1st International Mediterranean Science and Engineering Congress (IMSEC 2016)
Çukurova University, Congress Center, October 26-28, 2016, Adana / TURKEY
Pages: 2054-2063, Paper ID:598

Development of analysis methodology for engine brackets

Ezgi Eryılmaz[1], Özgün Küçük[2], Yeliz Pekbey[3]

[1]Ege University/Department of Mechanical Engineering, İzmir,TURKEY; Emaileryilmaz.ezgi@outlook.com,
[2]Hexagon Studio, İstanbul, TURKEY; Email : ozgun.kucuk@hexagonstudio.com.tr
[3]Ege University/Department of Mechanical Engineering, İzmir,TURKEY; Email: yeliz.pekbey@ege.edu.tr

Abstract

In this study, designing accelerated life test system of 18m Euro VI commercial vehicle which was produced in Hexagon Studio, has been done model of these test system in virtual platform, doing analysis of these test system (simulation) and doing comparing studies both test results and analysis results. This study is aimed; create a faster methodology to define design of engine brackets, one of the parts of vehicle which is exposed to dynamic effects, in terms of strength. In this context; improving test and virtual engineering abilities and coordination of this subject with each other has been provided. Vehicle and road model has been verified with using road data's which was taken in earlier tests. In the second part; engine brackets fatigue life has been defined with using this data's, verified models and finite element model. In the current product development process, it is impossible to do strength tests of vehicle which's designs is not completes at all. With the end of this study, verifying design in terms of strength will be done on computer. This situation will be provided big advances in terms of cost and time.

Keywords: Engine Brackets, Fatigue Life, Dynamic Analysis, Finite Element Analysis

1. INTRODUCTION

Durability and reliability of a product are important for customer. Reliability is defined as the measure of unanticipated interruptions or unexpected failures during customer use. It is crucial to maximize the fortunes for viewing unexpected failures during a reliability test [1]. There are significant warranty costs in automotive industries because of failure of their products. Failure modes investigation is important duee to reducing the design development expenses. The fatigue and accelerated life test investigations of the engine brackets investigated in this paper are examined during the durability tests under actual environment. The design against fatigue of automotive engine components very often covers a preliminary monotonic and cyclic stress-strain analysis, followed by a fatigue crack initiation or propagation criteria to specify the durability of the component.

The parts made of various materials we use in our daily lives or technological applications used in the system are meticulously made design and despite the well known characteristics of the appropriate choice of materials used in material, still loses its function during submission service or services are faced with situations not providing the expected performance. Fatigue is one of the most important mechanisms of damage, plays an important role in the election to determine the fatigue characteristics of the material [1].

Stress suffered by under dynamic loads running machine elements even if at a value below the yield strength of the material after the specific number of cycles can be occurred on the surface cracking and it harms part as to leave part of the surface of the material in progress. This event is called fatigue. The results of fatigue, could be disastrous if it would be secretly and occur without warning at all. A lot of work is done in understanding the mechanisms of fatigue, these studies were obtained through a lot of information of fatigue. However, with these studies were carried through in many studies on how to slow or completely prevent fatigue [2,4]. The reason for the use of fatigue word is; emerging breaks like this after a long the cycle of stress and strain.

Fatigue damage is always composed. To understand the fatigue damage we answer the questions such as, fatigue how it occurs, how we predict fatigue in advance, how to design against fatigue is sought [4]. Safe design against fatigue damage, to define of strenght of the product under repetitive loads, analysis and test results should be used at the same time. There are many methods available for this design approach based on fatigue, these approaches can be complicated and expensive. In addition, the development in computer technology, there has been significant steps in the modeling of fatigue problems [5].

In this study; design of the accelerated fatigue test system, comparison of test results with analysis and analysis of the test system computer will be made for 18m Euro VI commercial vehicle engine brackets which designed by Hexagon Studio (Hexagon Engineering and Design Inc.) operating in the automotive sector. Dynamic simulation and finite element analysis methods, with less ability to experiment with digital prototyping will attempt to reach the safe design against fatigue. Fatigue strength that will be examined in computer simulations and we will try to achieve the optimum design.

2. METHODS

In this study, the correlation between measurement data collected by the designated road conditions and prepared the way computer tools and models of prototype testing tool to determine the service life of the engine bracket will be provided. The first to do, the finite elementh model will be developed for body frame structure which will test, then fatigue tests will be performed by creating testing device which simulates the data that were taken from the road test which the vehicle was in flat-plain road.

Firstly in this study; to define behaviors of engine brackets with the road forces; correlation of prototype of test systems measure data and vehicle and road models which had designed on computer. Latterly; Fatigue characteristic of engine brackets had been defined with generating test procedures. Test results and analysis reports had been compared. Bracket's fatigue life had been defined in test system with defining force and cycle in two axes, then, this test system had been simulated with finite element methods.

2.1 Modelling of Euro6 Engine Room with Finite Element Method

In this study, the finite element model of the carcass structure body which will be tested were made with using commercial software program Hypermesh. Model benefiting from the company's customer experience is taken into account when establishing the criteria specified below. Rigid members are used in the welded joints.

Element size: 5mm

Element type: QUAD4

Required Element Criteria;

Warpage > 10 <20 -- <5% & Warpage > 20 < 0%

Aspect > 5 <10 -- <5% & Aspect > 10 < 0%

Quads Min angle < 45 >35 -- <5% Min Angle <35 -- < 0%

Quads Max angle >135 <150 --<5 Max angle >150 < 0%

Trias Min angle < 20 >15 -- <5% Min Angle <15 -- < 0%

Trias Max angle >120 <135 --<5 Max angle >135 < 0%

Jacobian < 0.7 5% -- < 0.5 < 0 %

Established model (Figure 1) jocobi and aspect factors more than all these values were determined as important criteria while maintaining control. Aspects Ratio is defined as the ratio of; the shortest length to the longest length for the tetrahedral element. This rate, so there is a good mesh nearer 1. However, due to the complexity of the geometry it is difficult to obtain an element like this type. Therefore, by keeping the value under 5 in structural analysis was obtained in a quality mesh structure [4,9].

Because of receiving polynomial property to the edges of each of the elements; elements used intertwining when cause deformation or creating the mesh. This is done with Jacobean control and may possibility. The control tools that are Jacobean point. This factor must be at least a specified range [4, 9].

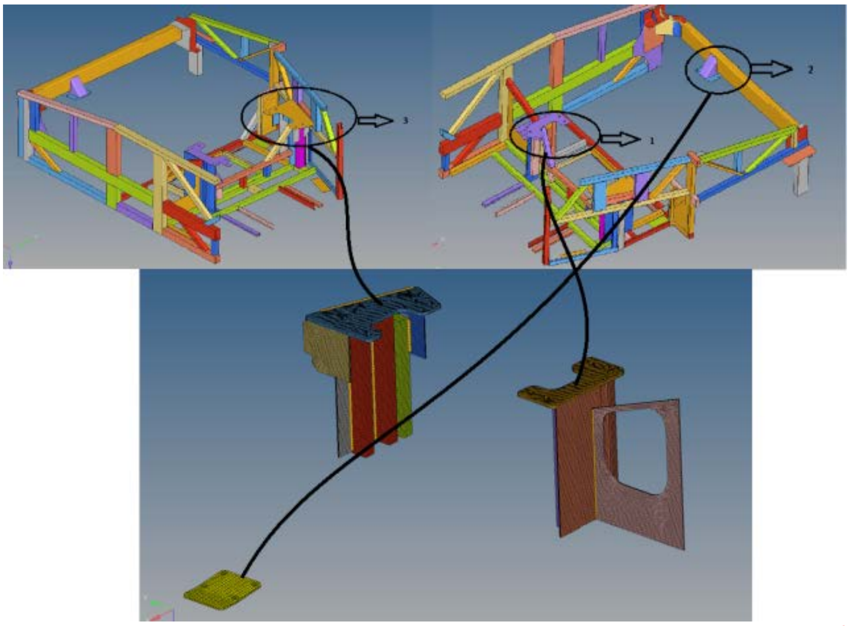


Figure 1: Finite Element Model of Body Carcass (Mount1 (1), Mount2 (2), Mount3 (3))

Figure 1 shows a finite element model of the body of the carcass structure. Mount 1, Mount 2 and Mount 3 shown here, are the brackets that connect motors and equipment. The inertia forces of the engine room and the forces from the road are covered by this bracket.

2.2 Preparation of Test Procedure for Engine Brackets and Making Tests

In this study, fatigue tests were conducted in the biaxial test system with fatigue testing procedures that prepared with determining precise point of the model which was done structural analysis taking into.

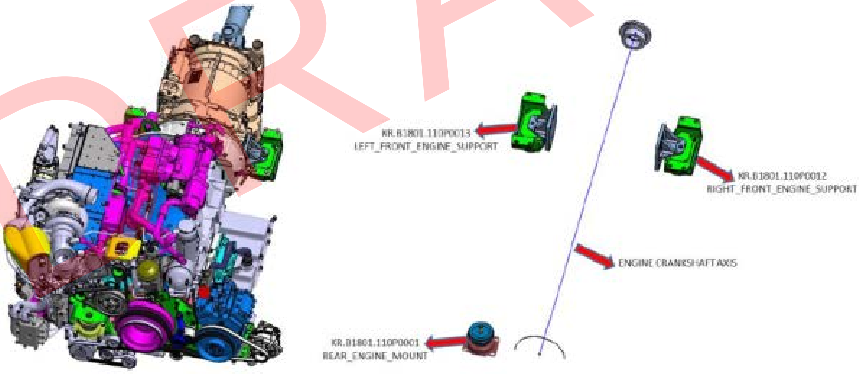


Figure 2: Layout of the engine and they are connected with the bracket in vehicle

Figure 3: Fatigue Testing Apparatus

The specified installation of the piston at a fixed frequency (0.5 Hz) was performed with the installation with entering the test procedures thars is prepared based on the data received in the road test system in fatigue test system with the installation. Test system shown in Figure 3.

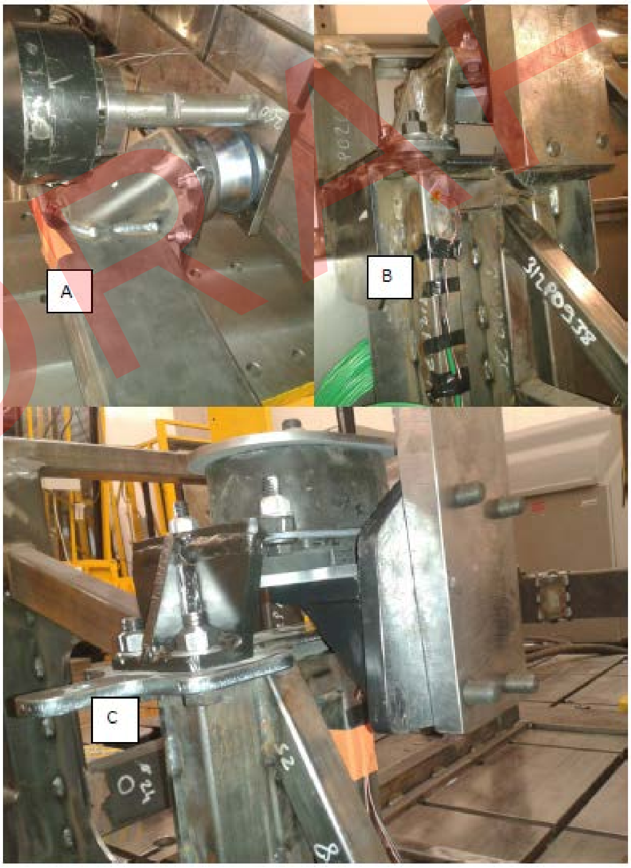


Figure 4: Areas of the Strainguage Placement

Strainguages are determined by taking into consideration CEA analysis and placed in the specified region in Figure 5.

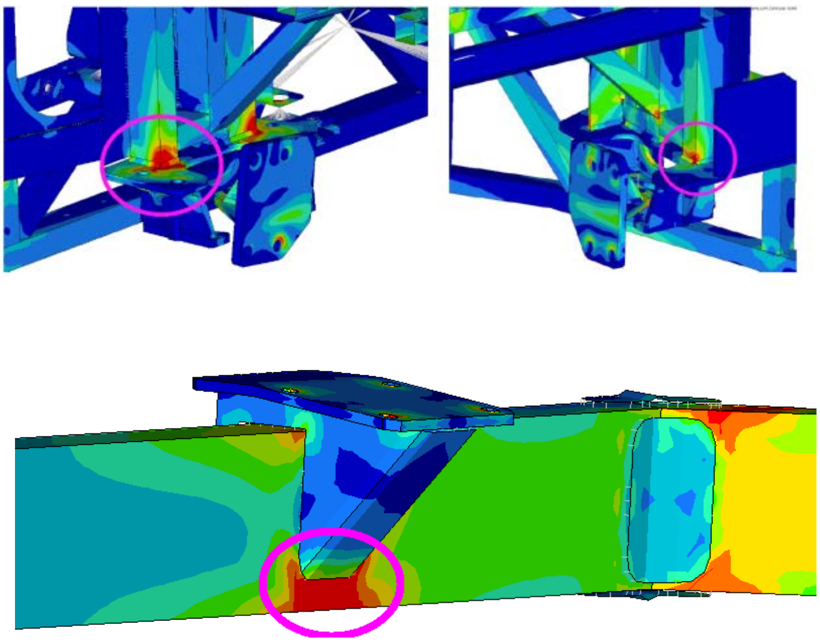


Figure 5: Areas of the Strainguage Placement

To make a point of stress analysis in case of plane stress (the presence of the principal stresses and direction, etc.) the strain at that point at least three directions must be measured. During the test 135 ° rosette type strainguages are used. Principal stresses have been identified with measurements in three directions.

$$\varepsilon_{\theta} = \varepsilon_x \cos^2\theta + \varepsilon_y \sin^2\theta + 2\,\gamma_{xy}.\sin\theta.\cos\theta \qquad (1)$$

When using Equation (1), angle θ is taken respectively 45 °, 90 ° and 135 ° depending on the use of rosette strainguages [3]. To determine the service life of the engine bracket, the correlation between measurement data collected by the designated road conditions and the way and vehicle model prepared computer tools of prototype testing vehicle will be provided. Static torsional strength was determined to be 13000 Nm torque. In the loading 3g bump condition, a large pot holes and no state, which is the most challenging form of road was simulated for each bracket with using values shown in Table 1. ± Z directions as shown in Table 1, shown in Figure 3 [6, 7].

Table 1 Fatigue Test Installation Requirements

(+)3g	± Z Direction
Uni-directional Tcs, 0,5 Hz, B50- 5000 cycle	**Mount 1** [6716 N (compression)] - [20150 N (compression)] **Mount 2** [4553 N (compression)] - [13660 N (compression)] **Mount 3** [5583 N (compression)] - [16750 N (compression)]

The safety factor in fatigue life testing conducted under the value of the yield strength of the body frame made of steel is determined to be 1.66. In results of test values which were reading from strainguages were examined with graphs in nCODE module. In test results deformation was observed in welds or other structural elements on the fixture.

3. FINITE ELEMENT ANALIYSIS

With creating finite element model of engine mounts mounting brackets of 18m Euro VI and applying 1718 kg engine + gearbox weight with in 3g bump loading case reverse of clockwise 14048 Nm maximum torque is made of static analysis. Body connections to connect the engine mount bracket assuming rigid boundary conditions were established. The mechanical properties of materials used in motor mounting bracket are given Table 2.

Table 2 The mechanical properties of materials used in motor mounting bracket

	Weldox	GEKA SG 2 (Weld Joints)
Modulus of Elasticity (GPa)	210	210
Yield Stress (MPa)	700	420
Poisson's Ratio	0.3	0.3
Density (ton/mm^3)	7.9x10^{-9}	7.9x10^{-9}

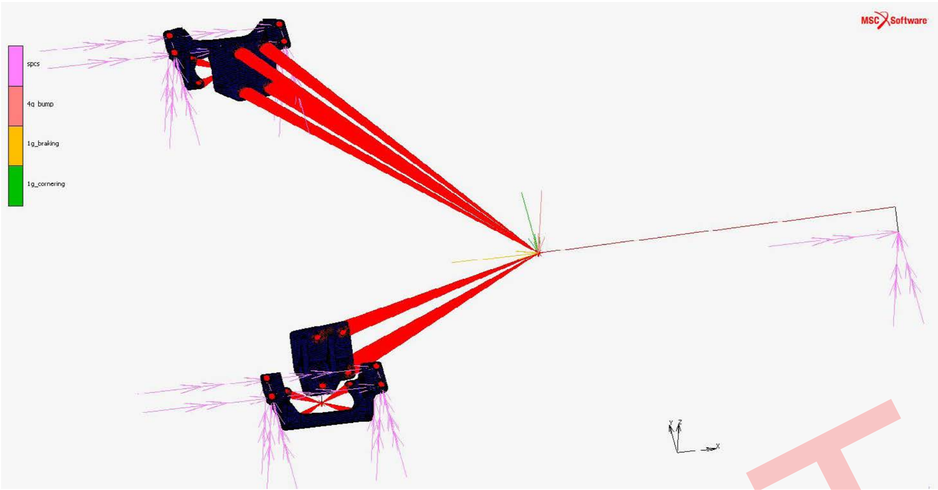


Figure 6: Finite Element Model of Engine Mounting Bracket Connection and Boundary Conditions

Engine mounts used in the system; the damping characteristic of the rubber were modeled as 6 degree of freedom nonlinear spring for being simulated more accurately. Figure 6 shows finite element model of engine mounting bracket connection and boundary conditions prepared by 3g bumper loading status. The manufacturer did not return by the return information provided stiffness in three axes at 10000 N / is taken as rad. Figures 7 and 8, respectively, shows an isometric view and stiffness curve of the wedge in front of the engine. Figure 9 and Figure 10 shows stiffness curve and isometric view of the wedge is located next to the gearbox.

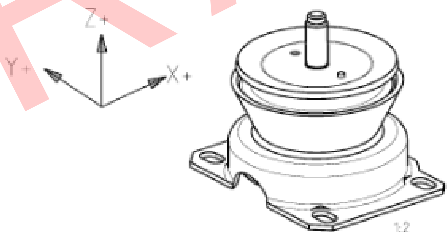


Figure 7: The Isometric Wiev of The Wedge in front of The Engine

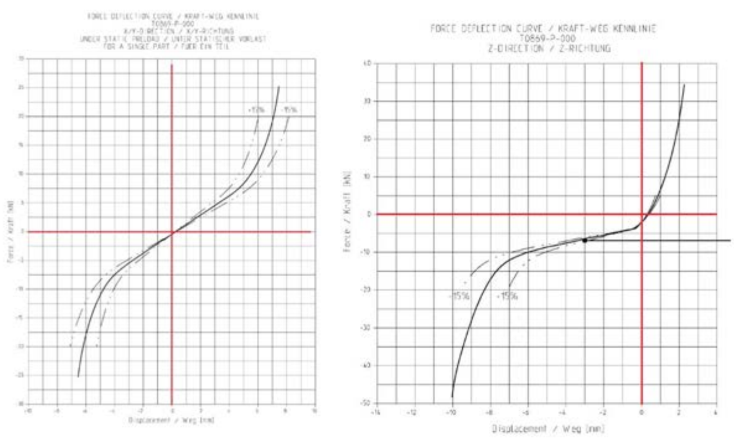


Figure 8: Stiffness curve of the wedge in front of the engine

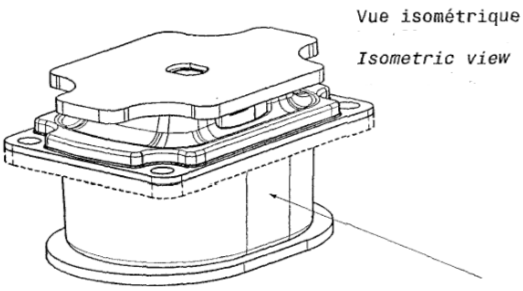


Figure 9: Isometric view of the wedge is located next to the gearbox

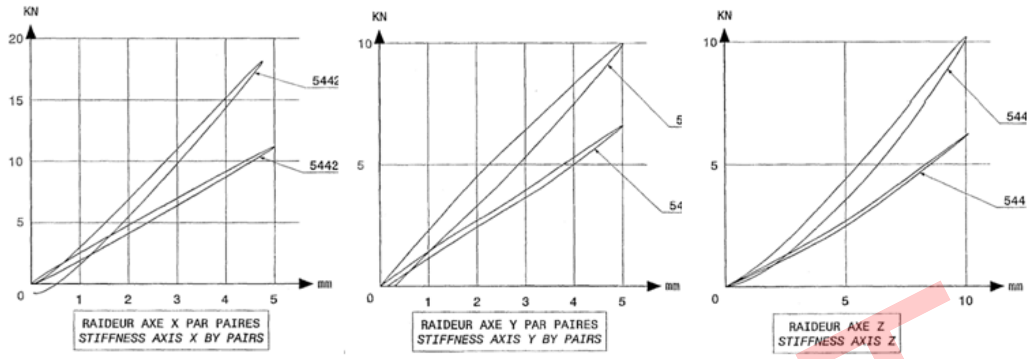
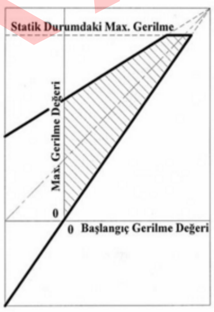


Figure 10: Stiffness curve of the wedge is located next to the gearbox

Model's fatigue analysis was made in nCODE with Goodman method preferred for fatigue calculation of vehicle sub-components [8, 9].

Goodman approach used to calculate when applying any σ_{ort} stress to the material how much of strain amplitude can be implemented. If the variable stress amplitude zero (static material is forced entirely) stress which breaks the material would be tensile strength of the material. Yield limit is accepted as damage and because of throwing out account of strength area safer calculation can be made (Figure 11) [8, 9].



$$\frac{\sigma_{Alternating}}{S_{Endurance\_Limit}} + \frac{\sigma_{Mean}}{S_{Ultimate\_Strength}} = 1$$

Figure 11: Goodman Diagram and Formulation

All models are defined by the material characteristics of steel S420 (σ_{akma}= 420MPa/ σ_{kopma}=600MPa), S-N curve is given in Figure 12.

First crack as a result of the start of the analysis carried out in 3g bumper conditions in the juntion of the profile and Mount 1 at the end of the cycle 914 000 [10]. Figure 13 shows analysis results of the first cracks start area and node. Start cycle fatigue cracks occurring in other engine mounts were obtained much higher (Figure 14).

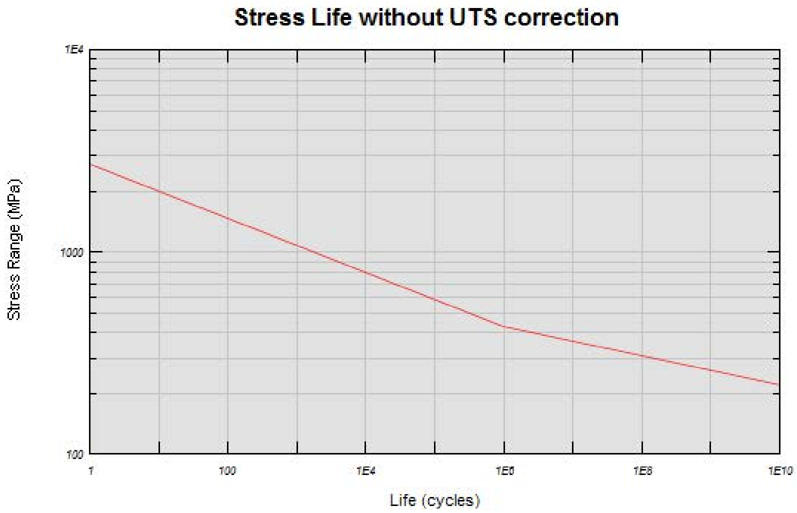


Figure 12: S420 Çeliği S-N Curve

Goodman approach was used in the analysis carried out in 3g bumper conditions.

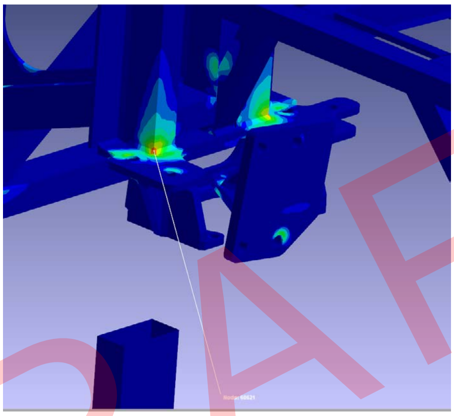


Figure 13: Analysis results of the first Cracks Start Area and Node.

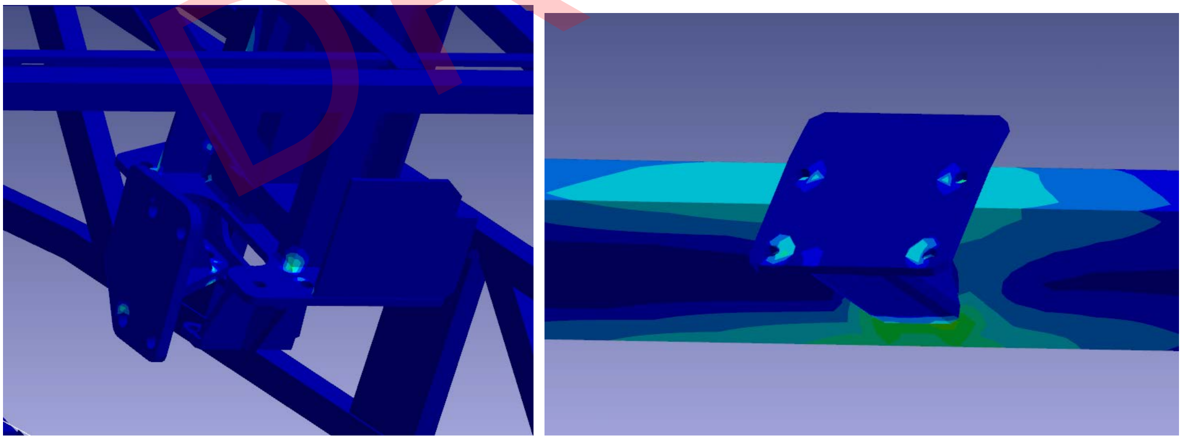


Figure 14: Analysis Results of Other Engine Mounting

4. RESULTS AND DISCUSSION

In this study, designing accelerated life test system of 18m Euro VI commercial vehicle which was produced in Hexagon Studio, has been done model of these test system in virtual platform, doing analysis of these test system (simulation) and doing comparing studies both test results and analysis results. With the end of this project, verifying design in terms of strength will be done on computer. This situation will be provided big advances in terms of cost and time.

First cycle strain values received from the analysis and deformation (strain) values which were calculated with (1) equetion and obtained from strainguages for average values are compared and verified with approximately 10% deflection. Analysis found that infinite life which is desired in bracket the vehicle sub-components has been obtained in the motor mounting as a result.

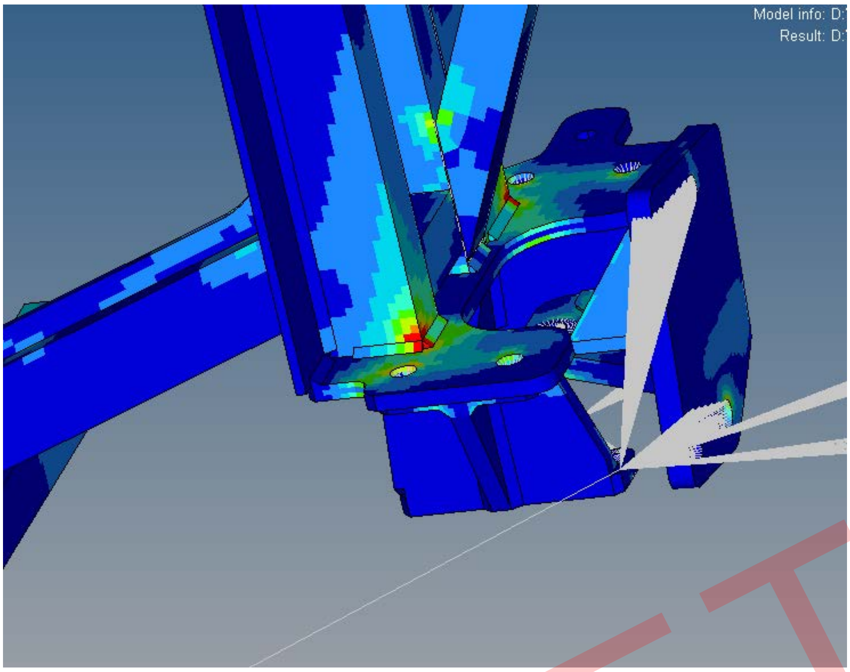


Figure 15: Area of First Start Fatigue Crack In 3g Bumper Condition

5. CONCLUSION

The purpose was in here to become as works that make shorter test and verification steps in Hexagon Studio. Along with work; the design process economically, to improve the efficiency of the mechanical packaging and ergonomics process for car layout, design and development process have been shorten and along with the shortening of the time between placing on the market of the product design is also reduced. These gains are providing a competitive advantage in bidding for public transport.

5. ACKNOWLEDGEMENTS

The authors gratefully acknowledge The Republic of Turkey Ministry of Science, Industry and Technology as a SANTEZ support program for the financial support 0826.STZ.2014 (Motor Braketlerinin Hızlandırılmış Yorulma Test Düzeneğinin Tasarlanması ve Analiz Metodolojisinin Geliştirilmesi) for carrying out this work. Therefore we offer our heartfelt thanks to The Republic of Turkey Ministry of Science, Industry and Technology and to Hexagon Studio (Heksagon Engineering and Design Inc.).

REFERENCES

[1] Subbiah S., Singh O.P., Mohan S.K., Jeyaraj A.P., (2011), “Effect of muffler mounting bracket designs on durability”, Engineering Failure Analysis, Vol.18(3), pp. 1094–1107

[2] Özsoy S., Çelik M., Kadıoğlu F. S.,(2008), “An accelerated Life Test Approach for Aerospace Structural Compenents”, Engineering Failure Analysis, Vol.15, Issue.7

[3] Kuntay A., İpek G., Güngör S., “Hızlandırılmış Ömür Test Prosedürü Geliştirilmesi”, http://ex-en.com.tr/imgup/Hizlandirilmis_Omur_Testi_Otekon.PDF.

[4] Özdemir N. Ö.,(2009), “Sonlu Elemanlar Yöntemi ile Yorulma Mekaniği ve Uygulamaları”, Karadeniz Teknik Üniversitesi, Mühendislik Fakültesi, Lisans Tezi, Trabzon.

[5] Şener A. Ş.,(2010), “Araç Parçaları Yorulma Ömürlerinin Sonlu Elemanlar Yöntemi ile Belirlenmesi”, Taşıt Teknolojileri Elektronik Dergisi (TATED), Cilt.2, No.3

[6] El-Ratal W., Bennebach M.,(2014), “Fatigue Life Modelling and Accelerated Tests For Components

Under Variable Amplitude Loads.", Journal of ASTM International, Vol.1, No.8

[7] Bishop N., Lack L.W., Li T., Kerr S. C., "Analytical Fatigue Life Assessment Vibration Induced Fatugue Damage", https://web.mscsoftware.com/support/library/conf/wuc95/p01895.pdf

[8] Lee Y., Pan J., Hathaway R., Barkey M.,(2005), "Fatigue Testing and Analysis" Elsevier Butterworth-Heinemann.

[9] Bishop N. W. M., Sherratt F.,(2000), "Finite Element Based Fatigue Calculation", NAFEMS, 22-44.

[10] Eryılmaz E., "Hızlandırılmış Yorulma Analizi", Yüksek Lisans Tezi, Ege Üniversitesi Fen Bilimleri Enstitüsü Makina mühendisliği Anabilim dalı, 2016.