

Stress Analysis and Optimization Of Quick Stop Device Through Finite Element Analysis Using ANSYS

Nur Shahida Azzahra Mohd Zamri¹, and Amiril Sahab Abdul Sani^{1*}

¹Faculty of Manufacturing and Mechatronics Engineering Technology, Universiti Malaysia Pahang, 26600 Pahang, Malaysia.

ABSTRACT – The quick-stop device (QSD) are design for the cutting metal that use in the turning process that produce the chip root formation. This device concentrated on time separation that rapidly stops the cutting when obtain chip root. In the process of cutting chip formation that needed the high impact of force to obtain the high force. The existing QSD designed with large size and weight that to accommodate load more 1000N. The area that high stress maybe can be fatigue when get the force more than the ability of strength of QSD. In this paper an analysis the stress of the critical part of the QSD via explicit dynamic and static structural of QSD with finite element method using ANSYS. In this paper also to study in finding the optimal design using the topology optimization to get the minimize size, design and the weight of the quick-stop device. Then, the values of equivalent stress and total deformation that identify in dynamic and static analysis. From that, it will continue in topology optimization to remove the unnecessary part based on the contour part that the low stress.

ARTICLE HISTORY

Received: 30th Oct 2022

Revised: 27th Nov 2022

Accepted: 2nd Dec 2022

KEYWORDS

Quick-Stop Device

Critical Part

Topology Optimization

Stress Analysis

Explicit Dynamic

INTRODUCTION

The machining process is advanced in the cutting process and contributed to the cutting tool concerning its surfaces, cutting edges, and angles in improving the cutting metal for better cutting-type results. In the machining of cutting metal, the metal is removed to form chips by a plastic deformation process. This chip deformation can be affected by the heat generated by that temperature, and the force significantly contributes to the quality of the cutting process. The heat generation that happens at the deformation of the chip that effected the temperature from the high productivity of machining in the cutting process needs the use of high cutting velocity and feed rate. In producing the cutting metal in the high cutting velocity and the feed rate that can be obtained of the chip root after quickly stopping a cutting process after producing the chip root that using the quick-stop device [1].

This device is the quicker device of the chip formation that results from the force. From the forced reaction need to figure out a Quick-stop device (QSD) right now able to control the stress from the impact of various forces to produce the chip formation. From that, we needed to investigate the critical part that gets the force's high impact after the device does the chip-cutting process. The FEA is the way of simulating the loading condition on the design and determining the response to the design. The Finite Element Analysis (FEA) is also used to calculate the machine tool's static stiffness or dynamic characteristics like natural frequencies and mode shapes [2].

ANSYS software analyzes the stress of the QSD to find the critical part of the device with a high impact of the stress with finite element analysis. ANSYS also applies topology optimization to find the optimal design by determining the best geometries and minimizing the shape and the weight [3]. It helps to increase productivity and help design better and more innovative products in less time. The topology optimization that categories in structural optimization is an application of optimization whose purpose is to achieve the best material distribution for a structure's specified needs. [4]

EXISTING QUICK-STOP DEVICE

In metal cutting, a quick-stop device is often used to investigate the forming of the chip root [5]. More detail about a QSD that to produce the chip formation that the device relates the speed between the tool was cutting and rotating of the workpiece is reduced to zero, and the chip in contact with the workpiece. The material head of the cutting tool that is contained within the zones of chip and workpiece surface generation constitutes the "chip root" that should faithfully represent the geometrical and metallurgical. The QSD in the categories of the cutting process involves the removal of metal in the form of chips from the workpiece by the action of the cutting tool. The operation of the QSD at high acceleration is that QSD should move the device away from the workpiece when the disengagement between the agency and the workpiece ends the cutting. There will be no more deformation of the root chip.

Figure 1 shows the existing QSD that needed to validate the design to redesign to a new QSD to improve the functional design. Before redesigning the new design, it is apparent to know the main part of the component of the QSD, which

makes it easy to identify each part and know its function. This device that the components were made of medium carbon steel, which balances ductility, and strength, is strong in handling high force and has good wear resistance to facilitate its production process in the cutting metal [6]. The main QSD components are the primary body (1), secondary body (2), trigger holder (3), tool holder (5), insert (6), the spring (7), base holder spring (4), house spring (8), and the pneumatic (9) [6].

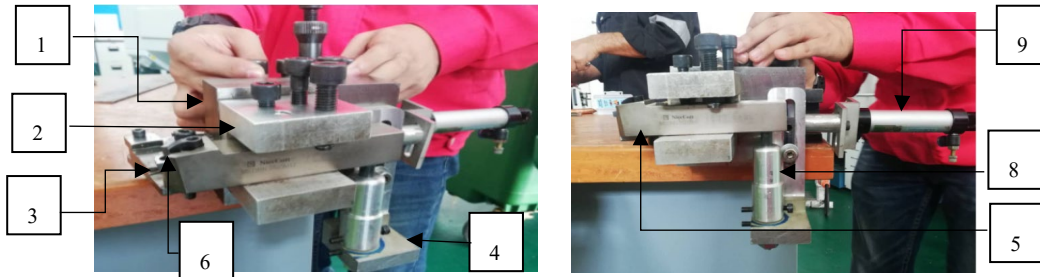


Figure 1. The existing of the Quick-Stop Device

Reaction To The Stress

By referring to Figure 2, it shows the part of the QSD with two forces of the tool pushed back. The top shows part (a) of the rotating workpieces that give force to the tool at the end of the tool that first touched the workpiece. Part (b) shows the spring area that have compressed of spring of the force. In this, the design is classified in the drop tool devices that employ the cutting force coupled with a spring force to retract the tool from the workpiece during cutting.

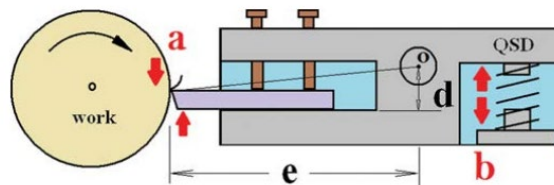


Figure 2. The schematic of the QSD mechanism cutting process at the area at parts (a) and (b).

The device’s reaction started when the device was activated by a compressed spring situated at the tool holder bottom and under the tool holder front that the trigger supports the cutting action. After activating, the trigger. the spring force exerted on the tool holder’s bottom makes it spin with the primary base removing the cutting tool from the cutting action.

FINITE ELEMENT ANALYSIS (FEA) USING ANSYS SOFTWARE

The FEA is a computational technique for estimating engineering solutions to boundary value problems. It can simply state that a boundary value problem is a mathematical problem in which one or more dependent variables must satisfy a differential equation everywhere within a known domain of independent variables and satisfy specific conditions on the boundary of the domain.

The explanation of basic FEA uses three processes in finite element analysis. Firstly, the pre-processor phase that finds the geometry problem and creates the meshing can hypothetically divide this object into smaller geometrical shapes. In the creation of the meshing, each shape is interconnected at two or more nodes, and the nodes are the joint points of elements, boundary lines, or planes. By being connected by nodes that make up the complete finite element mesh, each element type contains information on its degree-of-freedom set, material properties, and part of constraint. For the load condition that added load to explain the product loading scenarios such as machining forces in operation process that will push and pull effect to the certain part that get load.

Figure 3 shows the meshing selection in 0.0075m and the label (A, B, and C) based on the figure. It shows that A is the moment area, B is the part that gets the force (N) and C shows the blue area, which means it fixed support (boundary condition) selection that the fixed area that does not move and does not give any effected in this analysis the stress. Furthermore, it also shows the direction for the movement of the moment and the force will force.

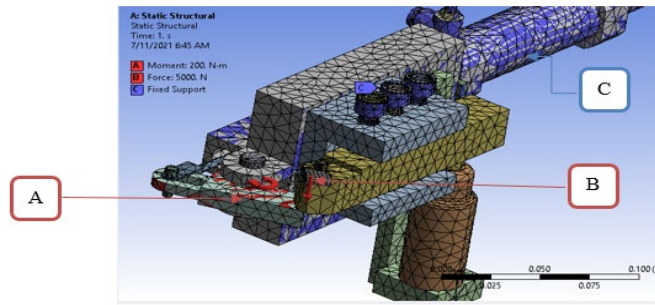


Figure 3. The direction of the part load applied and the boundary condition

Finally, boundary and load conditions are added to explain the design that constraint fully in the pre-processor. Next, solve the process that generated the simulation equation and the resulting value of degree freedom in the model. The final process is post processors that review the result, including the contour display of the stress level. Maximum stress is critical, and it has the chance of fatigue failure when it always gets higher stress at the same region [7]

Other than that, also finding dynamics that are the simulation solutions in engineering courses in solving the problem that helping by getting the ideal from the simulating physical that occur in a short period and may result in material damage or failure. The explicit dynamic of the simulating physical occurs quickly and may result in material damage or failure [8].

THE METHOD ANALYSIS USED IN ANSYS SOFTWARE

This paper analyzed using explicit dynamic analysis, static structural analysis, and topology optimization to prove and achieve the objective target to get the final result with finite element analysis using ANSYS software. All analyses have seven parts of the section in section analysis. It has engineering data to select the type of material used. In the geometry part, to import the design into ANSYS and the section of model until the result, it will connect the each other to do the analysis until the final result.

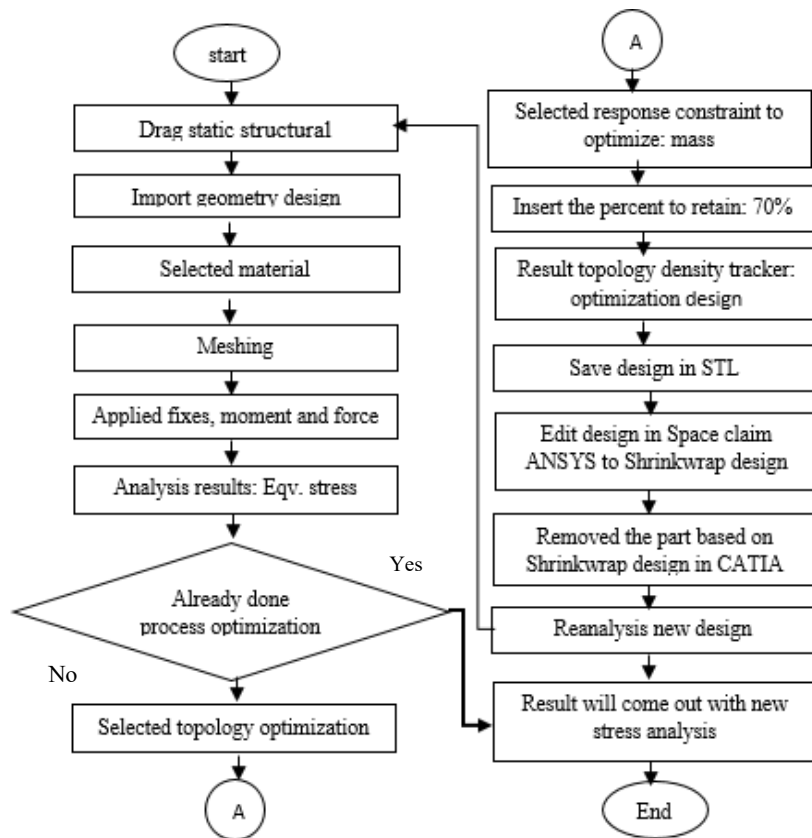
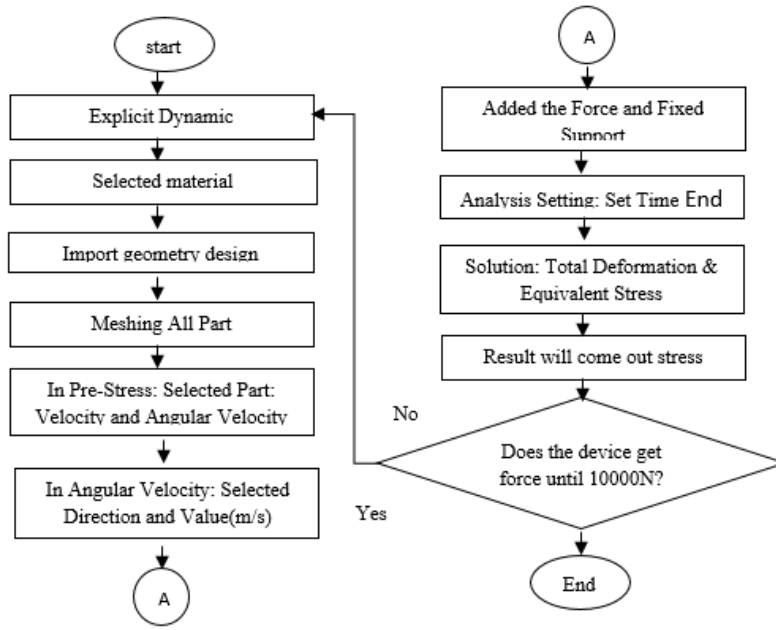


Figure 4 (a). Flow chart of the process analysis in static structural that include topology optimization and explicit dynamic



(b)

Figure 4 (b). Flow chart of the process analysis in static structural using ANSYS

EXPERIMENTAL RESULTS

Result of the Static Structural Analysis

This section discusses the results in finding the static structural that analyses the stress of the part that gets loads on the physical structures of the Quick-stop device. This static structure defined the stress by the maximum equivalent stress and maximum of the total deformation investigated by various load ranges. Besides that, for this QSD material that uses stainless steel, the yield strength for the maximum force on the structure is 250Mpa.

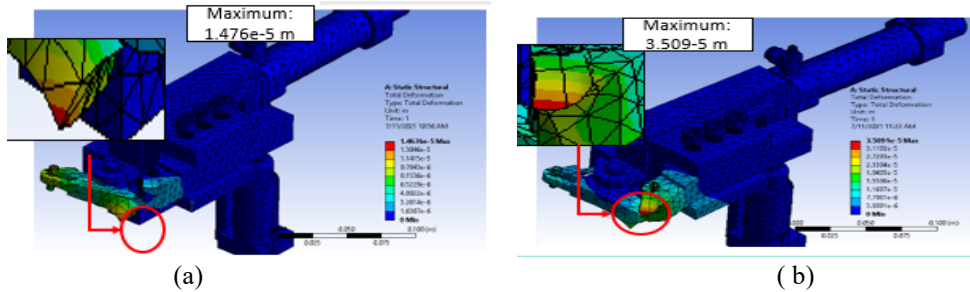


Figure 5. The result of the total deformation in (a) 1000N and (b) 5000N of load to toolholder

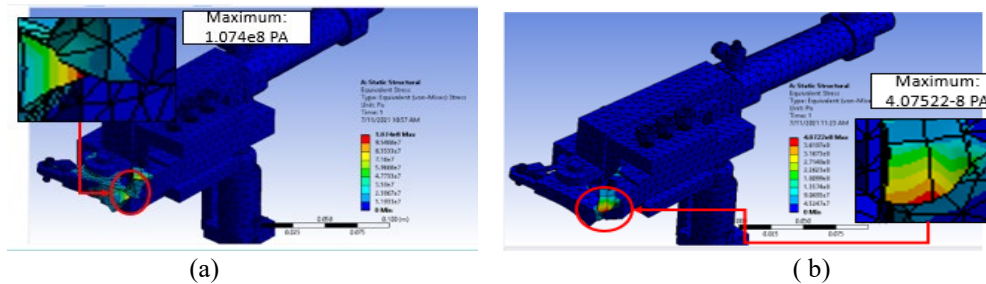


Figure 6. The result of the equivalent stress in (a) 1000N and (b) 5000N of load to tool holder

Based on Figure 5 and Figure 6 show the result of the total deformation and the equivalent stress in the static structural analysis. In referring figure 5 of the total deformation result show after the device gets a load of 5000 N, it starts to get stress on the toll holder, as shown in the contour of the level in red color compared to the load of 1000N that has only get on the trigger of the stress to the device. The total deformation result, the high of the stress level, means it gets the high of the curve to that part. Same as the equivalent stress, which also shows the same part of stress, but the result shows the stress. In this analysis that proved by increases the load, some parts will show the part of the level that gets the maximum

stress. In addition, if the part of the stress area reaches the maximum level of stress, its probability of getting fatigued from the physical structure.

Besides that, the equivalent stress in 1000 N gets 107.4 Mpa, which does not exceed the yield strength for this material in 250 Mpa. Mean the maximum of this structure material can able to support 1000 N. But compared with 5000 N, which is more than 250 Mpa of the yield strength, that is beyond the capacity of this material, and maybe it will affect the stress area that possibly can get a chance to fail. Furthermore, the stress deformation started at 3000 N, meaning there is no stress on the tool holder that can handle a force of less than 3000N.

Result of the Topology Optimization

Figure 7 shows the result of the suggested topology by the ANSYS software in a mass reduction of 30%. This optimization of the QSD gives the impact of load at 5000N. We can see that the result of the optimization of the structure of the part of QSD that removed the unnecessary spaces that unstressed spaces refer to the analyses of the stress already done in static structural. Furthermore, the result also reduced the mass and the volume by 30%, which has been removed as in Figure 7. After optimization of the mass and volume, the value was calculated through ANSYS software and is displayed in the topology results.

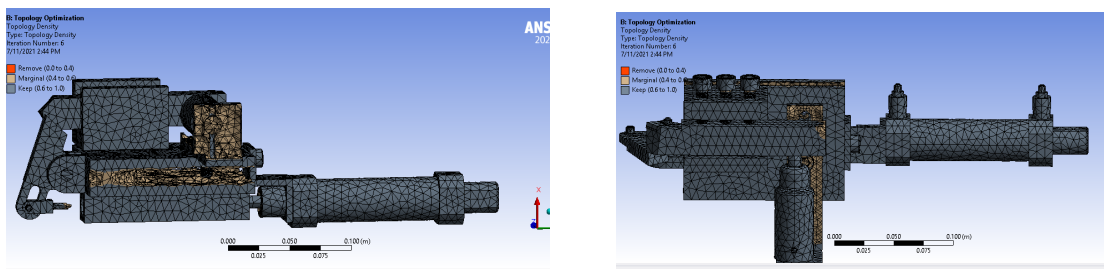


Figure 7. The result of the suggested Topology Optimization

Results of the redesign and reanalysis of the new design

After optimizing the quick-stop device, we needed to redesign the new QSD to repair the part that had already removed the unnecessary part in topology optimization. Figure 8 shows the new design that redesigned the design in the space claim that application in ANSYS used for design to be shrinkwrap of the surface at the area of the topology removed. The shrinkwrap also reduces the geometry’s complexity by enclosing holes and gaps and removing any sharp edges and corners based on the result of the topology suggestion that already removes the unnecessary part that unnecessary. After shrink wrapping the design, it needed to redesign following the shrinkwrap.

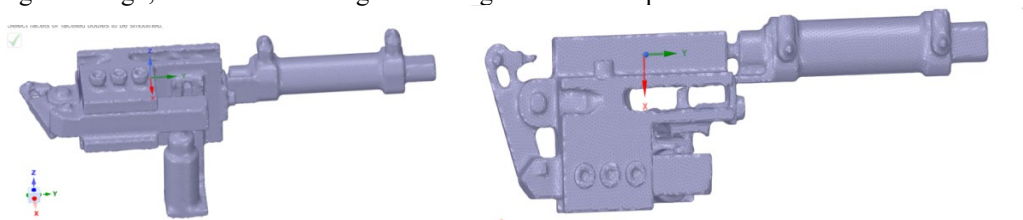


Figure 8. The result after shrinkwrapping the design

From the shrinkwrap design, the new design follows and refers to the measurement of the already shrinkwrap. Then, the design is redesigned using CATIA software, shown in Figure 9. Based on the result after shrinkwrap, the design shows that the topology optimization removes the unnecessary part, only that it removes part of the primary and the holder spring.

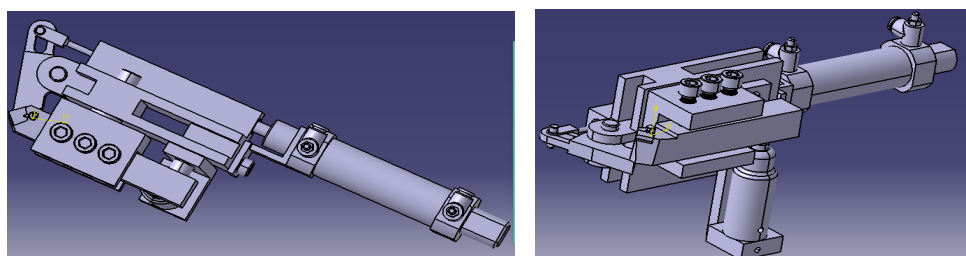


Figure 9. The design that redesign by using CATIA software

Result of mass before and after optimization

To prove the ANSYS result of mass that needed to measure with other applications. Before doing process topology optimization, the actual QSD measured the weight of the QSD using a weighing scale. The measurement before topology suggested that the result of the weight compared with the ANSYS weight and comparison both weights that get (a) 7.734 kg and (b) 7.4452kg that only different 3% of the error percentage between that two results. Hence, to know whether the ANSYS measurement of the mass is correct and the same as the actual of the Quick-Stop Device.

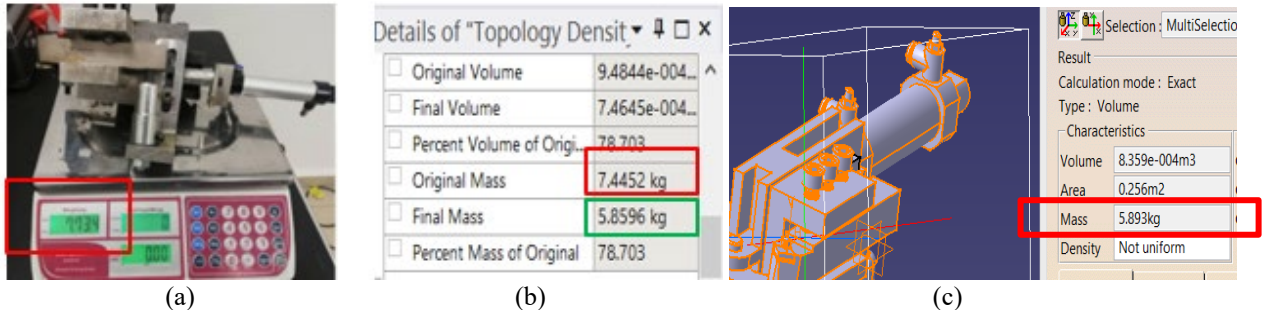


Figure 9. Method to measure the QSD mass (a) weight scale (b) ANSYS in topology results (c) CATIA

Then, after a redesign in Catia in Figure 9(c), it shows that this software has measured the mass and can also select the material of the device to check whether the measurement for the new design is the same as the ANSYS output of the developed design. Based on that, show the result of the mass after optimization that is the same as the final mass in ANSYS mass. From that, the design and the selected material of the new design are the same as the ANSYS measurement.

Result in static structural of maximum equivalent stress and maximum total deformation for six different load

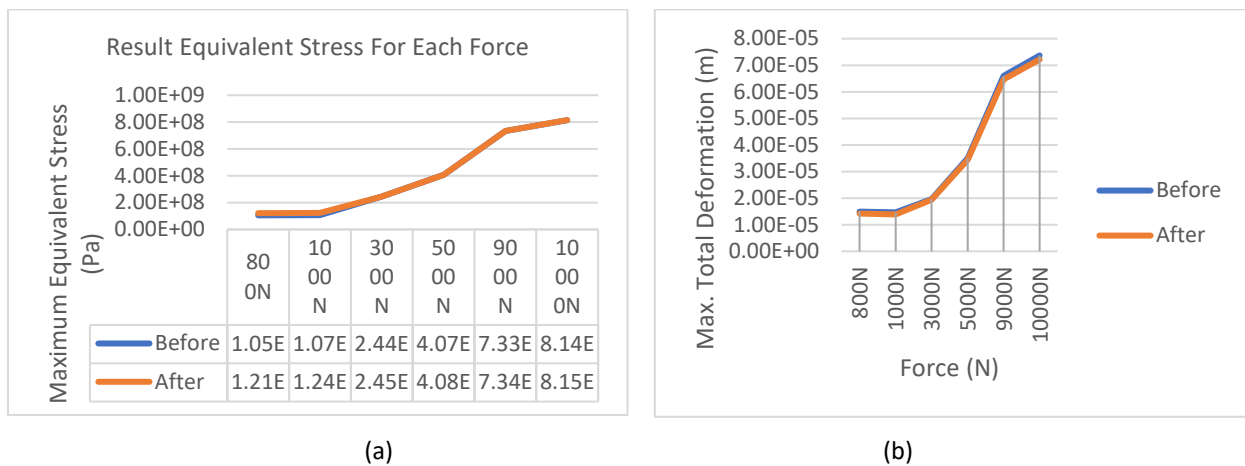


Figure 10. The comparison before and after optimization in equivalent stress (a) and total deformation (b)

In the equivalent stress that the table shows, the stress that compares with the yield strength for stainless steel does not exceed 250Mpa. For that result, the QSD can handle the stress in force 3000 N because the yield strengt for this material can support this force. Based on the result of the suggested topology by the software in mass reduction of 30%. In this optimization of the QSD, give the impact of load at 5000 N. We can see that the result of the optimization of the structure of the part of QSD that removed the unnecessary structure that unstressed spaces are referring the analyses of the stress that reduces the stress in the total deformation of stress. But the equivalent stress increases the stress result based on the calculation of the percentage error between before and after topology that the maximum increase stress is 0.18%. Besides that, the equivalent stress in force 800N and 1000N get the high stress in 15%.

Based on the redesigned analysis, the QSD already reduces the size and weight. Furthermore, it also shows the new design after optimization decreceses of the stress part that get the maximum stress in the total deformation compared to before optimization.

Result in the explicit dynamic of maximum total deformation before optimization of the QSD

This explicit dynamic analysis shows the reaction of the QSD through the trigger and the secondary body that shows the stress in part of the spring and tool holder. The QSD used the turning process when the tool holder gets the load in the cutting process before the trigger is needed to pull back to the end of the cutting. Based on the result, it shows the load response to the insert that gives import of the deformation of the stress on the tool holder (insert) and the spring. Furthermore, when the tool holder gets the load on the insert, the spring will compress to accommodate the force before the trigger pulls from the tool holder.

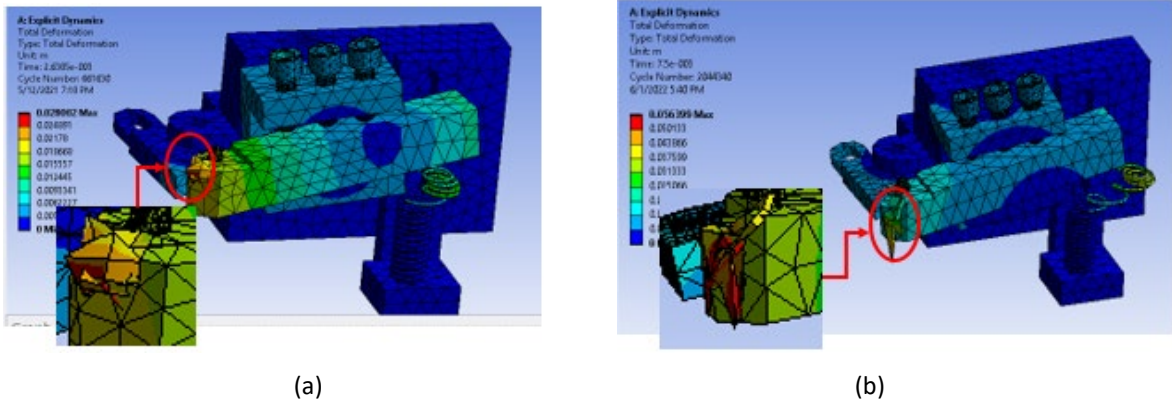


Figure 11. The reaction of the QSD before topology optimization (a) 5000 N and (b) 10000 N

Figure 11 (a) and (b) shows that the load started to break the insert on the tool holder and that the reaction of the QSD shows the stress in part of the spring and tool holder (insert). In addition, the figure also demonstrated that the tool holder gets the load in the cutting process after the trigger is needed to pull back to the end of the cutting. The maximum deformation in Figure 11(a) is 0.028002 m, in which the load gives the load high pressure. However, compared with 10000N, the QSD of the insert gets 0.056399 m that the maximum of the total deformation. Based on the result in Table 1, for 10000 N, the total deformation gives the impact until the insert on the tool holder move from the tool holder. This analysis proves that the QSD can be fatigued in 10000N.

Table 1. Results of error percentage of before and after optimization of the maximum of total deformation

Force	Before	After	Error (%)
800	0.01462	0.011470	21.67
1000	0.0146	0.012840	21.51
3000	0.018273	0.015255	16.52
5000	0.02500	0.02309	17.54
9000	0.048751	0.028488	29.74
10000	0.056399	0.048740	13.58

Result in the explicit dynamic of maximum total deformation after optimization of the QSD

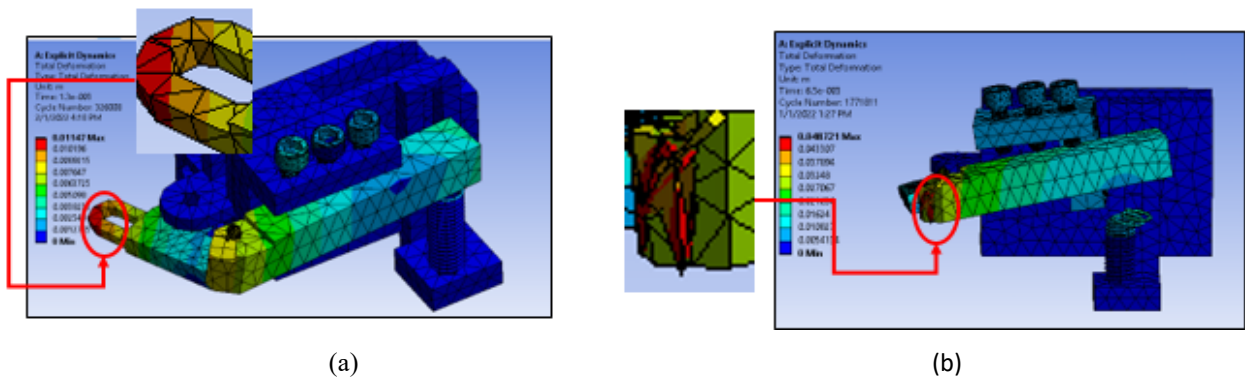


Figure 12. The reaction of the QSD before topology optimization (a)5000N and (b)10000

Figure 12 (a) and (b) illustrates the trigger’s reaction that shows the stress result in part of the trigger when it gets the load of 8000 N. The part of the trigger gets the high-stress cause the trigger needs to pull back to let the tool holder rotate downward to the end of the cutting process. On the contrary, the force on the insert is not affected, meaning

the insert is capable of handling 800N of the load. This ANSYS software can play a video of the reaction of the QSD in the cutting process that shows the reaction after getting the force (workpiece). So, from that, it can see the trigger's response to pull it away.

Figure 13 shows the reaction of the tool holder, and the spring shows the result of the stress that happens in part of the insert on the tool holder when it gets the load of 10,000 N. from that, it shows the faster object move, more kinetic has in the process of the workpiece to the insert of the tool holder. The figure also shows that the maximum of the total deformation is 0.01147 m. The load gives a high force of the load. Based on the result in 10,000 N, the total deformation gives the impact until the insert on the tool holder move from the tool holder. This analysis proves that the QSD has a chance to be fatigued or broken in load 10,000N.

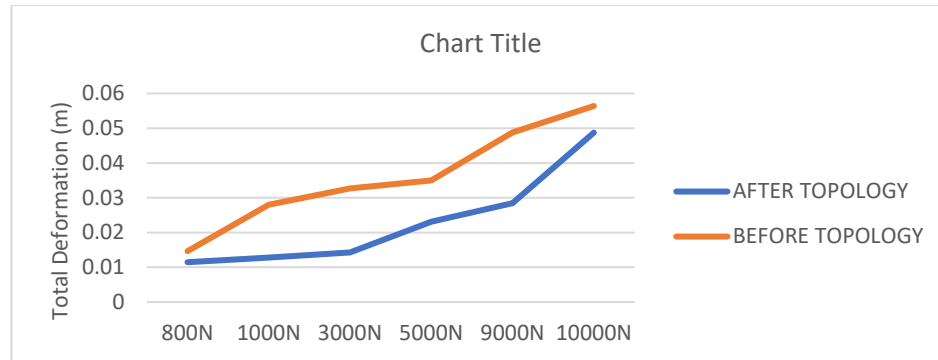


Figure 13. Comparison between before and after optimization of the result total deformation

Based on the comparison between before and after optimization in Figure 13. It shows that the total deformation also increases with the rise of the force. Furthermore, starting force of 3,000 N until 10,000 N affected the insert and the force of the loads.

Table 2. Results of error percentage of before and after optimization of total deformation

Force	Before	After
800	0.01462	0.011470
1000	0.0146	0.012840
3000	0.018273	0.015255
5000	0.02500	0.02309
9000	0.048751	0.028488
10000	0.056399	0.048740

Comparing before and after topology optimization proves that the value that in 800N until 1000N gets higher, reducing the stress in part on the trigger in the total deformation. This is after removing the unnecessary part in the primary body of QSD that can refer to in Table 2. From that, the results of total deformation affected the tool holder's momentum and the spring. The 800 N until 1000N is less momentum than the 5000N until 10000N of the force. Besides that, the momentum of the direction follows the direction of the velocity

CONCLUSION

To conclude, this paper analyzes the stress and optimization of the Quick-Stop device with finite element analysis using ANSYS, which can help analyze the structure's behavior after getting the load and find the optimal design to reduce the weight of QSD. Based on the results, increased load also raised the stress in the area of the critical part. Therefore, suggesting topology optimization is very helpful in defining the best design that can remove the unnecessary part in the QSD. Moreover, after doing optimization that needed redesign and then reanalyzing the stress analysis, the new design of the QSD comes by reducing the weight and stress on the critical part areas. Then, compare before and after optimization to see the improvement of the cutting process of chip formation. Finally, based on the results obtained, it can be concluded that using ANSYS software is very helpful in analyzing stress to increase productivity, create better designs after optimization, and produce innovative products in less production time. It is a very effective and efficient tool in ANSYS that is suitable for performing stress analysis and optimization of various engineering structures.

ACKNOWLEDGEMENT

REFERENCES

- [1] M. Satheesha, V. K. Jain, and P. Kumar, "Design and development of a quick-stop device (QSD)," *Precis. Eng.*, vol. 12, no. 4, pp. 205–212, 1990.

- [2] Y. Altintas, C. Brecher, M. Week, and S. Witt, "Virtual Machine Tool," *CIRP Ann. - Manuf. Technol.*, vol. 54, no. 2, pp. 115–138, 2005.
- [3] X. P. Li, L. Y. Zhao, and Z. Z. Liu, "Topological Optimization of Continuum Structure based on ANSYS," *MATEC Web Conf.*, vol. 95, pp. 4–7, 2017.
- [4] A. P. Apcoer, "Topology Optimization of Front Leaf Spring Mounting Bracket," *Int. J. Sci. Dev. Res.*, vol. 3, no. 7, 2018.
- [5] S. H. Mousavi Azam and E. Ahmadloo, "Analysis of Chip Removal Operations via New Quick-Stop Device," *Mater. Manuf. Process.*, vol. 31, no. 13, pp. 1782–1791, 2016.
- [6] R. Bruno, D. Pereira, and C. L. Santos, "Design of Quick-Stop Device Easy and Safe To Operate for Chip Design of Quick-Stop Device Easy and Safe To Operate for," no. January 2016, 2011.
- [7] A. Gautam, "Static Stress Analysis of Connecting Rod Using Finite Element Approach," *IOSR J. Mech. Civ. Eng.*, vol. 10, no. 1, pp. 47–51, 2013.
- [8] G. S. Kumar and R. R. Reddy, "Simulation of Turning Process using Explicit Dynamics," *Int. Res. J. Eng. Technol.*, vol. 6, no. 1, pp. 1463–1467, 2019.