ISSN No:-2456-2165

Optimization of the Mini Backhoe Excavator's Arm by HyperWorks Under Static Condition

Yousof Khairalla[1] (Students)
[1]Mechanical Engineering Department,
Bursa Uludag University,
Bursa, 16059, Turkey

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

Abstract:- A backhoe excavator is a type of excavating equipment consisting of a digging bucket, boom, and arm. The arm and boom work together to manipulate the bucket and get everything into position. A backhoe excavator arm is a two-part articulated arm that can dig, load, trench, and use different attachments. It is powered by hydraulic pressure. The primary objective of this research paper is to determine the optimal geometry of the backhoe excavator's arm through topology optimization applied to a nonempty part. This approach differs from previous studies which focused on arm designs made from plates with a specific thickness. By utilizing topology optimization, we aim to achieve an optimal design that incorporates material distributions in 2D/3D space without compromising its strength, and an optimal mass-to-strength ratio will be achieved for the arm component while ensuring its safety under all operating conditions. To accomplish this, HyperWorks program will be used after creating a detailed arm model using SolidWorks program. After applying topology optimization to the arm, the next step is to re-design the arm based on the results obtained. Once the re-design model is complete, the next stage involves conducting a shape optimization analysis to determine the optimal thicknesses of all arm's components. This optimization aims to find the ideal thickness for each component, taking into consideration factors such as structural integrity, weight reduction, and performance requirements.

Keywords:- *Topology; Backhoe Excavator Arm; FEA; Hyper Works; Optimization.*

I. INTRODUCTION

Backhoes, also known as backhoe loaders, are characterized by their smaller size and enhanced maneuverability compared to excavators. They are equipped with a boom, stick, and bucket at the rear, complemented by a sizable loader at the front. This dual functionality is the reason they are commonly referred to as backhoe loaders. Figure 1.

Designers and engineers utilize various techniques and methodologies to optimize the arm design of excavators. One common approach is using Finite Element Analysis (FEA), which allows for a detailed analysis of the arm's structural behavior under different loads and forces. It helps identify areas of high stress and deformation, enabling designers to make informed decisions regarding material selection, reinforcement, and geometry modifications.

The reduction in weight not only benefits fuel efficiency and lifting capacity but also contributes to reducing the environmental impact of construction operations. Designers always face the challenge of creating a design that is both safe and efficient under all working conditions, while minimizing weight and cost. To achieve this, Finite Element Analysis has emerged as a powerful technique in the field of engineering. By utilizing FEA, designers can simulate the behavior of the backhoe parts and analyze how they respond to various loads and forces. This enables them to identify potential weak points, determine the structural integrity of the parts, and make necessary modifications to ensure a safe and reliable design.

The use of FEA in the design of backhoes offers several advantages. It allows for a more accurate and detailed analysis compared to traditional methods, reducing the need for physical prototypes and costly testing. Additionally, it enables designers to explore different design iterations and evaluate their impact on the performance and safety of the backhoe parts. This iterative process helps in achieving an optimal design that meets the required strength criteria while minimizing weight and cost. Given the importance of the excavator arm and the significant forces it experiences, numerous studies have focused on optimizing its design to enhance its performance while reducing weight without compromising safety.

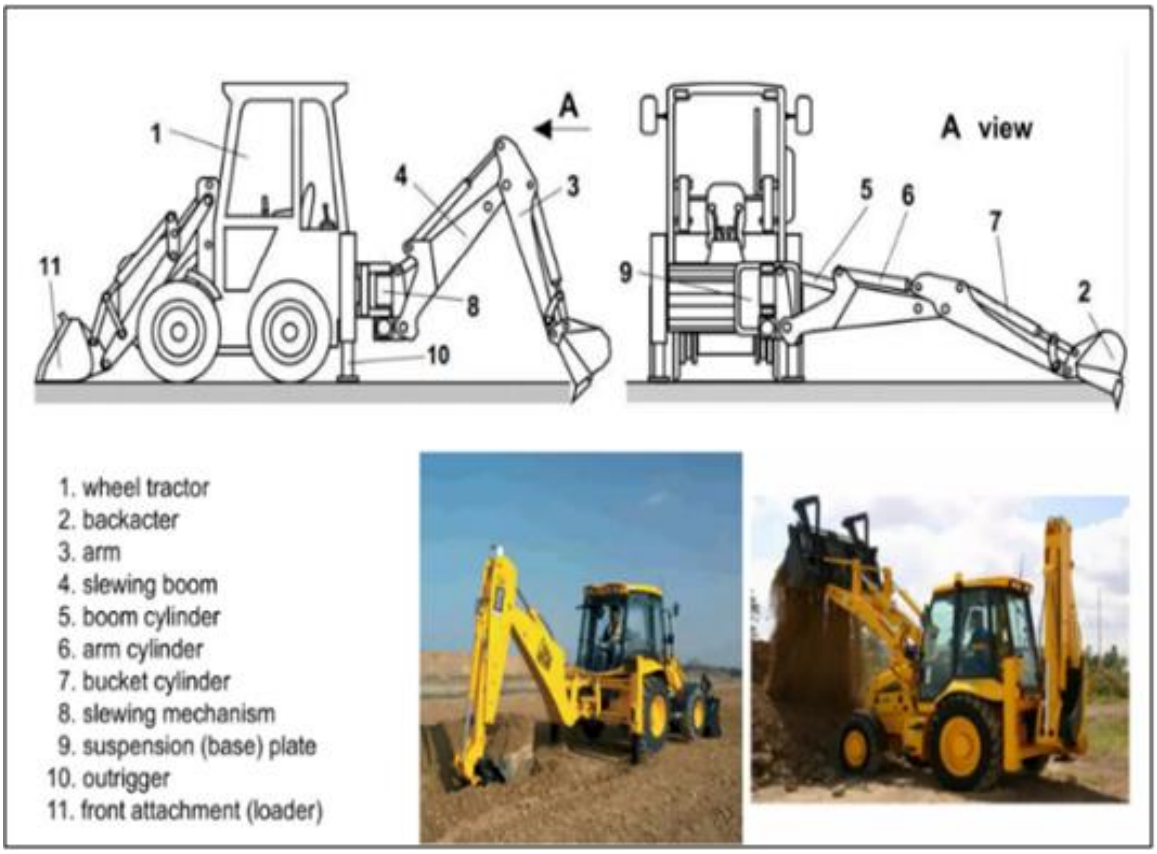

Fig 1 The Backhoe Excavator Components

II. LITERATURE REVIEW

Indeed, there is a significant body of literature dedicated to the finite element analysis (FEA) of moving machines, including backhoe-loaders, excavators, and bucket wheel excavators. These studies aim to analyze and optimize the structural performance, durability, and reliability of these machines under various operating conditions.

Bansode, S., Deore, V., and Sahu, D. [1] has successfully achieved an overall weight reduction of approximately 5% in the design. A detailed inspection has been carried out by FEA using the Ansys software to observe and examine the stress developed in the arm for all design considerations.

In a study conducted by Özer [2], the strength analysis of boom-stick groups with different digging reaches was examined using the FEA method.

In a study conducted by Ramesh, G., Krishnareddy, V. N., & Ratnareddy, T. [3] a weight reduction for the Lower Arm design was successfully achieved through the implementation of OptiStruct software. A noticeable weight reduction of 9.28% was realized when comparing the results to both the initial design and the newly proposed design model. This highlights the effectiveness of OptiStruct in optimizing the Lower Arm's structural configuration for enhanced efficiency and reduced weight."

Swapnil S. Nishane and et.al in their paper analyzed and optimized the design of a backhoe excavator bucket. They found that the optimized bucket had reduced stress areas and improved performance compared to the existing bucket. [4].

Ahmet Erklig and et.al in their work succeed in improving the backhoe loader arms by analysis it with Ansys, for back arm the safety factor increased from 1.59 to 1.98 and the strength up to 24.5% [5].

Bhaveshkumar P. Patel and et.al in their paper The FEA method used to optimize the weight of the backhoe excavator parts including arm and the reduction of all parts weight reached 27.91%. by applying shape optimization. [6].

Rao, D, Sekhar and et.al [7] work on different models for the arm to get by FEM (finite element method) the optimized weight of the structure with the acceptable stress limit.

III. ASSUMPTIONS

➢ *Those are Some Assumptions made During the Analysis.*

- Linear elastic material behavior means that the stress is directly proportional to strain within the elastic limit.
- Rigid pins and links simplify the analysis by considering them as perfectly rigid connections. Static and symmetric loading assumes that the loads applied to the arm are constant and applied in a stable manner.
- Material properties remain unchanged after welding.

IV. METHODOLOGY

This work involves creating a CAD model of a backhoe excavator arm using SolidWorks software. Figure 2. This CAD model is based on the geometrical dimensions of the arm of mini hydraulic backhoe excavator [8].

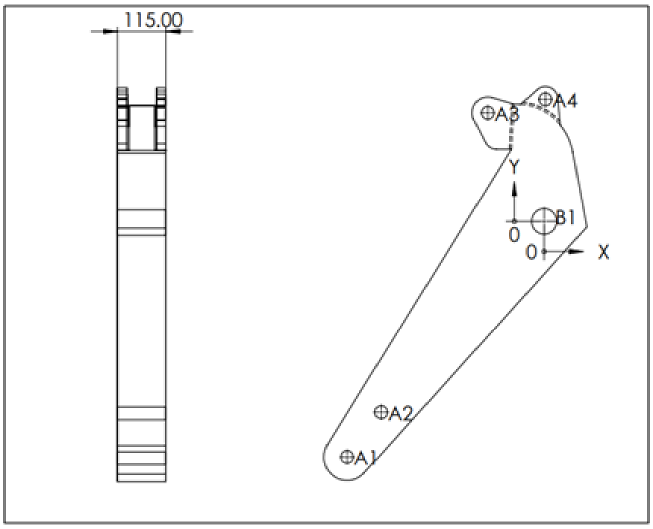

Fig 2 The Model of the Arm

This specific location experiences maximum breakout force [9]. In the design of the backhoe excavator arm, it is first assumed that the arm is solid and does not contain any hollow spaces inside. However, in the areas where the hydraulic cylinders are joined with the bucket and boom, the sides are represented as sheet metal with a certain thickness. This allows for the proper attachment and structural integrity of these components. Once the CAD model of the backhoe excavator arm is completed it is exported to HyperWorks program by converting it to IGES format for topology optimization. This involves meshing with 712,493 elements, modeling the arm determining the non-design areas that do not allow changes colored with blue and design one which will affected by topology optimization.

In Figure 3 shows the mesh that applied to apart of the arm with fine size of 3mm, to produce a detailed material distribution and get more accurate and detailed results.

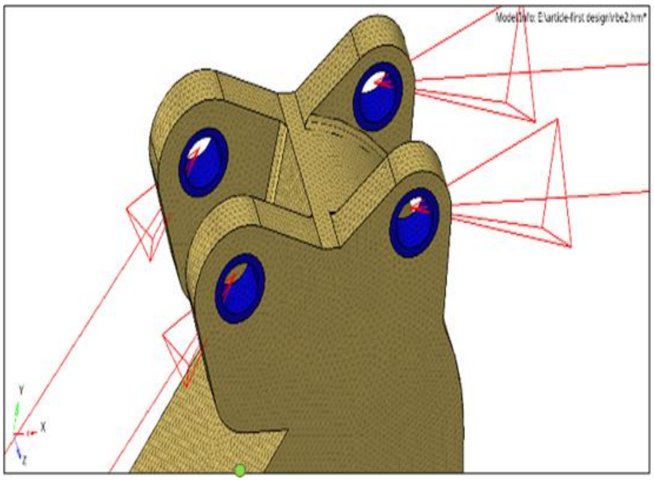

Fig 3 Meshing the Body of the Arm

Afterwards, the boundary conditions and loading for the excavator arm applied as in published paper [6]. in Figure 2, the excavator bucket is connected to the arm at positions A1 and A2. At position A1, the connection exerts a force of 22,423 N, while at position A2, the force is 7,783 N. The bucket is also connected at position A3 through two lug bushes of the arm. The bucket cylinder exerts a force of 11,203 N on each lug bush. Furthermore, the boom is connected at position A4 through two lug bushes of the arm. The arm cylinder exerts a force of 22,098 N on each lug bush. Additionally, at position B1, the arm is connected to the boom Table1.

Table 1 Boundary Conditions for the Arm

Forces on Joint of the Arm (KN)	Horizontal (X) Component	Vertical (Y) Component
A1	-15.97	-15.74
A2	-5.358	5.646
A3*	6.632	9.0285
A4*	-22.098	0
B1	0	0

* This Force Applied on the Two Sides for this Cylindrical Joint

In this configuration, the boom acts as a cylindrical support for the arm, and the displacement is considered to be zero at this connection point. Figure 4.

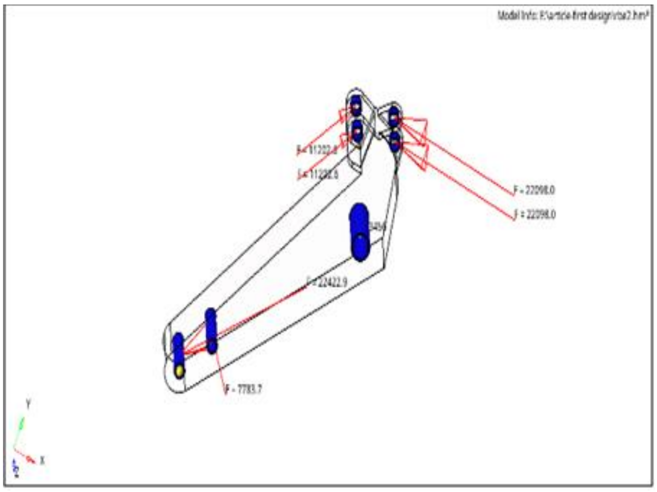

Fig 4 Applying the Boundary Conditions on the Arm

These boundary conditions define the constraints and forces acting on the arm at that position. After that, we define the load steps which define the study type as a linear static state analysis.

The next step is to define material properties, Table 2 explains the properties of applied material.

Table 2 Mechanical Properties for Applied Materials

Mechanical Properties	Ductile iron	Alloy steel
Elastic modulus (Mpa)	120000	210000
Poisson's Ratio	0.31	0.28
Density (Kg/m3)	7100	7700
Yield Strength (Mpa)	551	620
Failure Criterion (Mpa)	275.5	310

ISSN No:-2456-2165

Different modifications in this paper are justified based on the value of safety factor which has been considered as N = 2.

The stress in the components should not exceed the limit of failure criterion, as exceeding this limit can lead to structural failure.

Additionally, we need to consider the maximum displacement, which should be smaller than the thickness of the thinnest sheet. By minimizing mass, we ensure that the component remains stable and doesn't experience excessive movement. The next step in the optimization process is to perform topology optimization. During topology optimization, the software algorithm analyzes the loads, constraints, and design space to iteratively remove or redistribute material in areas where it is not necessary, while retaining material in regions where it is crucial for structural support. This process results in a generalized structure that achieves the desired balance between mass reduction and strength requirements. The outcomes of the topology optimization, as illustrated in Figure 5, highlight the most crucial regions by a gradient from blue to red, with the red areas indicating non-design components which are cylindrical attachments.

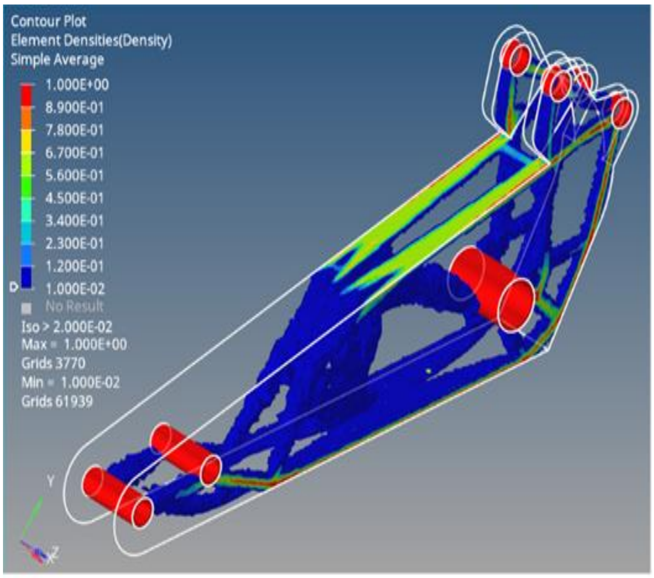

Fig. 5 Topology Optimization results

It is noteworthy that the arm has the potential to be fabricated using either sheet metal or profiles. In this stage, no significant differences are observed based on the applied materials after applying two different material properties.

After obtaining the mass density plot through the topology study, a new model is designed in SolidWorks based on the results. Dividing it into various components to change their thicknesses one by one by HyperStudy.

In Figure 6 a new CAD model designed depending on topology Optimization results structured from different component, everyone represented as 2-D surface and different color.

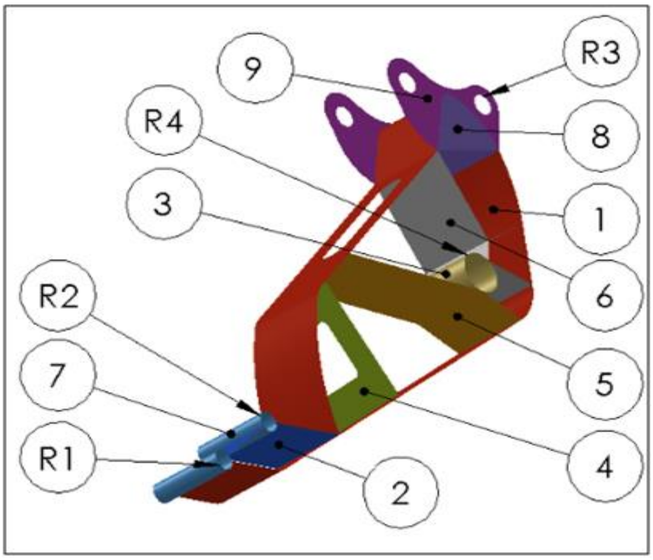

Fig 6 the New Model with Various Components

Through next shape optimization process, the goal is to achieve the best possible thickness for each component and get the best diameters for cylindrical attachment, while considering factors such as structural integrity, weight reduction, and overall performance. Various iterations are performed to finetune the design and find the optimal combination of thickness and mass. Once the optimized design is finalized, the weight of the modified arm is calculated.

➢ *To Optimize the New CAD Structure using Hyper Works, these General Steps are followed:*

- Define the optimization objectives as-Minimizing mass as possible.
- Create a finite element model: HyperMesh Used to create a detailed finite element model. This involves meshing all different components with a total of 48234 elements and 3mm mesh size 2D figure 7**.**

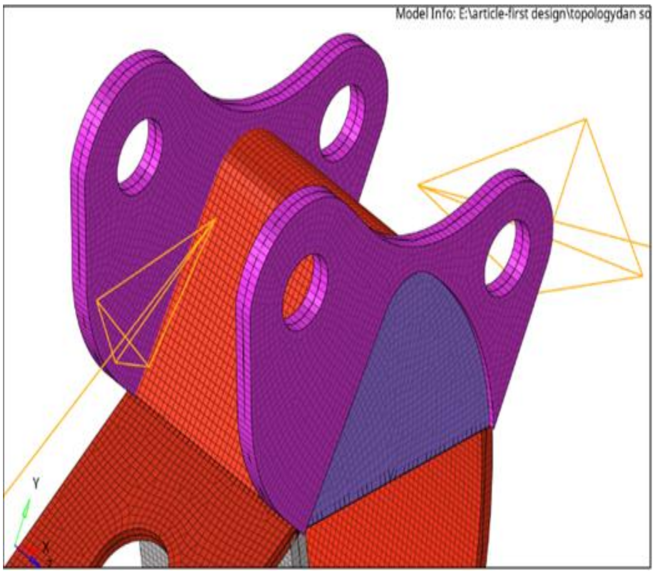

Fig 7 Meshing and Giving Initial Thickness for all Components

- Define design variables: the components thicknesses and radius of cylindrical attachments
- Give the Initial thickness for all components.
- Applying the same boundary conditions.
- Set up optimization constraints: These constraints are:
- Von mises stresses less than Failure criterion
- Max displacement as 1mm.

V. RESULTS AND DISCUSSIONS

The shape optimization process provides optimal thickness values for all variable inputs. Starting from the initial thickness values, the program systematically adjusts thickness within a specified range. For each component, the focus is on the function between its thickness and weight, ensuring stress limits and displacement are maintained. An alternative material is then applied to identify which component exhibits the most significant change in response to altered material properties. The stress and displacement results for alloy steel are illustrated in figures 8 and 9. demonstrating compliance with specified limits.

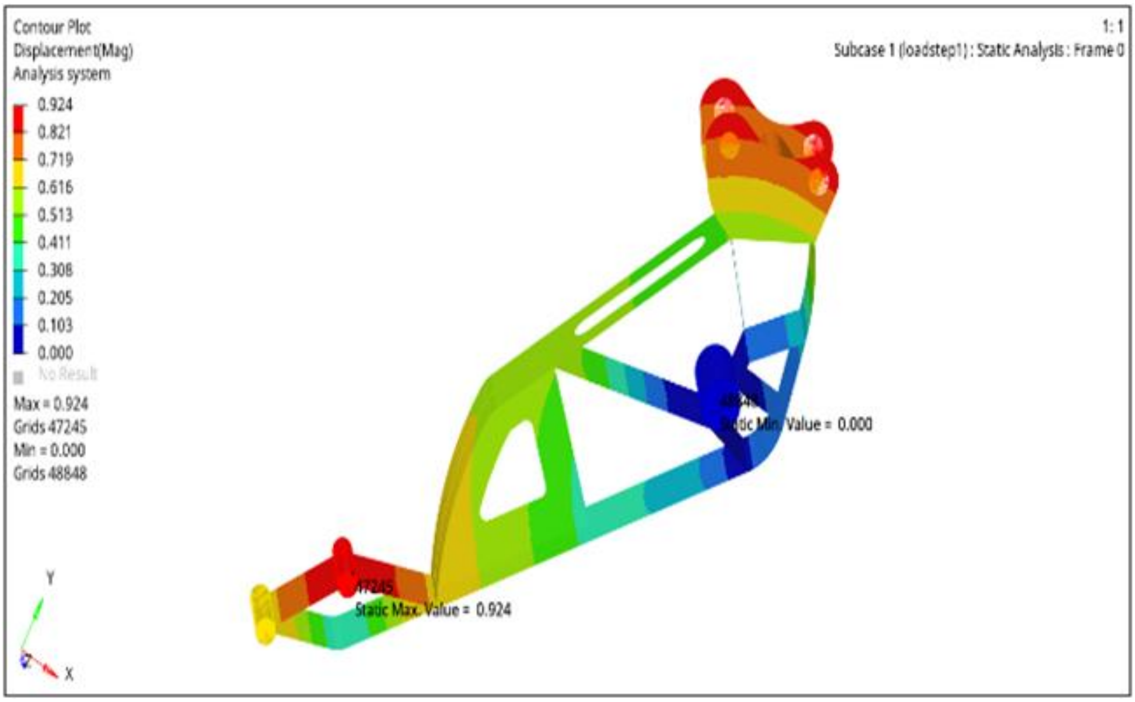

Fig 8 Max Displacement for the Arm is 0.9mm

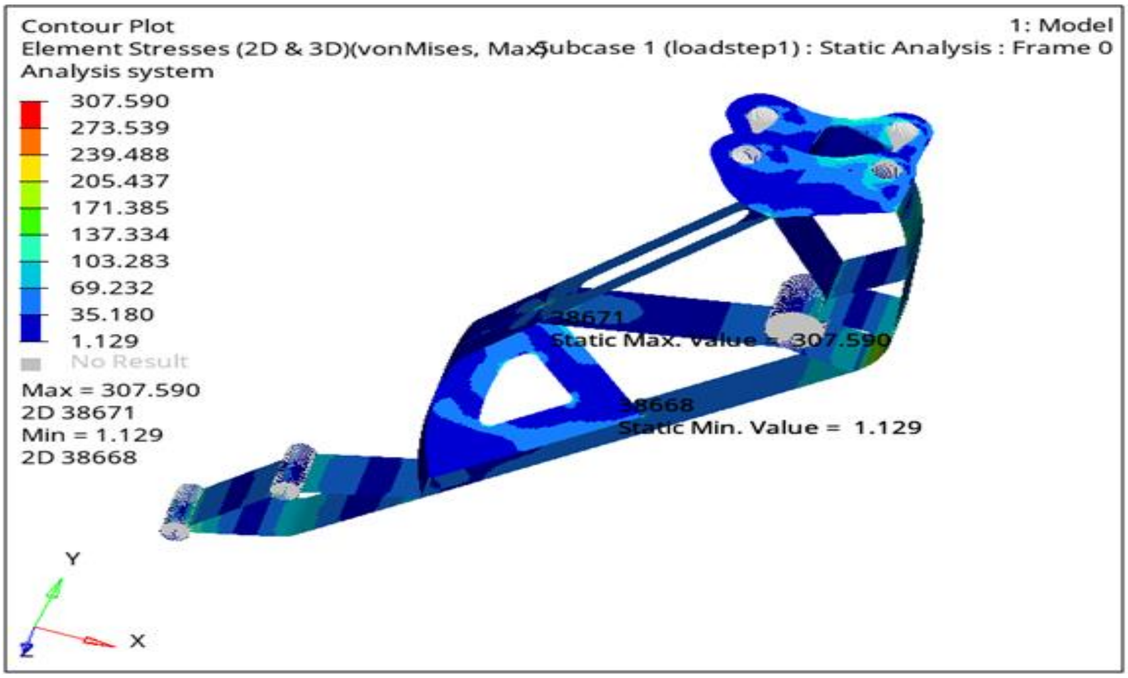

Fig. 9 Von Mises Stress for the Arm

To maintain structural integrity, the weight under these conditions is reduced to 20.7 kilograms, aligning with findings in article [10], which achieved a weight reduction to 20.9 kilograms.

It is noteworthy that while our results are comparable, this study and design exhibit a larger displacement but stay safe. The modification in the material composition resulted in changes to several components, namely T1, T2, T3, T4, T6, and R4. These modifications were made with the objective of decreasing the mass as small as possible, causing a decrease to 20.7 kg for both the ductile iron and alloy steel material structures. Additionally, the displacement was successfully controlled within the range of 0.9-1.1 mm, and the stress levels were kept below the failure criterion. This ensured a safety factor of 2, which was determined earlier in this analysis. These modifications demonstrate the successful optimization of the material composition for the desired performance and safety requirements. it was determined that only R3 needed to be modified.

The new requirement for R3 is that its diameter should be increased by 10 mm to reach a diameter of 40 mm. No changes are necessary for the other cylindrical components (R1, R2). This adjustment ensures that the design meets the specified dimensional requirements and allows for the desired performance and functionality without affecting the other components of the structure. Based on the provided information, it is noted that the highest stress and displacement occur in specific areas of the arm model.

Table 3 The Optimization Results

Component ID	**Initial Thickness**	**Range (mm)**	**Ductile Iron**	**Alloy Steel**
1.T.1	6.8	5 to 10	6.75	6
2.T.1	6	5 to 10	8.4	8.25
3.T.1	8	8 to 12	11.8	12
4.T.1	4.1	3 to 7	4.8	4
5.T.1	4.1	3 to 7	3	3
6.T.1	9.75	7 to 12	11.9	9.5
7.T.1	5.25	5 to 7.5	5	5
8.T.1	4.1	3 to 6	3	3
9.T.2	10	8 to 12	8	8
R1	15	12.5 to 17.5	15	15
R2	15	12.5 to 17.5	12.5	12.5
R3	15	10 to 20	20	20
R4	30	25 to 35	26	25
Von Mises (MPa)			277	307.6
Max Displacement (mm)			1.08	0.92
Mass (Kg)			20.71	20.79

VI. CONCLUSION

The research paper presents the results obtained from the optimization process using HyperWorks. The findings include the optimal design parameters for the mini backhoe excavator's arm, such as the dimensions and material selection. The results also include the evaluation of key performance metrics, such as stress distribution, displacement, and weight, for the optimized design.

The research demonstrates the effectiveness of using HyperWorks for optimizing the design of a mini backhoe excavator's arm under static conditions. The optimized design parameters obtained through this research can significantly enhance the arm's structural integrity, operational efficiency, and overall performance. The findings of this study can serve as a valuable reference for engineers and designers involved in the development of mini backhoe excavators and similar mechanical systems.

A structure with wight of 20.7 and total safe factory of 2 for ductile iron and alloy steel is obtained for backhoe excavator arm.

REFERENCES

[1]. Bansode, Sheela & Deore, Vishakha & Sahu, Dr.Anil. (2020). Design, Modelling and FEM Analysis of Excavator Arm. IARJSET. 8. 236-241. 10.17148/IARJSET.2021.8642.
[2]. ÖZER, S. (2007). Kazıcı-Yükleyicilerde Kazma Mesafesine Bağlı Olarak Bom-Stik Grubunun Tasarımı (Publication No. 215882) [Mersin ÜNiversitesi].
[3]. Ramesh, G., V. N. Krishnareddy, and T. Ratnareddy. "Design and optimization of excavator." International Journal of Recent Trends in Enginee-ring & Research 32.4 (2017): 535-549.
[4]. Swapnil, S. Nishane. "Modeling and static analysis of backhoe excavator bucket." International journal of research in advent technology 4.3 (2016): 103-108.
[5]. Erklig, Ahmet, and Eyüp Yeter. "The Improvements of the backhoe-loader arms." Modeling and numerical simulation of material science 2013 (2013).
[6]. Patel, Bhaveshkumar P., and Jagdish M. Prajapati. "Structural optimization of mini hydraulic backhoe excavator attachment using FEA approach." Machine design 5.1 (2013): 43-56.

[7]. Rao, D & Sekhar, Chandra & Chakilam, Shashikanth & Bharathi, V. (2016). Design And Optimization of Excavator Arm.

[8]. Patel, Bhaveshkumar P., and J. M. Prajapati. "Kinematics of mini hydraulic backhoe excavator–part II." International Journal of Mechanisms and Robotic Systems 1.4 (2013): 261-282.

[9]. Patel, Bhaveshkumar P., and Jagdish M. Prajapati. "Evaluation of bucket capacity, digging force calculations and static force analysis of mini hydraulic backhoe excavator." machine design 4.1 (2012): 59-66.

[10]. Khan, Fahim Mahmud, Md Shahriar Islam, and Md Zahid Hossain. "Design aspects of an excavator arm." International Review of Mechanical Engineering 10.6 (2016): 437-442.