Uluslararası İleri Doğa Bilimleri ve Mühendislik Araştırmaları Dergisi
Sayı 7, S. 163-169, 6, 2023

**Araştırma Makalesi**

International Journal of Advanced Natural Sciences and Engineering Researches
Volume 7, pp. 163-169, 6, 2023

**Research Article**

https://as-proceeding.com/index.php/ijanser
ISSN: 2980-0811

# TOPOLOGY OPTIMIZATION OF A HANGER FOR THE CRANE OF A BOAT

Feridun KARAKOÇ[1,*], Rafael Villalobos TORRES[2], Ahmet DAYANÇ[3], Melih CANLIDİNÇ[4]

[1] Kutahya Dumlupinar University, Faculty of Engineering, Departmant of Mechanical Engineering, Kutahya, feridun.karakoc@dpu.edu.tr, ORCID: 0000-0002-6210-4070
[2] The University of Oviedo, Faculty of Engineering, Departmant of Mechanical Engineering, Oviedo, UO260242@uniovi.es, ORCID: 0009-0000-5433-6405
[3] Kutahya Dumlupinar University, Faculty of Engineering, Departmant of Mechanical Engineering, Kutahya, ahmet.dayanc@dpu.edu.tr, ORCID: 0000-0002-5214-9021
[4] Kutahya Dumlupinar University, Faculty of Engineering, Departmant of Mechanical Engineering, Kutahya, melih.canlidinc@dpu.edu.tr, ORCID: 0000-0002-4011-9490

*(Received: 06 July 2023, Accepted: 24 July 2023)*

(5th International Conference on Applied Engineering and Natural Sciences ICAENS 2023, July 10 - 12, 2023)

**ATIF/REFERENCE:** Karakoç, F., Torres, R. V., Dayanç, A. & Canlıdinç, M. (2023). TOPOLOGY OPTIMIZATION OF A HANGER FOR THE CRANE OF A BOAT. *International Journal of Advanced Natural Sciences and Engineering Researches*, 7(6), 163-169.

***Abstract-*** Topology optimization is the process of optimizing the shape and material distribution of a part. This study examines the topology optimization of the hanging apparatus of a boat's crane. The goal here is to design a lighter and more efficient hanging apparatus. Initially, topology optimization was performed using a CAD software such as Autodesk Fusion 360. This software aids engineers in creating complex designs, simulating real-world scenarios, and optimizing material distribution. The design process began with the use of a non-equilateral triangle to determine the shape of the part. Later, two cylindrical rings that attach to the pole were added. These steps completed the initial geometry. Subsequently, it was indicated that the part needed to be light and resistant to weather conditions. Therefore, aluminum 6061 material was chosen. The material properties were chosen to ensure the proper functioning of the part. A static analysis was performed to determine the strength of the part. The forces and constraints applied to the part were simulated considering a one-ton capacity. A displacement of 1.32 millimeters and a safety factor of 6 were found. Then, following the topology optimization, an analysis of the new geometry resulted in a displacement of approximately 4 millimeters for the part. The predetermined minimum safety factor is 3, and the factor found in the analysis was 3.7. This is an acceptable value to ensure the structural integrity and safety of the part. The optimization process aimed to reduce the mass of the part. By preserving the initially determined safety factor, savings were made in the material and a successful reduction of approximately 42% in mass was achieved. This study demonstrates that topology optimization is a significant tool in engineering design and CAD software such as Autodesk Fusion 360 can be effectively used in this process. The analyses and optimizations conducted reveal that it's possible to design a lighter and more efficient hanging apparatus. This provides numerous advantages such as cost savings, performance increase, and reduction of environmental impact.

*Keywords: Fusion 360, Lightweighting, Structural Analysis, Topology Optimization*

## 1. INTRODUCTION

Topology optimization is a powerful technique used in engineering and design to optimize the shape and distribution of material within a given design space, with the objective of achieving desired performance criteria while minimizing mass and maximizing structural efficiency. By exploring and analyzing various design alternatives, topology optimization allows engineers to push the boundaries of conventional design practices, resulting in lighter and more efficient structures that meet safety requirements.

For instance, the inner bodies of gears are typically filled with solid material, which significantly increases the weight of the systems they are used in [1]. However, if the body weights of the gears can be reduced during the design process, the mechanical properties expected from the systems can be achieved with a minimum material cost [2]. A study has focused on the optimization of material placement in design rather than material selection. At the end of the study, the analysis of generative design geometry and lattice structure pattern yielded different results, and these outputs were also compared in terms of weight savings [3].

In a study examining the damage factor estimation of crane hooks by determining the trend of the load situation, FEM was used to predict the relationship between load situation and deformation. A load-deformation database containing the relationship between the load situation and deformation of the crane hook was created using numerical computation. Upon completion of the study, it was found that the load in the damaged hook was located between the end point and the lowest point, and its direction was towards gravity [4].

In a study predicting the dimensions of the hook for various section topologies while preserving depth and cross-sectional area, it was concluded that the trapezoidal section had the least stress [5].

In a study on the optimization of the crane hook, a solid model was created with the help of ANSYS workbench to predict the stress pattern of the crane hook under load. The pattern of real-time stress concentration was obtained with the 3D model of the crane hook, considering different cross-sectional areas. The stress pattern was calculated for various section topologies with triangular, rectangular, circular, and trapezoidal sections, and it was found that the rectangular section area provided minimum stress and deformation when the same area was preserved [6].

In one study, a trapezoidal crane hook model was created using CATIA V5R20. Then, using FEM, the stress point was estimated after applying a load of 2 tons. They also analyzed the effect of changes in the length of the two parallel sides of the trapezoidal hook on stress [7].

In another study on reducing the weight of the crane beam, the cost of the beam was reduced and its lifespan was increased. A mathematical design was made for the crane component using ANSYS workbench, and hook optimization was done using the trapezoidal cross-sectional area [8].

In another study, work was done on the optimization of the self-weight of low carbon steel. Considering the self-weight on the crane hook and component load, the optimization of the hook mass under the effect of static load was aimed. Finite element analysis was used for the verification of the final geometry and for the shape optimization of the crane hook. Also, during the optimization process, geometric and manufacturing constraints were taken into account, and the results showed that the optimized hook was 14% lighter than the original crane hook mass [9].

Finally, a solid model was created to predict the stress pattern of the crane hook, and the real-time stress concentration pattern was obtained using ABAQUS software [10].

In this paper, we present a case study on topology optimization conducted using Autodesk Fusion 360, a widely used computer-aided design (CAD) software. The primary goal of our experiment is to demonstrate the potential of topology optimization in reducing the mass of a designed part, while simultaneously considering the material utilization and safety coefficients. The study showcases the iterative process involved in designing and optimizing a part, providing valuable insights into the capabilities and benefits of topology optimization techniques.

To begin with, we utilize Autodesk Fusion 360, an advanced CAD software that integrates design, simulation, and manufacturing functionalities. Its intuitive interface and comprehensive toolset allow engineers to create complex designs and simulate real-world scenarios efficiently. Fusion 360

supports various optimization algorithms, making it an ideal platform for implementing topology optimization techniques.
Our focus in this study is on reducing the mass of the designed part while ensuring that safety and material utilization requirements are met. Mass reduction is a crucial aspect of modern engineering, as it directly affects the performance, cost, and environmental impact of a structure or component. By applying topology optimization techniques, we aim to identify the optimal material distribution within the given design space, resulting in a lighter yet structurally sound part.

Simultaneously, maintaining safety standards is of paramount importance. Safety coefficients are considered during the optimization process to ensure that the designed part can withstand expected loading conditions while providing an adequate level of structural integrity. The safety coefficients are defined based on industry standards and guidelines, which vary depending on the specific application and requirements of the part under consideration.
For this project it will be assumed that an engineering case applied to create a part for a specific use with all the details of how said part will be used and some details needed for case. With that information a part will be created according to what was asked and a topology optimization will be performed to make the part as good as it could be done with the programs available for such duty.

## 2. MATERIAL AND METHOD

### 2.1.Engineering Assumptions

The topology optimization for the crane hanging apparatus design has been executed to be used with an electric motor that will be attached to a pole and hang down to lift things. The apparatus is intended to have a carrying capacity that can withstand one ton and be attached to the pole in a way that allows 360-degree rotation. Moreover, the apparatus must be as light as possible structurally and needs to be resistant to weather conditions since it will be operating along the shoreline.
Considering the features of the designed part, the piece will be created to meet all these conditions in the best possible way. Initially, the basic shape of the part is chosen as a triangle. The shortest edge of the triangular structure is attached to the pole and is designed to have two circular sections where the bearing will be mounted. At the corner on the opposite side, there is a hole for connecting the motor with a chain. After obtaining the initial geometry, the analysis of the material was made. Since the material selection needed to be resistant to weather conditions, iron or steel was not preferred, and stainless steel was not used due to the weight constraint. Since there was no issue with cost and precision wasn't necessary during operation, aluminum was chosen as the material.

To meet one of the requirements, which is the crane being as light as possible, a topology optimization was carried out. Therefore, the minimum amount of material was used while preserving the necessary mechanical properties.

### 2.2.Initial Design

For the initial geometry a scalene triangle will be extruded with a hole in its most acute angle. Then, two holed cylinders will be attached to its shortest side with the axis of them being parallel to said side. These operations will complete the initial design of the crane.

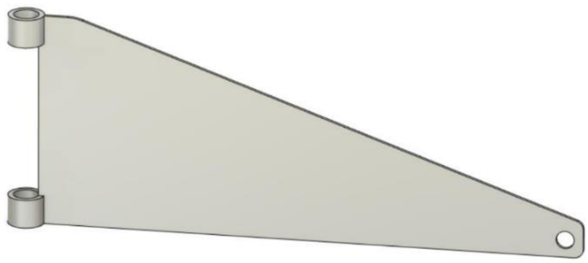


Fig 1. Initial Design of Hanger

According to our engineering case, it should be weather resistant, and also lightweighted, for that reason aluminum 6061 was selected. With its properties we can ensure a correct functioning of the part.
For the analysis to determine the strength of the structure, a static analysis was performed on the crane, considering the request made of a one-ton capacity.
For the analysis, the part was restricted with a rolling constrain in both inner parts of the cylinders of the left side and a fixed face that would be the lower section of the lower cylinder to imitate the stop. About the force applied, a bearing-like force was applied inside of the lower-right circle, its

value was one ton as requested by the company. Its important to notice that a bearing force should be used because that way the force is applied in the lower half of the inner face of the cylinder imitating a chain being hanged to it with the motor to lift the load.

## 2.3.Topology Optimization

Keeping in mind that we found a security factor of 6 as a result of first analysis, a topology optimization will be performed to save on material and keep working conditions for the part.
Initially for the analysis, a mesh and regions to keep must be created. In the picture below the mesh is displayed and the green semi-transparent volumes are the regions to keep.

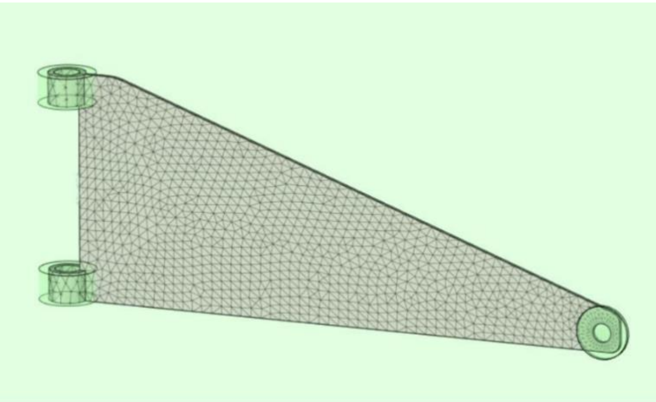


Fig 2. Volume Mesh and Regions to Keep

The cylindrical regions on the left, are intended to keep the anchor points to the pole, the one on the right is to keep the anchorage point for the motor. Later, the constraints and load are applied exactly as done in the static stress analysis, as a stop boundary it was set to keep 30% of the original mass, but we will adjust it until we get a SF of 3.

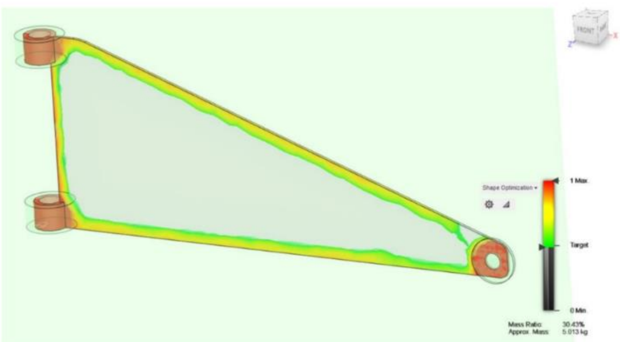


Fig 3. Optimization Failed at %30 Mass Ratio Approximately

As seen in the picture, if the boundary of 30% mass is kept, the part wouldn't work, for that reason we adjust it until it has at least 40%.

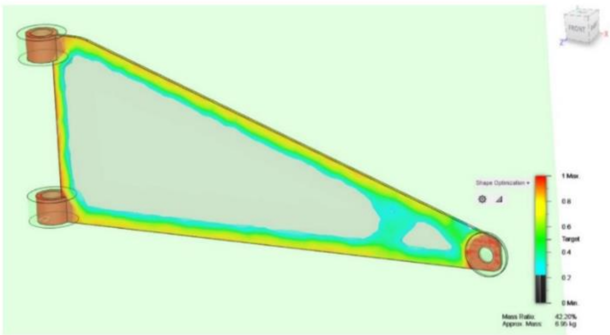


Fig 4. Optimized Geometry at %40 Mass Ratio Approximately

In doing so, we get a result suitable to promote and use to re-design the original part and test it again.

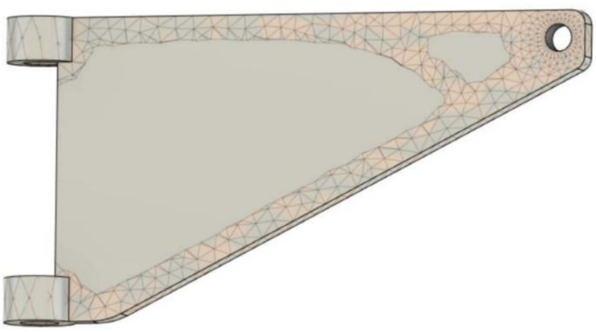


Fig 5. Selected Volume Meshes to Create a New Geometry

Once we get the result, we promote it on top of the original part and, following the shape of the result, we extrude cutting the extra material until obtaining the desired part.

Fig 6. CAD Model of the New Geometry

## 3. RESULTS

Once the analysis was performed, the results were as follows:

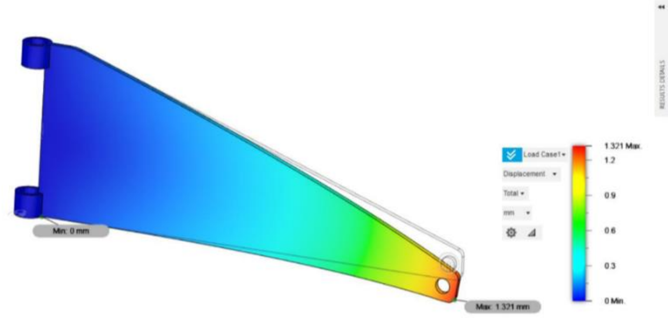



Fig 7. Displacement of Solid Model

As seen in the picture, the displacement is neglectable for the request made.

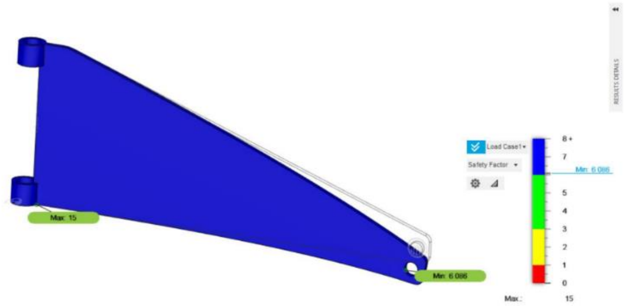



Fig 8. Security Factor of Solid Model

The security factor will dictate most of our study, since will be the boundary that will tell us about the desired result. We established a security factor of 3 to be acceptable, as we can see, we have an excess of material since we have a minimum SF of 6.

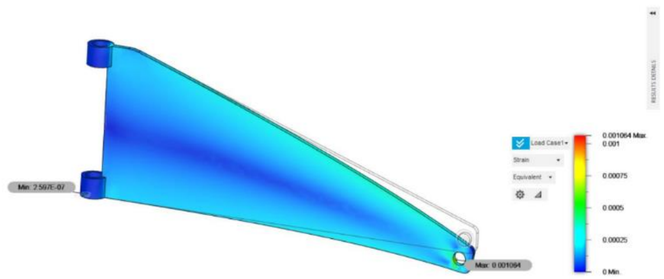


Fig 9. Strain Values of Solid Model

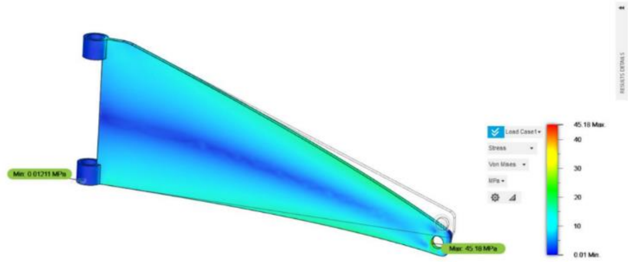


Fig 10. Stress Values of Solid Model

These results are orientative to verify the results of the optimization further on.

Once we have the new design ready, we apply a new study on the re-designed piece with the same constraints and loads of the original case, keeping the 1 ton load.


Fig 11. Displacement of Topology Optimized Model

Now we can see the displacement is of almost 4 millimeters, but although it is not too much, it was said in the request that precision was not required so 4 millimeters are fully acceptable.


Fig 12. Security Factor of Topology Optimized Model

This is the most important result in our study, the limit set was of a SF of 3, with the modifications made we get a 3.7, which is acceptable since less material might be compromising for the structure in other directions.

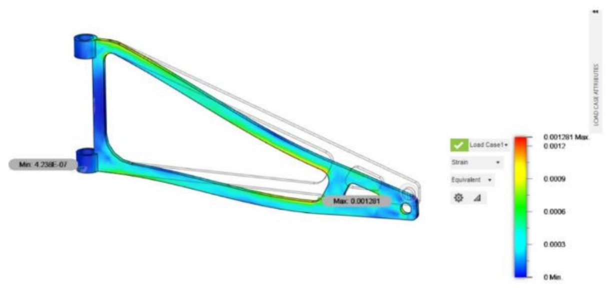



Fig 13. Strain Values of Topology Optimized Model

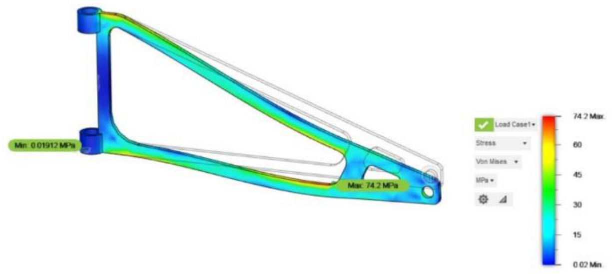



Fig 14. Stress Values of Topology Optimized Model

Compared with the results of the first test we can see that these are expected, reducing the SF almost to half will increase the Stress to about its double, so the results are true.

## 4. DISCUSSION

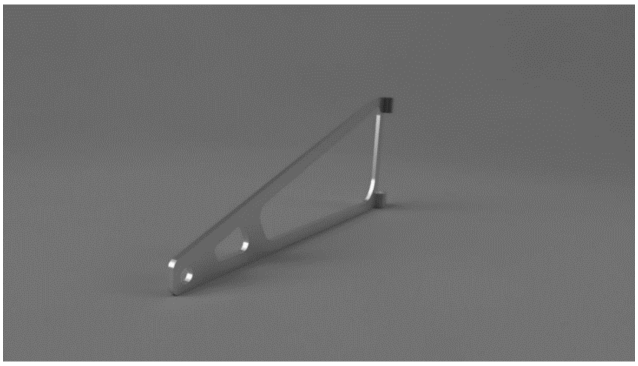
The optimization successfully achieved a remarkable reduction of 42% in the mass of the part. This reduction in mass not only leads to improved performance and cost savings but also contributes to minimizing the environmental impact associated with the part's production and usage. By systematically exploring the design space and redistributing material, we were able to uncover an optimized topology that exceeded our expectations in terms of mass reduction.

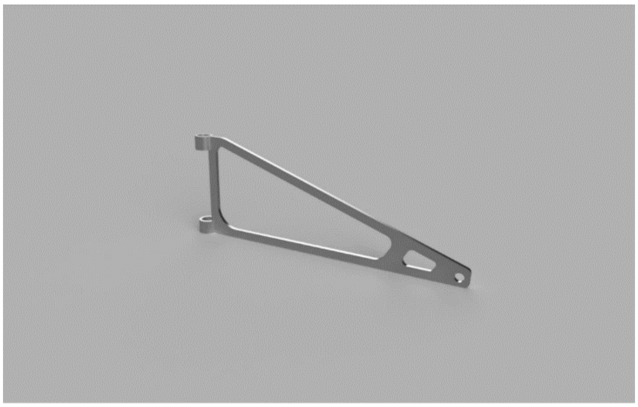
Furthermore, our study took into consideration the security factor, an essential aspect of engineering design. Although the security factor decreased from the initial value of 6 to 3.7, it remained within an acceptable range, ensuring the structural integrity and safety of the part. This demonstrates that topology optimization, when appropriately guided by safety coefficients and industry standards, can achieve substantial mass reduction without compromising the overall reliability of the structure.

The results obtained from our experiment indicate that topology optimization techniques, implemented through Autodesk Fusion 360, offer significant benefits for engineering design. By harnessing the capabilities of digital tools, engineers can explore innovative design alternatives, pushing the boundaries of conventional design practices. The reduction in mass achieved not only enhances the efficiency of the part but also opens new possibilities for weight-sensitive applications, such as aerospace and automotive industries.

Moreover, the successful integration of topology optimization into the design process demonstrates the efficiency and versatility of Autodesk Fusion 360 as comprehensive CAD software. Its intuitive interface, coupled with advanced optimization algorithms, enables engineers to create complex designs, simulate real-world scenarios, and optimize material distribution within the given design space effectively.

In conclusion, this case study highlights the effectiveness of topology optimization in reducing the mass of a designed part while considering material efficiency and safety requirements. The achieved mass reduction of 42% and a coefficient of almost 4 we can assure that the part compliments all desired features of the requested part. The render images of the part are provided in the image below.





## ACKNOWLEDGMENT

We conducted our topology optimization study with the resources provided by Kutahya Dumlupinar University and the support from Autodesk company, which provided the student license for the Fusion 360 software. We would like to express our gratitude to both institutions for providing us with this opportunity.

## REFERENCES

[1] TS 429, Dişli Çarklar-Genel ve Ağır Sanayii Makineleri İçin Silindirik Dişli Çarklar-Modüller, Türk Standartları Enstitüsü, Ankara.

[2] Ramadani R., Belsak A. Kegl M. Predan J. Pehan S. (2018). Topology Optimization Based Design of Ligthweight and Low Vibration Gear Bodies. Int J. Simul. Model, 17 (1): 92-104.

[3] Becergen, B., Cakmak, M., Maral, M., F., Dayanc, A. & Karakoc, F. (2022). Design Approaches on Inner Bodies of Gears with Methods Topology Optimization and Lattice Structures. European Journal of Science and Technology, (39), 85-90.

[4] Takuma, N., Takao, M., Kazuyuki, H., (2010). Damage factor Estimation of Crane Hook (A database approach with Image, Knowledge and simulation) in Research publishing services, ISBN: 978-981-08-5118-7.

[5] Pradyum, N., (2012). Computer Aided Analysis and Design of Hoisting Mechanism of an EOT Crane, Thesis, National Institute of Technology Rourkela.

[6] Chetan, N., Benkar, N. A. W., (2014). Finite Element stress Analysis of Crane Hook With Different Cross Sections, International Journal For Technological Research In Engineering, ISSN 2347-4718.

[7] Santosh, S., Ritesh, D., Manas, P., Narendra, Y., (2012). Study of Crane Hook Having Trapezoidal Section by Finite Element Method & Design Experiments, International Journal of Modern Engineering Research, vol. 2.Issue-4,pp-2779-2781.

[8] Apeksha, K., Patel, V. K. J., (2013). Design and Dynamic Analysis of 70T Double Girder Electrical Overhead Crane, Journal of Information, Knowledge and Research in Mechanical Engineering, Vol 2, ISSN-975-668X.

[9] Nishant, S., Crane-Hook Shapes Optimization And Thermal Analysis Using Finite Element Too, International Journal Of Advanced And Innovation Research, ISSN: 2278-7844.

[10] Shaban, M., Mohamed, M. I., Abuelezz, A. E., Khalifa, T., (2013). Determination Of Stress Distribution in Crane Hook by Caustic, International Journal of Innovative Research in Science, Engineering and Technology, Vol. 2, Issue 5, ISSN: 2319-8753.