Kütahya Dumlupınar University
Institute of Graduate Studies

Journal of Scientific
Reports-A
E-ISSN: 2687-6167
Number 50, September 2022

RESEARCH ARTICLE

DESIGN OPTIMIZATION of a BRACKET PLATE for an AMMUNITION FEED MECHANISM of a MEDIUM CALIBER CANNON

Cihan TURAN [1,2*], Hacı Abdullah TAŞDEMİR[3]

[1]Istanbul Technical University, Mechanical Engineering, Istanbul,
[2]Aselsan Inc., Ankara, turanc21@itu.edu.tr , ORCID: 0000-0003-2849-8368
[3]Istanbul Technical University, Mechanical Engineering, Istanbul, tasdemirh@itu.edu.tr,
ORCID: 0000-0002-2836-5488

Receive Date:07.07.2022 *Accepted Date: 29.07.2022*

ABSTRACT

Topology optimization has been one of the major concerns for mechanical engineers over the years. With increasing utilization of the finite element method, mechanical analyses can be done easily these days and their results are quite reliable. In weapon systems, high loads act on system components. Due to high loads, every component must be designed to operate without any failure. While designing them, attention must be given in order to avoid excessive weights. So, topology optimization is needed in weapon system components. In this study, design with topology optimization of a bracket plate of an ammunition feed mechanism were investigated using the finite element method. By utilizing topology optimization concept, the dimensions, material and the number of mounting holes of the bracket plate of an ammunition feed mechanism were changed to see their effects on the elemental Von-Mises stress and nodal displacement values. The results show that the increase in mounting hole number and the thickness of the material with selecting a material having higher strength properties decreases the elemental Von-Mises stress and nodal displacement values. According to the results, a safer bracket plate for an ammunition feed mechanism was designed to operate in the given working conditions without any failures.

Keywords: *Ammunition feed mechanism, Weapon turret, Topology optimization, Finite element method, Design*

1. INTRODUCTION

In the life of human beings, some tools have always been used for hunting, defensing and building. For defensing purposes, some primitive items such as animal teeth and wooden sticks were used as weapons [1]. Different from the past, today, weapon systems are complex systems and contain various auxiliary components such as ammunition feed mechanisms, weapon turrets, pneumatic and hydraulic systems, etc. [2]. In terms of loads, working conditions of ammunition feed mechanisms are quite tough. Also, depending on the types of weapons, various ammunition feed mechanisms can be designed and used [3].

The finite element method is extensively used for structural analyses. In order to simulate physical phenomena, the finite element method gives quite good results [4]. These results are close to physical testing results generally. Especially for complicated systems that analytical solutions are impossible, one can utilize numerical method called finite element method [4]. In some cases, analytical solutions are not even close to real results [5]. By specifying the inputs and goal of the problem clearly, more reliable and high-quality results can be obtained [6]. The finite element method can be utilized for almost every case such as linear, nonlinear, transient and steady cases [7]. Also, many problems such as fluid flow and thermal, can be solved by using the finite element method [8].

In addition, topology optimization is used commonly in mechanics. Topology optimization is a process that determines the shape, location and number of features such as holes and ribs to make parts more reliable [9]. Specifically, for components carrying high loads, topology optimization is quite beneficial. Results of classical topology optimization methods are not suitable for classical machining processes generally [10]. Therefore, in this study, classical topology optimization methods were not used because solid geometry was required and the plate was intended to be produced by conventional machining processes like CNC milling. In fact, if one has limited design experience, manual topology optimization can be troublesome. For that applications commercial software can be more beneficial [11].

In this study, due to high loads acting on it, a bracket plate which connects an ammunition feed mechanism to a weapon turret wall was designed and topologically optimized manually by utilizing the finite element method. The design and optimization processes were done by using Siemens NX and NX Nastran software, respectively.

2. SYSTEM CHARACTERISTICS AND THEORETICAL MODEL

The bracket plate, which is colored as yellow in the figure 1, is used for assembling the ammunition feed mechanism to the weapon turret wall. Figure 1 shows the bracket plate, ammunition feed mechanism and weapon turret wall. Also, the plate has the dimensions of 500 mm width and 220 mm height.

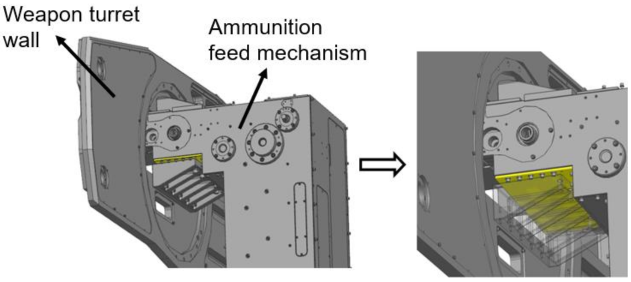

Figure 1. General view of the assembly.

In order to achieve optimization and make the bracket plate safer, various changes were made such as changing the thickness, material, hole numbers and hole locations on it. According to the changes, a new analysis was run and the part was topologically optimized. Two iterations were conducted in total and they were explained in this paper.

The bracket plate is mainly responsible from mounting of the ammunition feed mechanism to the weapon turret wall by using bolts. Due to the assembly, various forces act on the bracket plate. These forces can be classified into 2 cases since they act on the bracket plate in separate situations. In the first case, only the operational forces of the weapon turret act on the bracket plate; however, in the second case, only the transportation loads act on the plate. Tables 1 and 2 show the loads of the two cases.

Table 1. Loading case-1.

Rotational torque of the elevation	77.5 Nm (along +y axis)	Acts on the COG of the mechanism
Centrifugal force of the elevation	40 N (on +z axis)	Acts on the COG of the mechanism
Rotational torque of the ring gear	246.8 Nm (along +z axis)	Acts on the COG of the mechanism
Centrifugal force of the ring gear	131.5 N (on -y axis)	Acts on the COG of the mechanism
Mass of the feed mechanism	400 kg (on +z axis)	Acts on the COG of the mechanism

Table 2. Loading case-2.

Transportation load	1.4 x mg (on the 3 axes simultaneously)	Acts on the COG of the mechanism

3. ANALYSES

3.1. First Iteration

For the initial iteration of the optimization, a rough geometry with 7050-T7451 Aluminum material was designed. An initial thickness of 10 mm with 6 side holes were given to the bracket plate. The diameter of the side holes is 6.6 mm and they are used for mounting the bracket plate to the ammunition feed mechanism. The side holes can be seen in red balloons in the figure 3. In addition, table 3 shows some mechanical properties of 7050-T7451 Aluminum. Then, finite element method was applied.

Table 3. Some mechanical properties of 7050-T7451 Aluminum.

Ultimate Tensile Strength	524 MPa
Tensile Yield Strength	469 MPa
Elongation at Break	11 %
Modulus of Elasticity	71.7 GPa

3D tetrahedral CTETRA (10) mesh with 5 mm element size was created. Then, the forces, mass and torques were applied to the center of gravity (center of gravity = COG) of the feed mechanism. The COG of the feed mechanism was connected to the bracket plate by a deformable connection element, namely RBE3. RBE3 element is generally used for transmission of distant loads. Also, the components between the bracket plate and the COG of the feed mechanism behave as deformable. Figure 2 shows the given mesh and COG of the feed mechanism, where the loads act. Also, 6 fixed holes with 10.3 mm diameter can be seen as orange-colored in the figure 2. These holes fixate the plate to the weapon turret wall bracket; so, they were modelled as fixed.

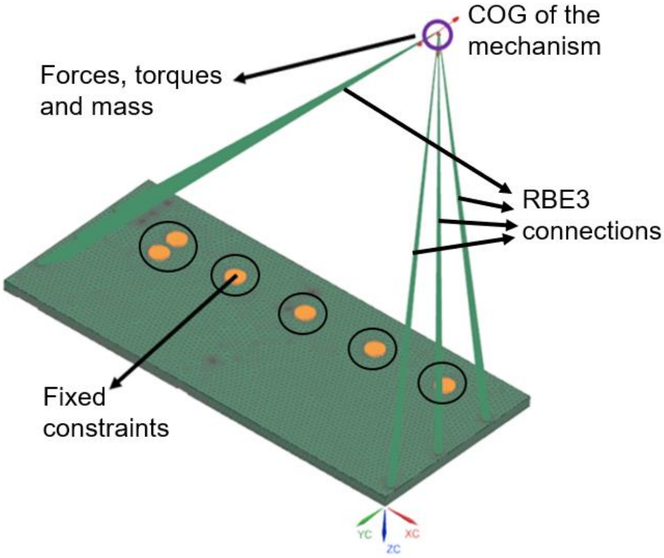

Figure 2. The mesh, loads, fixed constraints, RBE3 connections and COG of the mechanism for the first iteration.

Moreover, since this bracket plate is assembled to the feed mechanism with 6 bolts at the side holes, 1D rigid connection element, namely RBE2 was given to that 6 side holes as in the following figure 3. RBE2 connection element is a rigid element. Since the bolts are much stiffer than the bracket plate, the holes of the bolts were modelled as rigid. This yields more conservative results than the common hole modelling methods. In weapon systems, staying at the safer side is better.

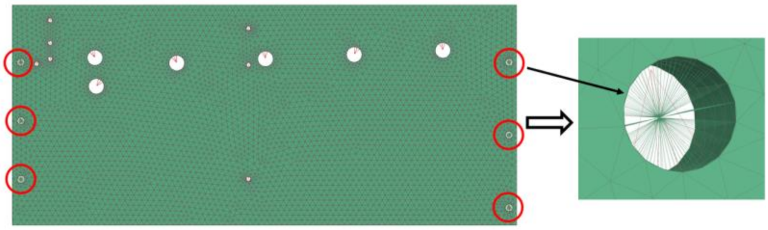

Figure 3. RBE2 connections for the bolts.

After that, loading cases were applied.

3.1.1. Loading case-1 of the first iteration

In the first run, loading case-1 was applied which was stated in the table 1. The rotational torque of the elevation, the centrifugal force of the elevation, the rotational torque of the ring gear, the centrifugal force of the ring gear and the mass of the feed mechanism were given to the COG of the feed mechanism. For the boundary conditions, fixed constraints were given to the 6 mounting holes on which the plate is mounted to the turret wall. Figure 2 shows the fixed constraints as orange-colored holes. Table 1 and figure 2 show the magnitudes of the forces, torques, mass and the graphical representation of them, respectively. After completing that step, analysis was done and the results were given in the "Results" section. According to the results, there was no need to apply the loading case-2 for the first iteration as explained in the "Results" section.

3.2. Final Iteration

For the final iteration, the loading case-1 and 2 were applied separately, and 5 fixed mounting holes were used for the boundary conditions. Also, the number of side holes for mounting the bracket plate to the feed mechanism was 9 in this iteration, so the stress was wanted to be distributed around the side holes. Similarly, COG of the mechanism was connected to the side holes with RBE3 connections and the bolts at the side holes were modelled with RBE2 connections as in the first iteration. Figure 4 shows the modelled structure. Also, the material was Hardox 450 tool steel with 14 mm thickness in this iteration. Table 4 shows some mechanical properties of Hardox 450 steel.

Table 4. Some mechanical properties of Hardox 450 tool steel.

Ultimate Tensile Strength	1400 MPa
Tensile Yield Strength	1200 MPa
Elongation at Break	10 %
Modulus of Elasticity	200 GPa

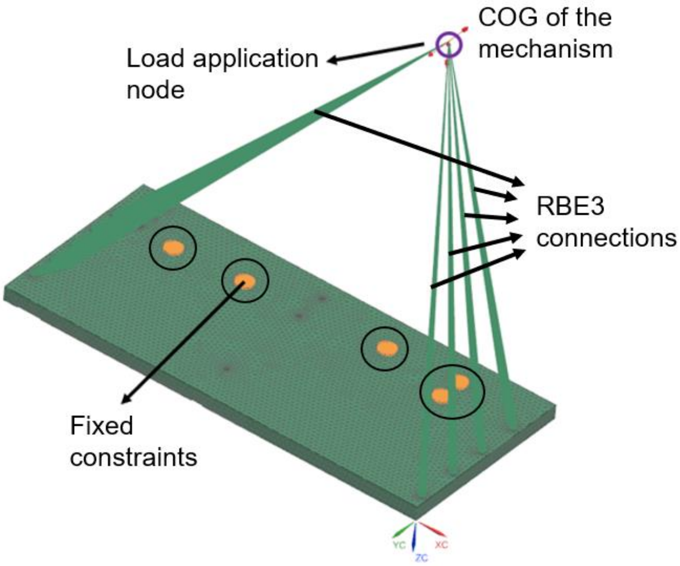

Figure 4. The mesh, loads, fixed constraints, RBE3 connections and COG of the mechanism for the final iteration.

Similarly, 3D tetrahedral CTETRA (10) mesh with 5 mm element size was created and the loads of loading case-1 and 2 were applied to the COG of the mechanism.

3.2.1. Loading case-1 of the final iteration

After completing the preparation steps, the loads of loading case-1, which are the rotational torque of the elevation, the centrifugal force of the elevation, the rotational torque of the ring gear, the centrifugal force of the ring gear and the mass of the feed mechanism, were given to the COG of the mechanism similar to the first iteration. Then, the analysis was done and the results were given in the "Results" section.

3.2.2. Loading case-2 of the final iteration

Different from the loading case-1 of this iteration, this time, the loads of loading case-2, which are the transportation loads, were given to the COG of the mechanism. After that, the analysis was done and the results were given in the "Results" section.

4. RESULTS

4.1. First Iteration

4.1.1. Loading case-1 of the first iteration

As stated previously, only the loading case-1 was applied for the first iteration. Also, fixed boundary conditions were applied as shown in the figure 2. The elemental Von-Mises stress and nodal displacement values were adequate to examine in this paper. The following figures 5 and 6 show the elemental Von-Mises stress and displacement values, respectively.

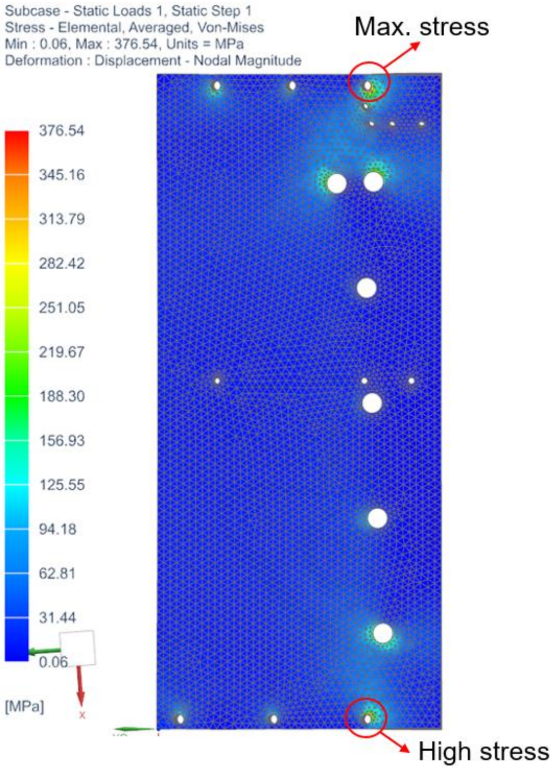

Figure 5. Elemental Von-Mises stress values of the loading case-1 of the first iteration.

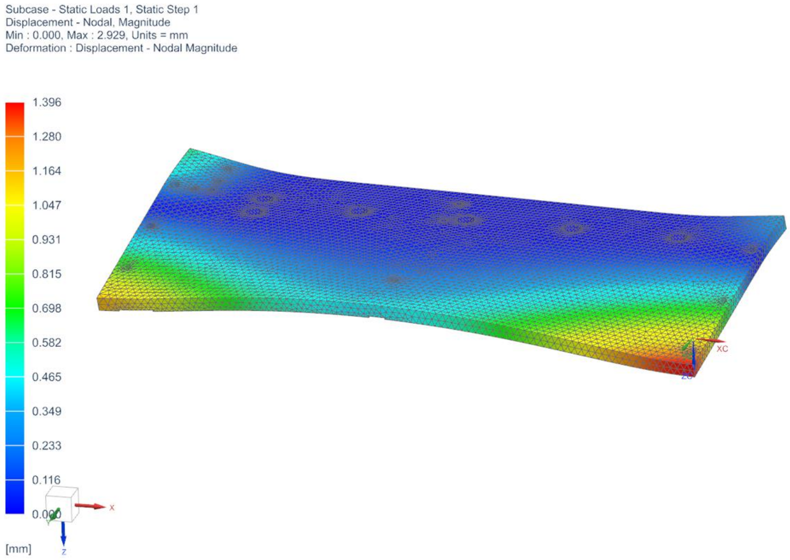

Figure 6. Nodal displacements of the loading case-1 of the first iteration.

As can be seen from the figures 5 and 6, the maximum elemental Von-Mises stress and nodal displacement values are 376.54 MPa and 1.396 mm, respectively. These values are quite high for the easier situation which is the loading case-1. Also, we can understand from the figure 5 that the stresses are concentrated near the side holes as expected.

Since the loading case-2 is much tougher than the loading case-1, there is no need to apply the loading case-2. The plate is nearly safe for the loading case-1 and it is apparent that it cannot withstand the loading case-2. So, optimization was needed and a new iteration was done.

4.2. Final Iteration

As stated previously, for the final iteration, the loading case-1 and 2 were applied separately and fixed boundary conditions were applied as in the figure 4.

4.2.1. Loading case-1 of the final iteration

Figures 7 and 8 show the elemental Von-Mises stress and displacement values for the loading case-1, respectively.

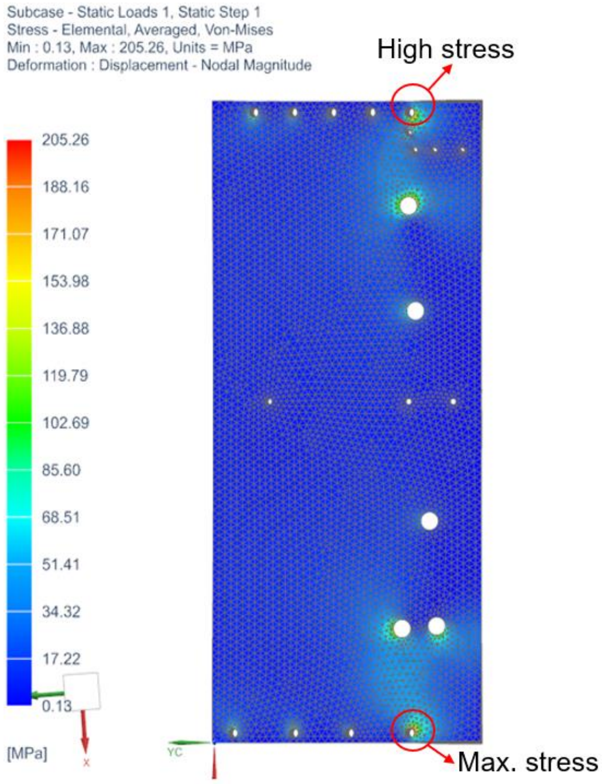

Figure 7. Elemental Von-Mises stress values of the loading case-1 of the final iteration.

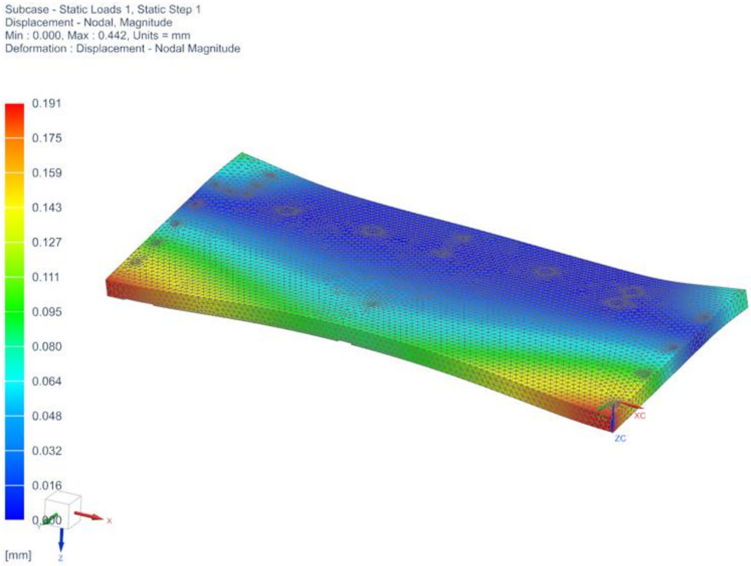

Figure 8. Nodal displacements of the loading case-1 of the final iteration.

As can be seen from the figures 7 and 8, the maximum elemental Von-Mises stress and nodal displacement values are 205.26 MPa and 0.191 mm, respectively. These values are quite acceptable since the tensile yield strength of Hardox 450 tool steel, which is 1200 MPa, is much higher than 205.26 MPa, and the maximum nodal displacement value is quite low.

4.2.2. Loading case-2 of the final iteration

According to the stress and displacement values above, the bracket plate is adequately safe in loading case-1. So, the loading case-2 can be applied now, and the elemental Von-Mises stress and displacement values can be seen for the loading case-2 in the figures 9 and 10, respectively.

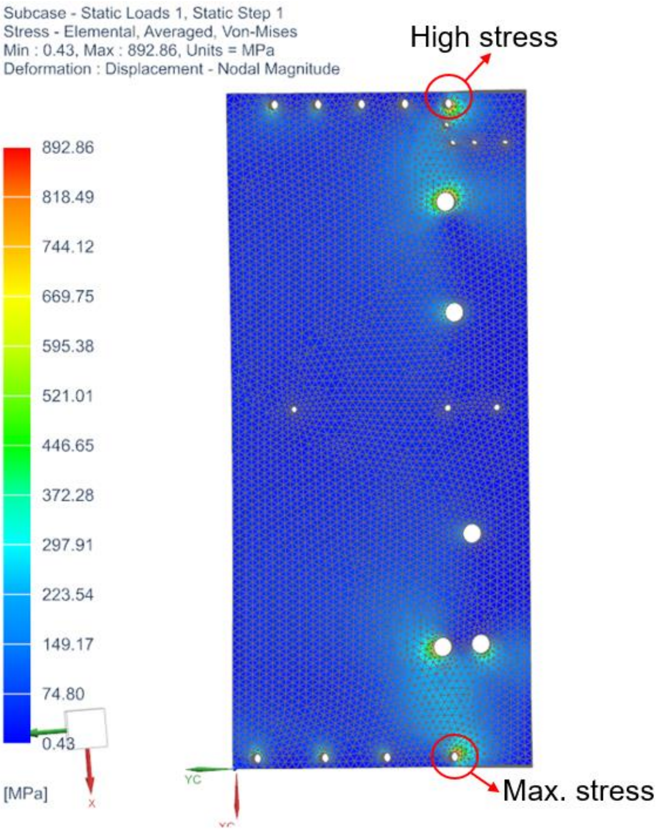

Figure 9. Elemental Von-Mises stress values of the loading case-2 of the final iteration.

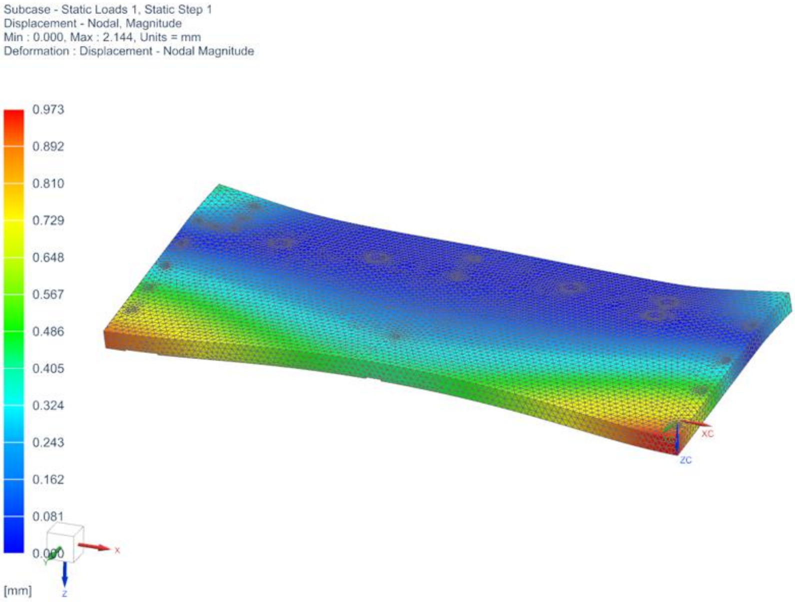

Figure 10. Nodal displacements of the loading case-2 of the final iteration.

As can be seen from the figures 9 and 10, the maximum elemental Von-Mises stress and nodal displacement values are 892.86 MPa and 0.973 mm, respectively. These values are quite acceptable again since the tensile yield strength of Hardox 450 tool steel, which is 1200 MPa, is much higher than 892.86 MPa, and the maximum nodal displacement value is quite low.

Table 5 summarizes the results clearly.

Table 5. Comparison of the results for the two iterations.

Iterations	**Maximum Elemental Von-Mises Stress Value [MPa]**	**Maximum Nodal Displacement Value [mm]**
First Iteration – Loading Case 1	376.54	1.396
Final Iteration – Loading Case 1	205.26	0.191
Final Iteration – Loading Case 2	892.86	0.973

As can be seen from the table 5, for the loading case-1, approximately 45% decrease in the maximum elemental Von-Mises stress value was achieved. In addition, the maximum nodal displacement value was decreased approximately 86%. These were achieved by changing the before-mentioned design parameters.

5. DISCUSSION

In order to see the effects of loads on components during design stage, the finite element method is quite beneficial for engineers. By utilizing the finite element method, critical stress concentration regions can be seen and they are minimized by changing the design of the part. For that purpose, in this study, topology optimization procedure was followed by utilizing the finite element method, and Von-Mises stress and nodal displacement values were reduced to make the bracket plate safer. Also, manual topology optimization process was followed in this study different from the topology optimization processes that use classical methods. Due to the manual process, all iterations and design changes were done manually. For future work, different finite element models for bolts can be applied and their results can be compared. Also, if non-conventional manufacturing methods, such as additive manufacturing, are applicable, optimization results of commercial finite element analysis software can be directly used by applying post processing.

6. CONCLUSION

In this study, by using the finite element method, design of a bracket plate was done. According to the finite element method results, a bracket plate having Hardox 450 tool steel material with 14 mm thickness, 9 mounting side holes and 5 fixed constraints is mechanically safer than a bracket plate having 7050-T7451 Aluminum material with 10 mm thickness, 6 mounting side holes and 6 fixed constraints. In the first iteration, it was seen that the elemental Von-Mises stress values accumulated near the side holes. Increasing the side hole number with increased plate thickness and decreased fixed hole number decreased the elemental Von-Mises stress values significantly. In addition, the results of Hardox 450 tool steel are quite enough to be safe for the given working conditions. The tensile yield strength and the ultimate tensile strength values of Hardox 450 tool steel is adequately higher than the maximum elemental Von-Mises stress value obtained in the figure 9. So, the topology optimization process was achieved.

ACKNOWLEDGMENT

In this study, special thanks to Aselsan Inc. for continuous support.

REFERENCES

[1] Herbst, J., (2006), The History of Weapons (1st ed.), Minneapolis: Twenty-First Century Books, 5-7.

[2] Williams, G. A. and Gustin, E., (2004), Flying Guns of the Modern Era (1st ed.), Marlborough: The Crowood Press Ltd., 100-120.

[3] Williams, G. A., (2003), Rapid Fire: The Development of Automatic Cannon, Heavy Machine Guns and Their Ammunition for Armies, Navies and Air Forces (1st ed.), Marlborough: The Crowood Press Ltd., 50-60.

[4] Reddy, J. N., (2019), Introduction to the Finite Element Method (4th ed.), New York: McGraw Hill Education, 1-2.

[5] Rao, S. S., (2018), The Finite Element Method in Engineering (6th ed.), Oxford: Butterworth-Heinemann, 3.

[6] Babuska, I. and Strouboulis, T., (2001), The Finite Element Method and its Reliability (1st ed.), Oxford: Oxford University Press, 5.

[7] Dhatt, G., Touzot, G. and Lefrançois, E., (2012), Finite Element Method (1st ed.), London: ISTE Ltd., Hoboken, NJ: John Wiley and Sons, Inc., 1.

[8] Liu, G. R. and Quek, S. S., (2014), The Finite Element Method a Practical Course (2nd ed.), Oxford: Butterworth-Heinemann, 3.

[9] Bendsoe, M. P. and Sigmund, O., (2003), Topology Optimization Theory, Methods, and Applications (2nd ed.), Berlin: Springer-Verlag, 1.

[10] Zhou, M., Fleury, R., Shyy, Y. K., Thomas, H. and Brennan, J. M., (2002), Progress in topology optimization with manufacturing constraints, 9th AIAA/ISSMO Symposium on Multidisciplinary Analysis and Optimization, AIAA 2002-5614.

[11] Jikai, L. and Yongsheng, M., (2016), A survey of manufacturing oriented topology optimization methods, Advances in Engineering Software, 100, 161-175.