/106186004X12632

Celik H.K., Cinar R., Kunt G., Rennie A.E.W., Ucar M., & Akinci I. (2018). Finite Element Analysis of a PTO Shaft Used in an Agricultural Tractor. *Ergonomics International Journal*, *2*(3), 1–6. https://doi.org/10.23880/EOIJ-16000147

Ganjiani, M., & Homayounfard, M. (2021). Development of a ductile failure model sensitive to stress triaxiality and Lode angle. *International Journal of Solids and Structures*, *225*, 111066. https://doi.org/10.1016/j.ijsolstr.2021.111066

Ghorpade, U. S., Chavan, D. S., Patil, V., & Gaikwad, M. (2013). FINITE ELEMENT ANALYSIS AND NATURAL FREQUENCY OPTIMIZATION OF ENGINE BRACKET. In *International Journal of Mechanical and Industrial Engineering* (Vol. 3, Issue 1).

Hughes, T. J. R., Cottrell, J. A., & Bazilevs, Y. (2005). Isogeometric analysis: CAD, finite elements, NURBS, exact geometry and mesh refinement. *Computer Methods in Applied Mechanics and Engineering*, *194*(39–41), 4135–4195. https://doi.org/10.1016/j.cma.2004.10.008

Keoun, B. C. (1995). Designing an electric vehicle conversion. *Southcon Conference Record*, 303–308. https://doi.org/10.1109/southc.1995.516121

Kerem, A. (2014). Elektrikli Araç Teknolojisinin Gelişimi ve Gelecek Beklentileri. *Mehmet Akif Ersoy Üniversitesi Fen Bilimleri Enstitüsü Dergisi*, *5*(1), 1–13. http://febed.mehmetakif.edu.tr

MatWeb, M. D. (2020). *MatWeb- Material Data: 304 Stainless Steel*. Source. http://www.matweb.com/search/DataSheet.aspx?MatGUID=abc4415b0f8b490387e3c922237098da

Narasaiah, G. L. (2008). *Finite Element Analysis*. B.S. Publications. https://www.biblio.com/9788178001401

Nguyen, V. P., Anitescu, C., Bordas, S. P. A., & Rabczuk, T. (2015). Isogeometric analysis: An overview and computer implementation aspects. *Mathematics and Computers in Simulation*, *117*, 89–116. https://doi.org/10.1016/j.matcom.2015.05.008

Pancoast, D. (2009). *Solidworks Simulation-2010 Training Manual* (PMT1040-EN ed.). Dassault System - Solidworks

Corporation.

Pardhi, D. G., & Khamankar, S. D. (2014). Stress analysis of spline shaft using finite element method and its experimental verification by photo elasticity. *Int. J. Mech. Eng. & Rob. Res*, *3*(4), 451–458. www.ijmerr.com

Salmi, S. (2008). *Multidisciplinary Design Optimization in an Integrated CAD / FEM Environment*. 81. https://jyx.jyu.fi/handle/123456789/18752

Subbiah, S., Singh, O. P., Mohan, S. K., & Jeyaraj, A. P. (2011). Effect of muffler mounting bracket designs on durability. *Engineering Failure Analysis*, *18*(3), 1094–1107. https://doi.org/10.1016/j.engfailanal.2011.02.009

Wada, M. (2009). Research and development of electric vehicles for clean transportation. *Journal of Environmental Sciences*, *21*(6), 745–749. https://doi.org/10.1016/S1001-0742(08)62335-9

Zeraoulia, M., Benbouzid, M. E. H., & Diallo, D. (2006). Electric motor drive selection issues for HEV propulsion systems: A comparative study. In *IEEE Transactions on Vehicular Technology* (Vol. 55, Issue 6, pp. 1756–1764). https://doi.org/10.1109/TVT.2006.878719