Available online at www.sciencedirect.com

ScienceDirect

Procedia Structural Integrity 33 (2021) 75–83

Structural Integrity
Procedia

www.elsevier.com/locate/procedia

IGF26 - 26th International Conference on Fracture and Structural Integrity

Topology optimization and fatigue analysis of a lifting hook

Ibrahim T. Teke[a]*, Mustafa Akbulut[b], Ahmet H. Ertas[a]

[a]*Department of Mechanical Engineering, Faculty of Engineering & Natural Sciences, Bursa Technical University, Bursa 16330, Turkey*
[b]*TUBITAK Marmara Research Center, Kocaeli 41400, Turkey*

Abstract

In this study, a lifting hook has been redesigned with topology optimization and fatigue analysis has been done. With a density-based method, volume minimization has been achieved. All processes have been realized with the help of finite element method implemented in a commercial software. A standard lifting hook model has been used in the process. After optimization, the standard lifting hook was remodeled with CAD software. Then the new model was analyzed. This process has been repeated for three different models. On the basis of the obtained results, the topology optimization undertaken within the scope of the study demonstrates that it is possible to redesign a lifting hook with reduced weight and at the same time to satisfy the required strength properties.

Keywords: ANSYS; Finite Element Method (FEM); lifting hook; topology optimization; fatigue analysis

---

1. Introduction

Engineering is a set of research and development processes including design, analysis and production. In many of these processes, there are certain challenges that need to be tackled. These problems can be different from each other in some aspects. There are also many engineering tools that solve specific problems encountered and improve their results. The utilization of topology optimization methods is one of these tools, which helps engineers to achieve some engineering objectives on specific problems [Arora (2017)]. One of the problem areas in which topology modification is used is weight reduction in mechanical designs.

---

* Corresponding author.
E-mail address: ibrahimtekeiu@gmail.com

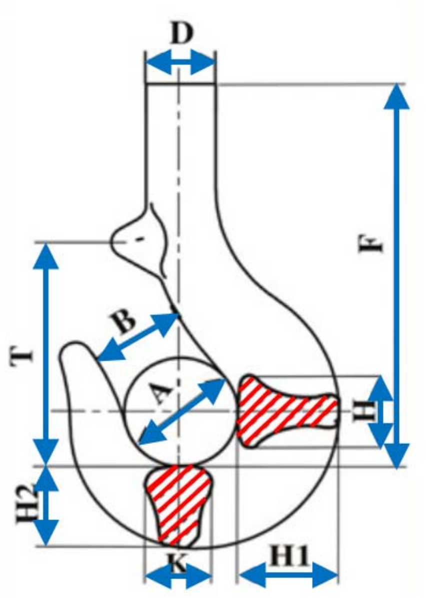
10.1016/j.prostr.2021.10.011

Hooks are commonly used in construction sites, transport centers for lifting and placing heavy-weight objects. These hooks vary in carrying different weights. With the increase of load capacity of a lifting hook, its dimensions become bigger and also its weight heavier. The increased weight will cost a lot of material and money to the manufacturer. That's why alternative designs are needed. There are some important studies in the literature. For instance, a stress-based analysis of hooks with different cross-sections has been conducted by Bundela and Shrivistaya (2017). In another study conducted by Tigabey (2018), fatigue analysis of different models for lifting hooks has been carried out. Maneengam et al. (2017) investigated the dimensional optimization of a lifting hook. In this study, a stress-based optimization approach similar to that of a previous model generated by Solanki et al. (2015) has been used. There are also some studies benefitting from topology optimization directly. For instance, Hajare and Jadhav (2020) undertook topology optimization of a laminated hook. In that study, only equivalent stress and deformation of the hook were investigated numerically and also experimentally. Topology optimization of a forging hook has also been examined by Thejomurthy and Ramakrishn (2018).

The objective of this study is to examine and make comparisons of different optimized models with respect to their fatigue life, strength, and also weight. For this purpose, stress analyses of lifting hooks were carried out using a finite element method for various optimized shapes. Based on the predicted stress states, fatigue life analyses were performed. The results of this study will provide the designer with some guidelines in designing lifting hooks.

2. Models and Simulation

The model used in the study is the standard DIN 15401-2.5 hook. The geometrical properties and dimensions – according to DIN 15401-2.5 standards- are shown in Figure 1. It has a load capacity of 5 tons maximum.



DIN GM 15401-2.5 Hook Dimensions (mm)

A	B	D	F	H	H1	H2	K	T
63	50	42	263	56	67	58	48	135

Fig. 1. DIN 15401-2.5 type hook and its dimensions.

In simulation "Structural Steel" material available in ANSYS software material database has been used. In the FE model, a 3D 10-node tetrahedral solid element, SOLID92, has been used. This element has plasticity, stress stiffening, large deflection, and large strain capabilities. The hook has been modeled using the material properties of 2.05E5 MPa and 0.30 for young modulus and Poisson's ratio respectively.

Static analysis has been done with 5 tons -which approximately equals to 49050 N- load and fatigue analysis of the standard hook model has been achieved via ANSYS 2017 R2. In the finite element models, all displacement and rotation degrees of freedom of the nodes on the top surface of the hook have been fixed. All degrees of freedom on the bottom end of the hook have also been fixed except the loading direction. A tensile load is applied in plane to the hook. The cyclic loading has been applied in two load steps. First, the load has been incrementally increased to its maximum value –which is 49050 N-, and the resulting stress state has been obtained. In the second load step, the load has been incrementally decreased to its minimum value, which is zero. In the next load cycles, the stresses have been assumed to fluctuate between the stress levels corresponding to maximum and minimum loads.

Then topology optimization of the standard hook model and the fatigue analysis of the new models are done. Figure 2 shows both the standard hook model taken into consideration and also optimized models.

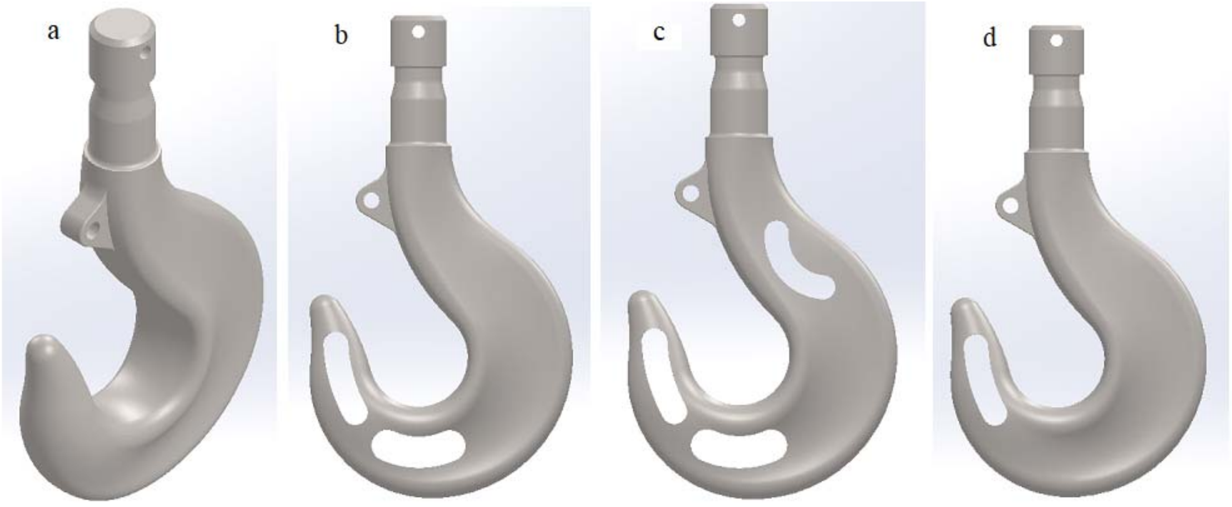


Fig. 2. (a) standard hook model; (b) first optimized model; (c) second optimized model; (d) third optimized model.

Table 1 contains the obtained values after the first load step which can be accepted as static analysis. Figure 3, on the other hand, shows the topology optimization result of the considered hook.

Table 1. Static analysis results.

Model Name	Weight (kg)	Equivalent Stress [MPa]	Maximum Principal Stress [MPa]	Total Deformation [mm]
Standard	5.762	224,85	223,78	0,37745
First	5.195	301	310,12	0,42651
Second	4.933	300,98	306,56	0,44704
Third	5.477	224,69	223,52	0,37805

3. Results

In fatigue analysis, life, damage, and safety factors were obtained for both the standard model and optimized models as well.

In fatigue life analysis, a stress-based approach was employed for all cases. Because the minimum fatigue life of the standard model has been found as 17162 cycles (as shown in Figure 4) that life value was chosen as the design criterion for the other models.

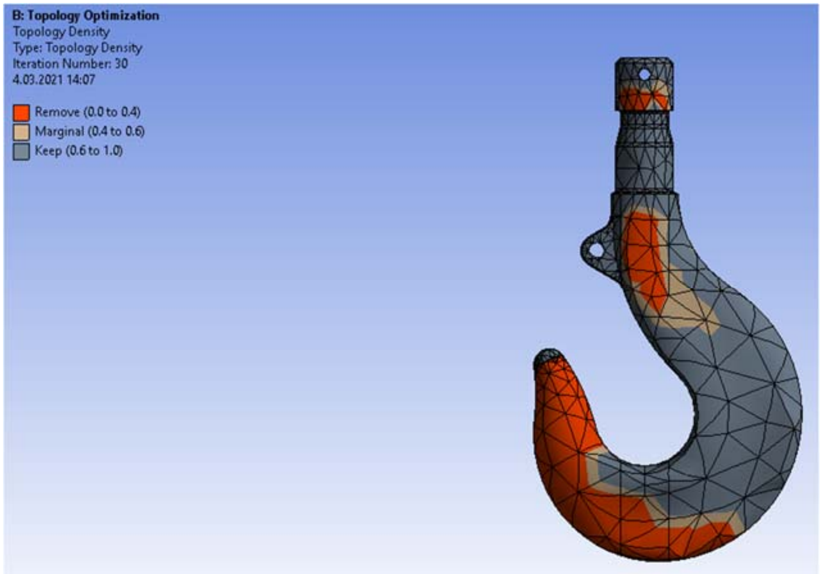


Fig. 3. Topology optimization of the hook.

In damage analyses, directly the definition of damage is used. Hence damage contours (shown in Figures 5, 8, 11, and 14) were obtained by taking into account previously chosen design life which is 17162 cycles in our cases. In these figures, the values that are bigger than one indicate that damage will occur, otherwise, the structure will be safe. In other words, if the maximum damage value is bigger than one then damage will occur before reaching the design life. Otherwise, the structure will be safe.

Since the material in question is isotropic structural steel, in safety calculation, Von-mises equivalent stress values are computed and the ratio of yield strength to equivalent stress is taken as a safety factor. The value of safety factor less than one means again that damage will occur before reaching the defined design life. Otherwise, the structure will be safe. As the safety factor grows, the reliability of the structure will increase.

3.1. The standard model's fatigue analysis

Figures 4, 5, and 6 show fatigue life, damage, and safety factor of the standard model, respectively. High stresses develop at regions on the inner surface of the hook close to the inner curvilinear surface because of the stress concentration as expected. The maximum stress develops close to the inner surface of the hook. This location also conforms to the fatigue crack initiation sites.

3.2. The first optimized model's fatigue analysis

Figures 7, 8, and 9 show fatigue life, damage, and safety factor of the first optimized model, respectively. High stresses again develop at regions on the inner surface of the hook close to the inner curvilinear surface because of the stress concentration. The maximum stress develops close to the inner surface of the hook.

3.3. The second optimized model's fatigue analysis

Figures 10, 11, and 12 show fatigue life, damage, and safety factor of the first optimized model, respectively. High stresses again develop at regions on the inner surface of the hook close to the inner curvilinear surface because of the stress concentration. The maximum stress develops close to the inner surface of the hook.

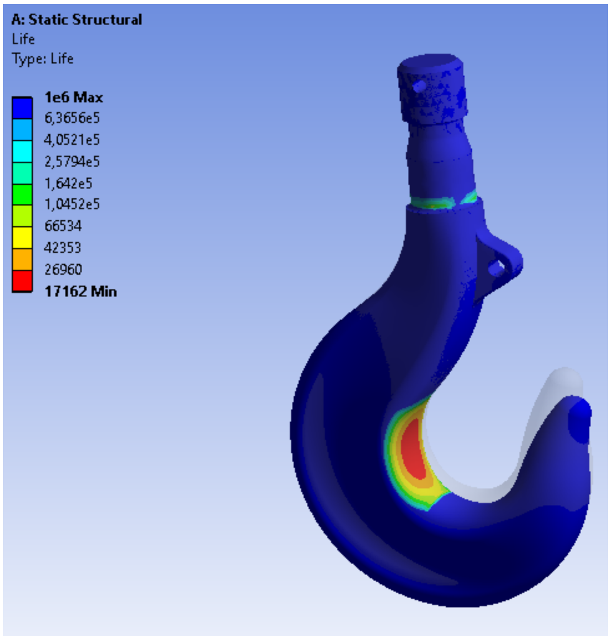


Fig. 4. Fatigue life of the standard model.

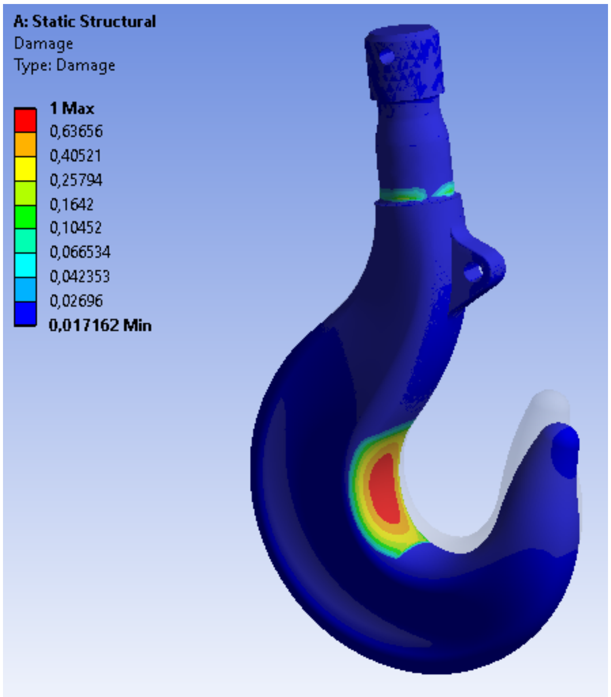


Fig. 5. Damage of the standard model.

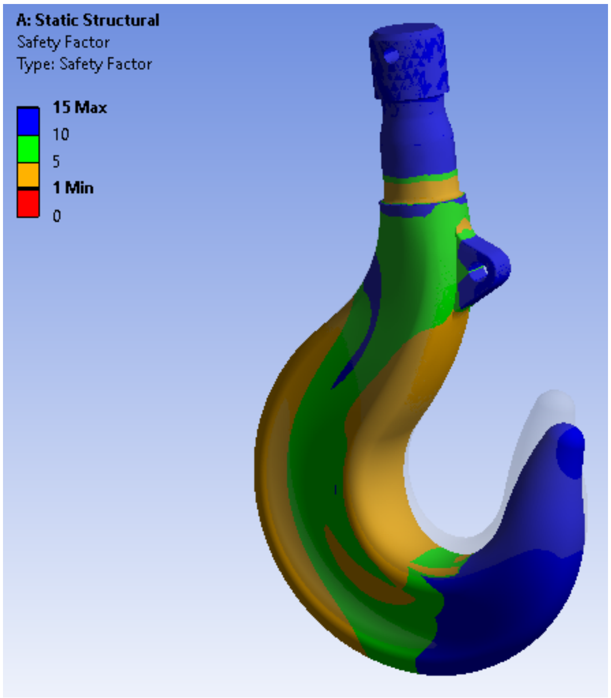


Fig. 6. Safety factor of the standard model.

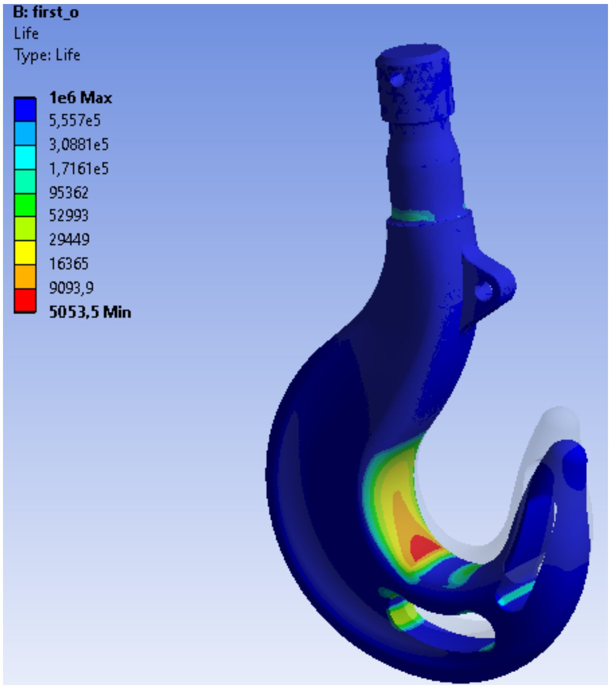


Fig. 7. Fatigue life of the first optimized model.

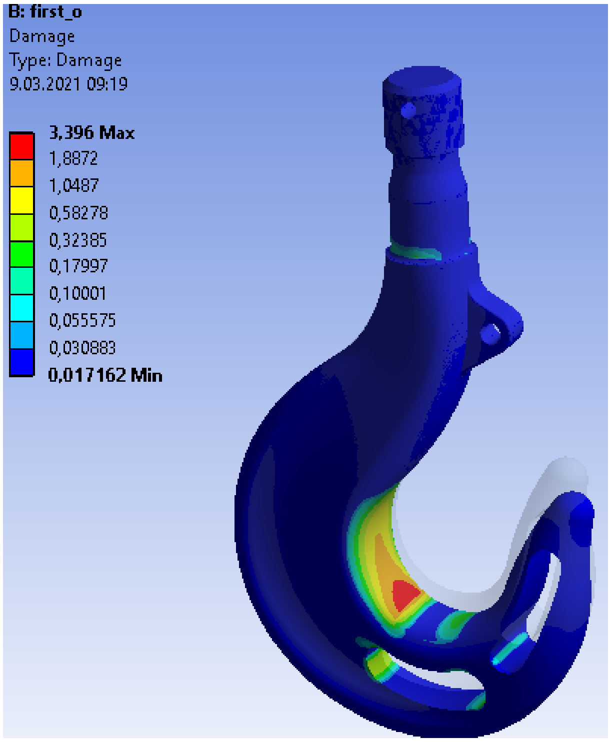


Fig. 8. Damage of the first optimized model.

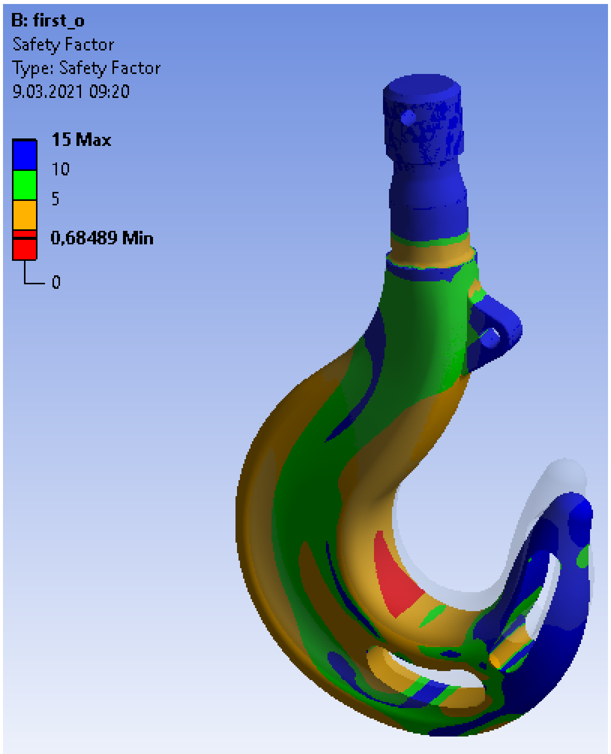


Fig. 9. Safety factor of the first optimized model.

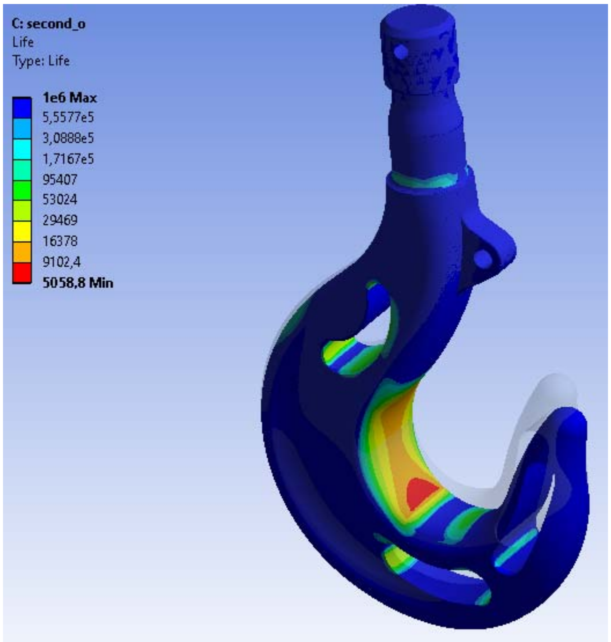


Fig. 10. Fatigue life of the second optimized model.

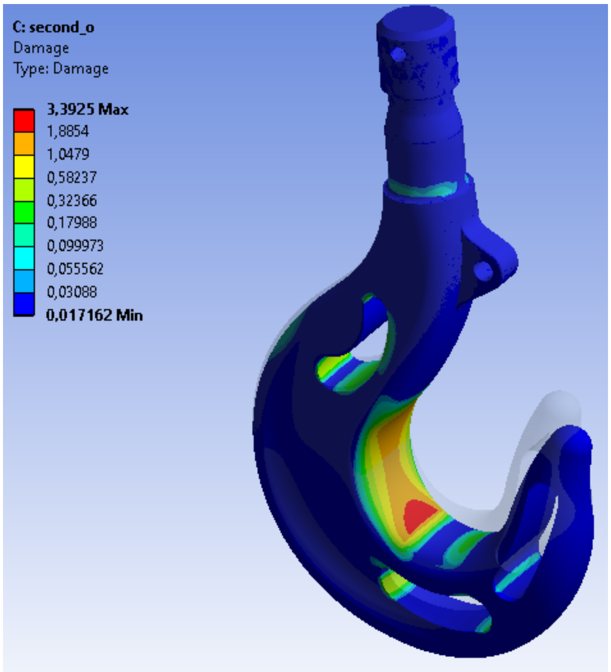


Fig. 11. Damage of the second optimized model.

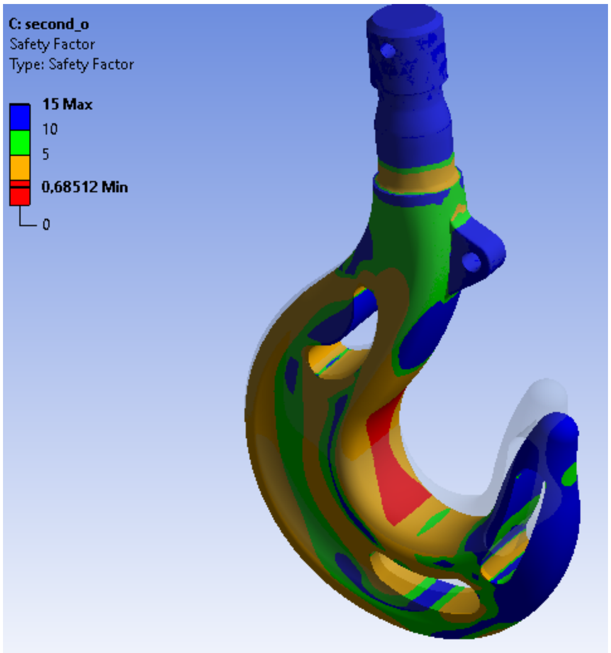


Fig. 12. Safety factor of the second optimized model.

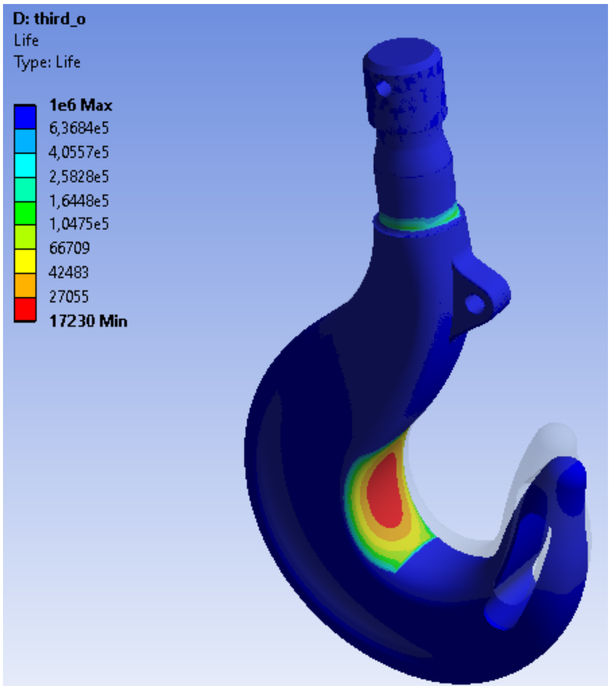


Fig. 13. Fatigue life of the third optimized model.

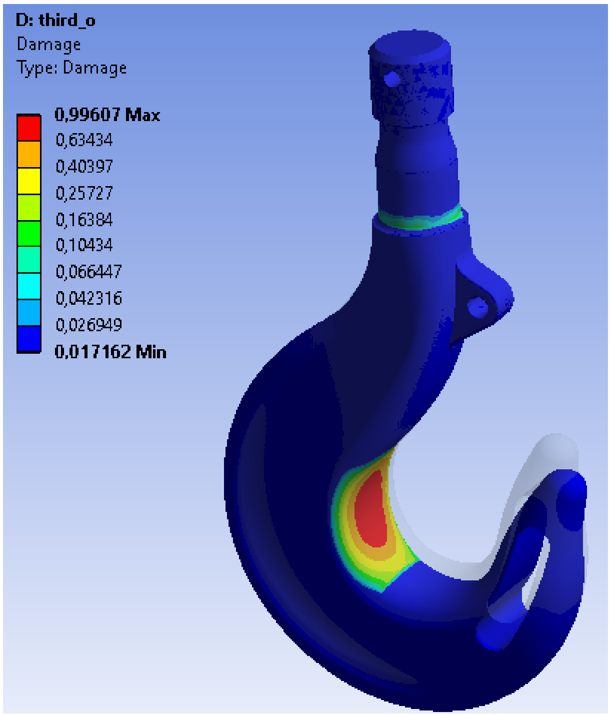


Fig. 14. Damage of the third optimized model.

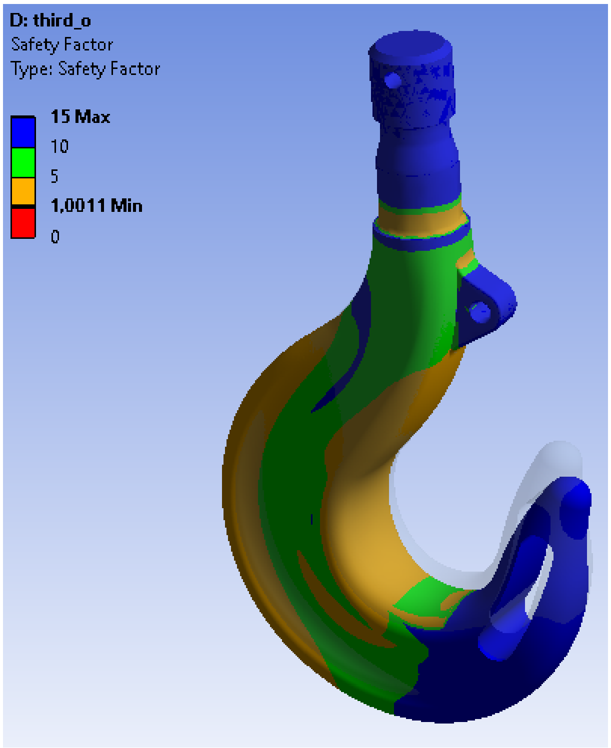


Fig. 15. Safety factor of the third optimized model.

3.4. The third optimized model's fatigue analysis

Figures 13, 14, and 15 show fatigue life, damage, and safety factor of the first optimized model, respectively. High stresses again develop at regions on the inner surface of the hook close to the inner curvilinear surface because of the stress concentration. The maximum stress develops close to the inner surface of the hook.

The comparison among the models in terms of fatigue life, safety factor and damage are summarized in Table 2.

Table 2. Comparison of alternative designs.

Model Name	Weight (kg)	Minimum life [Cycle]	Minimum Safety Factor	Damage
Standard	5,762	17.162	1,000	1,000
First	5,195	5.053	0,685	3.396
Second	4,933	5.059	0.685	3,392
Third	5,477	17.230	1,001	0,996

4. Conclusions

Fatigue life, damage, safety factor, maximum stress, deformation, and also weight of the standard hook model were investigated for all alternative models designed within the scope of this study. Based on the discussions in the preceding sections, the following conclusions can be drawn:

- Numerical analysis showed that the number of cycles to failure changes depending on the geometry of the hook.
- Among these studies, the most appropriate model is the third model. It does not differ from the original model too much. For instance, in addition to fatigue life, damage, and safety factor; equivalent stress, and also total deformation are approximately the same for both the standard model and the third optimized one. The other models (that is, the first and second models) are not appropriate at least for the loading of 5 tons - which approximately equals to 49050 N- load which is considered in this study.
- When the first and second optimized models were used, the stress values of the models increase approximately by % 30. The discrepancies might result from stress concentration arising from opening holes onto the hooks. But when the third optimized model was used, it is seen that the stress value is equal to the standard one.
- Consequently, opening holes onto a standard crane hook especially to the back of the maximum stress area and back of the load area decrease the fatigue life of the hooks approximately by 70 %.
- The third optimized model has similar fatigue life with the actual model. However, its weight 285 grams less than the standard model which is a big advantage in terms of economical point of view. This is also important in terms of environmental regulations.
- To eliminate stress concentration and also increase the fatigue life of a hook, optimized geometry should be used. Considering four different models in this study, the geometry of the third optimized model can be utilized, for which fatigue life value is approximately close to the original model, but it is lighter than the original model, which makes it attractive in the industry especially in terms of economical point of view.
- Comparing both fatigue life and also weights of the models, it is obvious that the geometry and accordingly the method used -which is topology optimization in this study- play an important role in the fatigue strength of the hooks.

References

ANSYS user's manual, version 2017r2.

Arora, J. S., 2017. Introduction to Optimum Design: Introduction to Design Optimization. Elsevier Press, New York.

Bundela, S., Shrivastava, A. K.,2017. Design and Static Stress Analysis of Various Cross Section of Hook. International Journal of Recent Technology Science and Management 2(12), 52-59.

DIN 15401-2.5 hook standards.

Hajare, P.R., Jadhav, S.M., 2020. Experimental Stress Analysis and Optimization of Crane Lifting Tackle. International Research Journal of Engineering and Technology 7(8), 215-221.

Maneengam, A., Saisirirat, P., Suwankan, P.,2017. Hook design loading by the optimization method with weighted factors rating method, International Conference on Alternative Energy in Developing Countries and Emerging Economies. Bangkok, Thailand, pp. 337-342.

Solanki, M., Bhatt, A., Rathour, A., Thakka,r S., 2015. Weight Optimization in Crane Hook. Journal of Mechanical and Civil Engineering 1(6), 192-196.

Thejomurthy, M.C., Ramakrishn, D.S., 2018. Topology Optimization and Analysis of Crane Hook Model. Journal of Engineering Research and Application 8(9), 60-64.

Tigabey, S., Optimization and Fatigue Analysis of Crane Hook Using Finite Element Method. MSc Thesis Dissertation, Addis Ababa University, 2018.