ISSN No: -2456-2165

Fatigue Analysis of Excavator Arm Reduced in Weight by Structural Optimization Methods

Salih SOYLU[1]
[1]Students, Mechanical Engineering Department
Bursa Uludag University, 16059
Bursa, Turkey

Yahya IŞIK[2]
[2]Mechanical Engineering Department
Bursa Uludag University, 16059
Bursa, Turkey

Abstract:- Today, the excavator arm, which is one of the important parts of excavators that are widely used in many areas, especially in the construction sector, provides the movement of the excavator's bucket. Excavator arms are produced in different sizes and shapes according to the usage areas of excavators. In this study, a solid model of a sample excavator arm was created using the SolidWorks design program and the values of the forces that may occur depending on the load lifted on the excavator arm were calculated if the excavator lifted a load of 1000 kg. Matlab and Gauss Elimination Method were used to find the force values. In line with the force values obtained, linear static analysis and fatigue analysis were applied to the excavator arm, whose weight was reduced by topology optimization and shape optimization. According to the results obtained from the fatigue analysis, the areas where the fatigue limit was exceeded on the excavator arm were determined and it was aimed to create a more durable and safer design for the excavator arm in terms of damage and life. While creating the design, attention was paid to the weight of the excavator arm and the maximum stress and maximum displacement values on the excavator arm. HyperMesh program was used for fatigue analysis and linear static analysis.

Keywords:- *Excavator Arm, Fatigue, Hyper Mesh, Linear Static.*

I. INTRODUCTION

Today, excavators are construction machines that are widely used in many fields, especially in the construction sector. The fact that excavators have different models according to the variety of work they do and their working areas enables them to be used in many areas other than the construction sector. Excavators can be used in any area where there are works such as digging, loading, crushing, transporting. Excavators are generally formed by combining the main parts such as boom, arm, bucket, hydraulic cylinders, operator's cabin, walking and rotating hydraulic motors, etc.

Used in industry, many machine parts and structural elements are exposed to variable stresses during their use. Even if the intensity of these stresses is smaller than the yield strength, cracking and following this rupture events occur in the materials after a certain repetition over time. In these events, it is defined as material fatigue [1]. The analysis processes performed to determine the life of structures exposed to variable stresses are called fatigue analysis. The condition of fatigue of materials, although it has been known since the early nineteenth century, the first serious research on this topic began in the middle of the twentieth century [2]. In the 1970's, fatigue analysis has been started to be applied as an engineering solution in many industrial applications [3, 4, 5]. Although many parts work well at the beginning, after a certain period of time it suffers fatigue damage depending on the loads it is exposed to. As a result, it also loses its functionality [6]. The fatigue event is very dangerous from a safety point of view because it occurs suddenly without showing any symptoms in the materials and causes the materials to break or rupture without creating a significant plastic shape change in the stress values under the yield strength [7, 8, 9]. For this reason, fatigue analysis should be performed on parts exposed to variable stresses in structures such as aircraft, bridges, vehicles and machine and conclusions should be reached about how much these parts can be used during their lifetime. It should be known that a material that starts to get tired, the fatigue state will never end. Even if the system does not work, fatigue will continue where it left off when it starts working again [2]. In addition, the fatigue event does not occur only in materials exposed to variable stresses under dynamic loads. In some cases, even if the acting loads are static, the stresses formed in the cross-sections of the materials may be variable [6].

There are many parameters that cause materials to get tired. These parameters are generally related to part geometry, material microstructure, load distribution, production process and environmental effects. In particular, holes in the part geometry, wedge slots, sharp corners and screw threads are weak areas in terms of fatigue. These areas geometrically always increase the stress value [10, 11]. For this reason, it is necessary to avoid structural irregularities that will increase stress value and sudden cross-sectional changes that cause the formation of sharp corners.

In this study, the excavator arm, which is one of the important parts of the excavator, was examined. In the event that the excavator lifts a load of 1000 kg, the values of the forces that will occur depending on the load lifted on the excavator arm were calculated. Matlab and Gauss Elimination Method were used to find the force values. Fatigue analysis was applied to the excavator arm, whose weight was reduced by using structural optimization methods, in the HyperMesh program. According to the results obtained from the fatigue analysis, it is aimed to create a more durable and safer design for the excavator arm in terms of damage and life.

ISSN No: -2456-2165

II. MATERIALS AND METHODS

Today, excavators are one of the first construction machines that come to mind when it comes to the construction industry. It is usually possible to see excavators in jobs such as digging, loading, transporting and crushing. The working hardware consists of three main parts. These are; boom, scoop and arm. Hydraulic cylinders that give movement to these parts are also included in this group. In this study, the excavator arm given in Figure 1 was examined. In line with the determined targets, in the first stage of the study, the designs of the excavator parts that we will use in the study were made using the SolidWorks design program. If the excavator lifts a load of 1000 kg after the design processes, the forces that may occur depending on the load lifted on the excavator arm are calculated. In the second stage, linear static analysis was applied to the excavator arm, whose weight was reduced by using structural optimization methods. By linear static analysis, the stress and displacement values on the excavator arm were determined. In the third stage, fatigue analysis was applied to the excavator arm, whose weight was reduced by using structural optimization methods, and the areas where the fatigue limit was exceeded were determined. According to the results obtained in the fatigue analysis, a more durable and safer design has been created for the excavator arm in terms of damage and life.

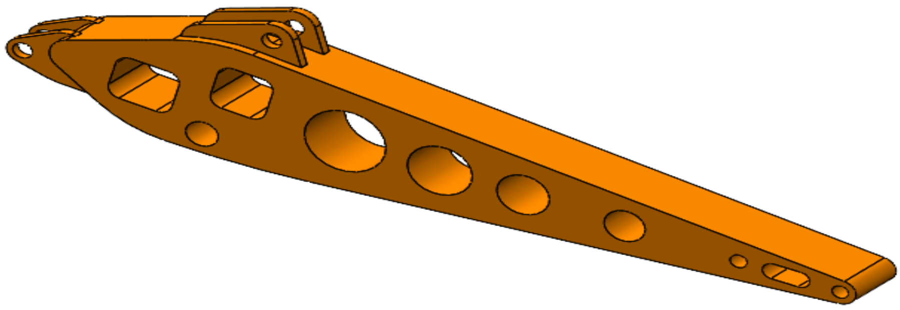

Fig. 1: Excavator arm design with reduced weight by structural optimization methods

A. Forces to Occur in the Excavator Arm

In the event that the excavator lifts a load of a specified value, the forces that may occur on the bucket and arm parts depending on the lifted load are determined. Afterwards, free-body diagrams of the bucket and arm parts were created. In the selection of the load, attention was paid to the safety of the excavator arm in terms of yield strength. As a result, it was decided that a load of 1000 kg would be suitable for work. Static equilibrium equations were used to find forces.

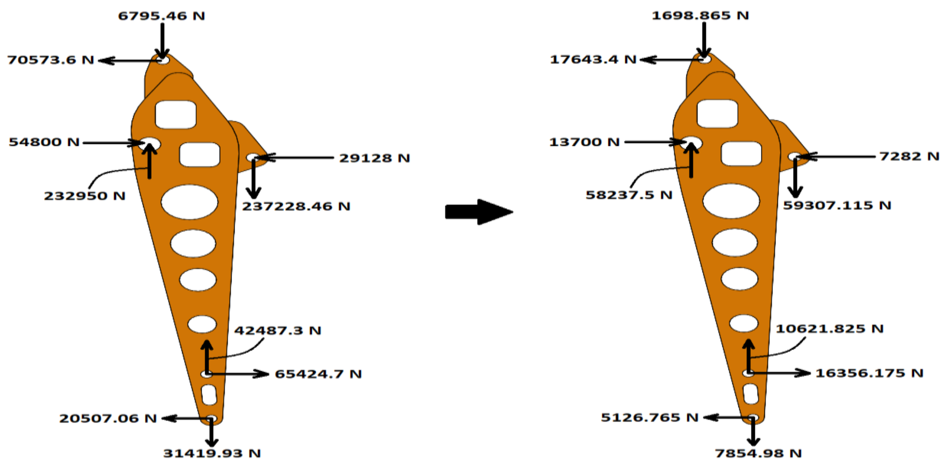

Fig. 2: Values of the forces generated in the excavator arm

In order to find the values of the forces on the excavator arm, 5 equations were obtained since there are 5 unknown forces with the help of balance equations. These equations were converted into matrix form and solved with the help of the Gauss Elimination Method code written in the Matlab program. The forces obtained are given on the excavator arm in Figure 2. Since the excavator arm is very large, the dimensions of the excavator arm have been reduced by 1/4 and the forces acting on the excavator arm have been reduced by this proportion in order to shorten the analysis and optimization solution times.

B. Linear Static Analysis of Excavator Arm

Linear static analysis was applied to the excavator arm in the HyperMesh program. The operations performed for linear static analysis are given in Figure 3. The element size for the mesh was made 4. S355JR was preferred for the material of the excavator arm. The selection of anchor points (SPC) is shown in Figure 3. All forces generated in the excavator arm were applied as shown in Figure 2. When the linear static analysis was solved from Optistruct, the result given in Figure 4 was obtained. According to the analysis results, the maximum stress on the excavator arm was determined to be 339,1 MPa and the maximum displacement was 0,3279 mm.

ISSN No: -2456-2165

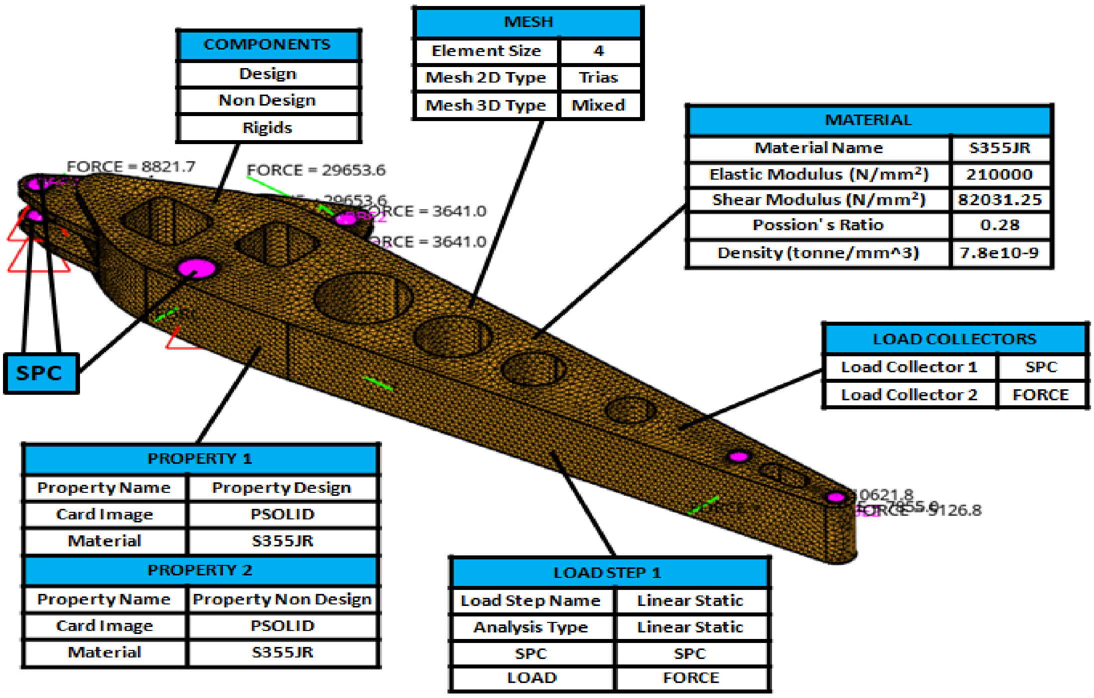

Fig. 3: Operations applied to the excavator arm for linear static analysis

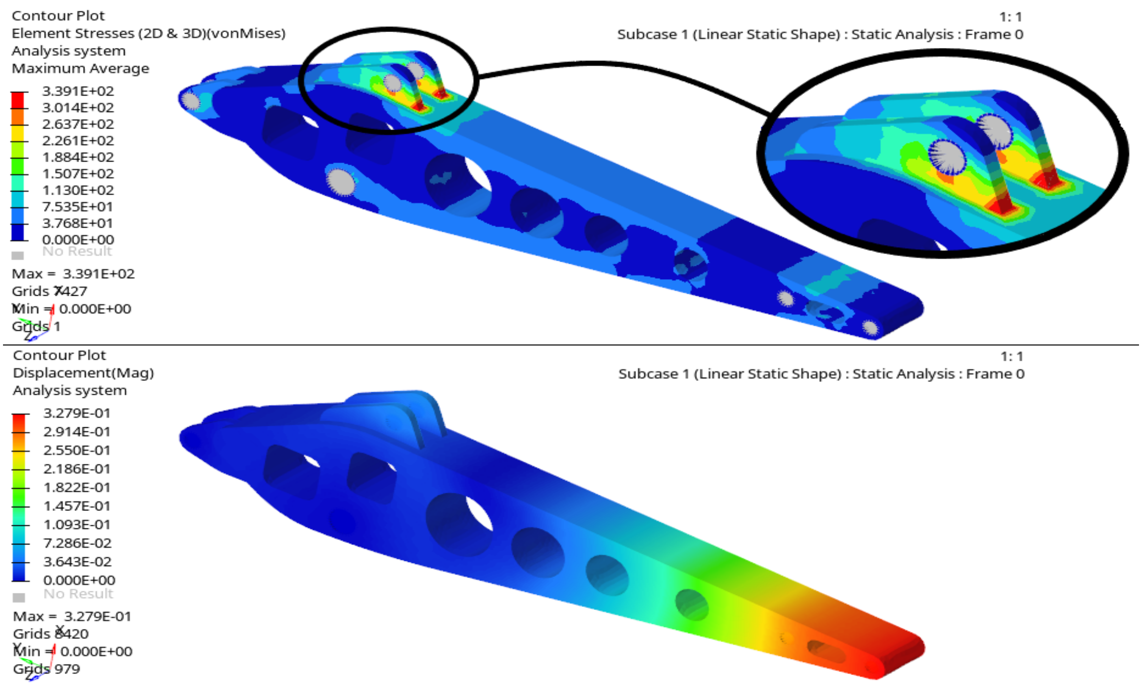

Fig. 4: Linear static analysis von mises stress and displacement results

III. FINDINGS

After finding the values of the forces generated in the excavator arm and determining the maximum stress and maximum displacement values by linear static analysis, fatigue analysis was applied to the excavator arm at this stage.

➢ *Fatigue Analysis of Excavator Arm*

Fatigue analysis was applied to the excavator arm. For the excavator arm material S355JR, the specifications YS, UTS, SRI1 (Fatigue Strength Coefficient), B1 (First Fatigue Strength Exponent), NC1 (Cycle Limit of Endurance), FL (Fatigue Limit) and SE (Standard Error of Log (N)) given in Figure 6 are defined. YS and UTS refer to the yield and tensile strength of the material. SRI1, B1, NC1, FL and SE properties are the parameters that affect the fatigue life of the material. In order to create the fatigue analysis in the HyperMesh program, the card structures given in Figure 5 are defined in Load Collectors, respectively. A new Load Step has been opened and Fatigue has been selected for Analysis Type. In the FATDEF, FATPARM and FETSEQ rows, Load Collectors selections containing data related to fatigue analysis were made.

ISSN No: -2456-2165

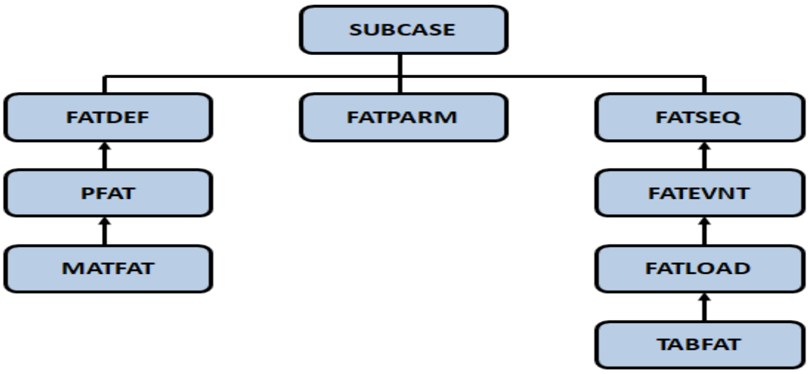

Fig. 5: Card structures that need to be created in Hyper Mesh for fatigue analysis

In FATDEF, data including material parameters and component properties that affect fatigue life are defined. FETSEQ is the main load collector. It adds up all the fatigue loading and creates a loading sequence. In FATPARM, material properties, component properties and all other parameters except loading are defined.

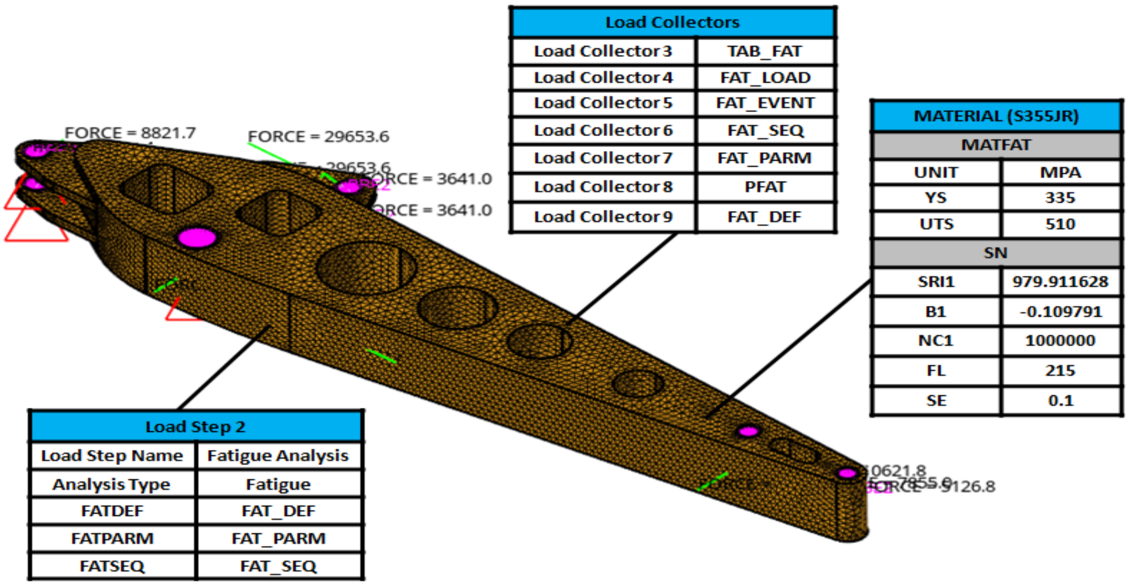

Fig. 6: Operations applied to the excavator arm for fatigue analysis

The results given in Figure 7 and Figure 8 were obtained from the fatigue analysis. As a result of the applied loads, the fatigue distribution on the excavator arm was examined and it was tried to determine in which areas the fatigue limit was exceeded. When the result given in Figure 7 is examined; It was observed that the red colored areas were safe, but the fatigue limit was exceeded in the marked area, especially in the blue colored areas, and these areas were unsafe. When the result given in Figure 8 is examined; It is safe as no damage will occur as a result of the loading applied in blue colored areas. However, it has been determined that these areas are unsafe as damage will occur as a result of stress accumulation during loading in the red colored areas in the marked area.

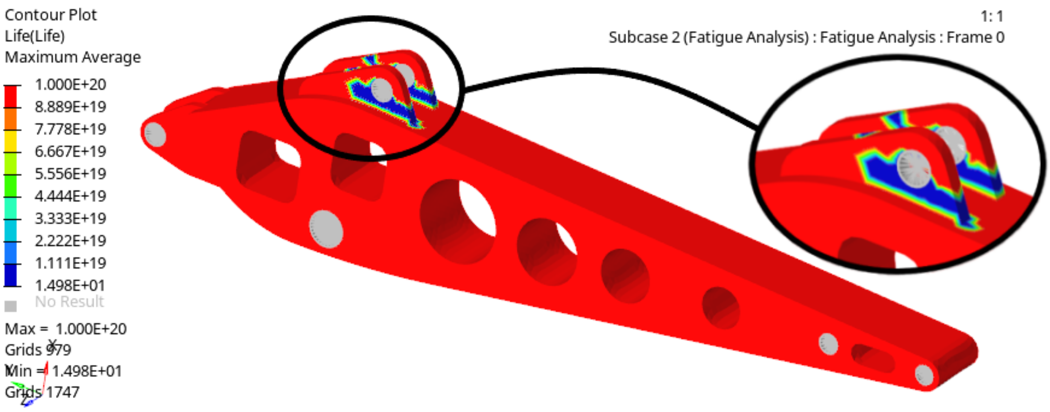

Fig. 7: Fatigue analysis life result

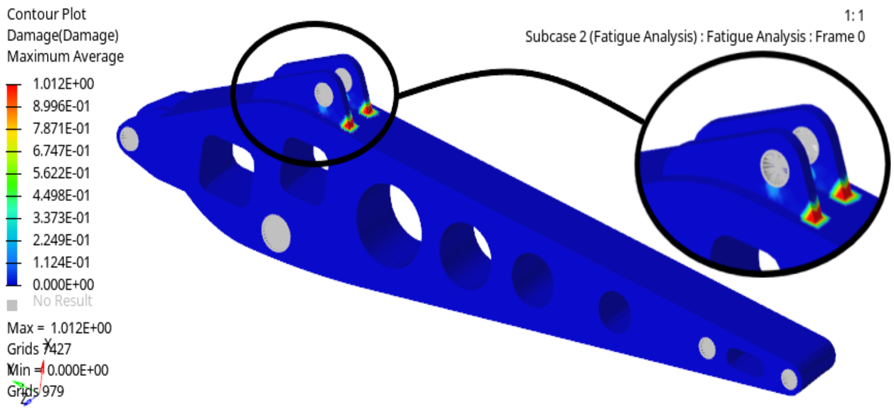

Fig. 8: Fatigue analysis damage result

An important point to note here is that the fatigue behavior moves in parallel with Von Mises stresses. In areas where the fatigue limit is exceeded on the excavator arm, von Mises stresses have also seen their maximum values. In order for the excavator arm to be more resistant to fatigue and safer, improvements and arrangements to be made in areas where the fatigue limit is exceeded will also reduce the maximum stress value in the excavator arm.

- *New Design and Analysis as a Result of Fatigue Analysis*

In line with the data obtained as a result of the fatigue analysis, a more durable and safer design was tried to be created for the excavator arm in terms of damage and life by using the SolidWorks design program. Smoother transitions are preferred in areas where damage may occur as a result of stress accumulation caused by sharp transitions during loading. In terms of material life, the thickness has been increased in the areas where problems may occur on the excavator arm. Linear static analysis was applied to the design created as a result of fatigue analysis. The results of linear static analysis are given in Figure 9. According to the results of the analysis, the maximum stress in the new excavator arm design was determined as 250,5 MPa and the maximum displacement was 0,3199 mm. It was observed that fatigue behavior moved in parallel with Von Mises stresses.

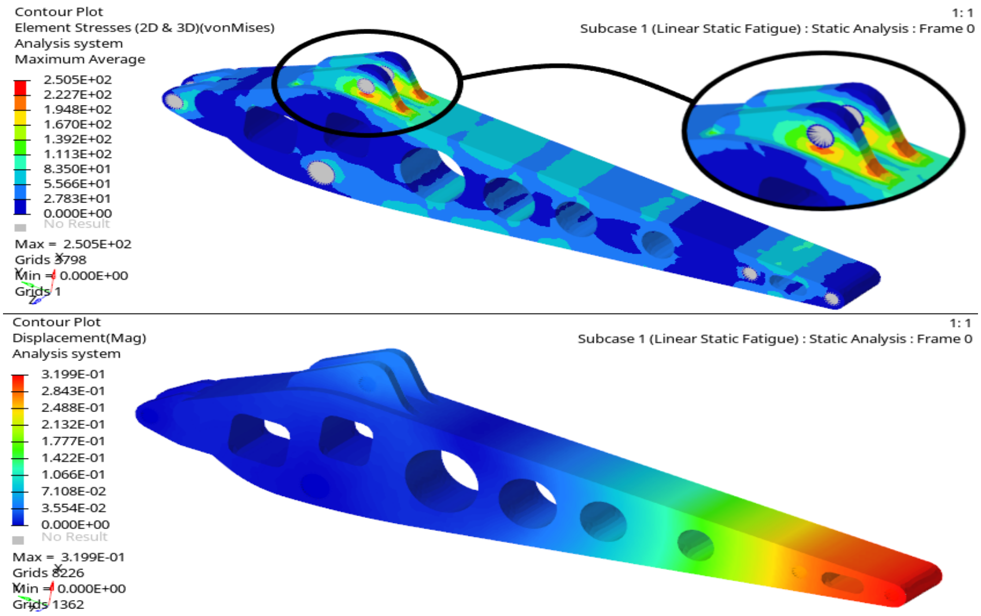

Fig. 9: Linear static analysis von mises stress and displacement result

With the design created after fatigue analysis, better results were obtained in maximum stress and maximum displacement values compared to the previous design. A more fatigue-resistant and safer design has been created for the excavator arm. The excavator arm designs created before and after the fatigue analysis are compared in Table 1.

Table 1: Comparison of excavator arm designs

Design Phase	Weight (g)	Max. Stress (MPa)	Max. Displacement (mm)	Weight Gain (%)
Before Fatigue Analysis	6737,42	339,1	0,3279	+2,94
After Fatigue Analysis	6802,02	250,5	0,3199	-0,96

ISSN No: -2456-2165

IV. CONCLUSION

In this study, if the excavator lifts a load of 1000 kg, the forces that will occur depending on the load lifted on the excavator arm are calculated. In line with the force values obtained, fatigue analysis was applied to the excavator arm and the areas where the fatigue limit was exceeded were determined. According to the results obtained in the fatigue analysis, a fatigue-resistant and safe design was created for the excavator arm.

- Using the SolidWorks design program, a solid model of the excavator arm was created and the forces that would occur depending on the load lifted on the excavator arm were calculated if the excavator lifted a load of 1000 kg.
- Linear static analysis was applied to the excavator arm in the HyperMesh program. The maximum stress and maximum displacement values of the excavator arm were found to be 339,1 MPa and 0,3279 mm, respectively.
- Fatigue analysis was applied to the excavator arm and the areas where the fatigue limit was exceeded were determined. According to the results obtained from the fatigue analysis, a fatigue-resistant and safe design was created for the excavator arm. With the design created after fatigue analysis, better results were obtained in maximum stress and maximum displacement values compared to the previous design. The maximum stress and maximum displacement values in the excavator arm were found to be 250,5 MPa and 0,3199 mm, respectively.

REFERENCES

[1]. Şık, A., Önder, M. ve Korkmaz, M. S. (2015). Determination of fatigue strength of vehicle rims by structural analysis, Gazi University Journal of Science, 3(3), 565-574.

[2]. Yıldız, B. (2019). Numerical investigation of fatigue behavior of patched and unpatched aluminum pipes, Master's Thesis, Batman University Institute of Natural and Applied Sciences, Department of Mechanical Engineering, Batman

[3]. Doğan, M. (2007). Fatigue analysis in vehicle elements, Master's Thesis, Uludag University Institute of Natural and Applied Sciences, Department of Mechanical Engineering, Bursa

[4]. Zhao H, Zhai Z, Mou Y, et al. (2022). Fatigue life prediction and reliability analysis of the forage crusher rotor. Journal of Mechanical Sciences Technology, 36: 1771–1781.

[5]. Tong Y B., (2017). Establishment of Fatigue Test Load Spectrum of Excavator Boom and Its Fatigue Life Prediction Xi'an: Chang' an University

[6]. Düzcan, Y. (2019). Development of vehicle suspension components with structural optimization techniques, Master's Thesis, Uludag University, Institute of Natural and Applied Sciences, Department of Mechanical Engineering, Bursa.

[7]. Yıldız, B. S. (2016). Development of new generation integrated techniques for innovative product design in the automotive industry, Ph.D. Thesis, Bursa Technical University, Institute of Natural and Applied Sciences, Department of Mechanical Engineering, Bursa.

[8]. Qi W, Sun X. (2014). Analysis on fatigue of hinge sleeve in cubic press based on ANSYS. Applied Mechanical Materials, 543–547, 323–327.

[9]. Chan T-C, Ullah A, Roy B, et al. (2023). Finite element analysis and structure optimization of a gantry-type high-precision machine tool. Sciences Report, 13: 13006.

[10]. Konez, O., Metin, M. ve Demir, Ö. (2019). Structural analyses of the Y32 bogie Chapter 2: Fatigue analyses, European Journal of Science and Technology, (17), 395-411. DOI: 10.31590/ejosat.604857.

[11]. Li C L, Song S S and Han Z N., (2014). Fatigue Reliability Analysis of Frame Based on Code Design-Life, Journal of Graphics 35 42-5.