DESIGN AND ANALYSIS OF THE KOMATSU PC400 EXCAVATOR'S TOOTH BUCKET THICKNESS USING THE FINITE ELEMENT METHOD

by Yulia Yusrini Djabir

MECHTA
International Journal Of Mechanical Engineering Technologies & Applications
ISSN 2722-3213 articles 2, pp. 130 – 138, 2024

V05 N.2

DESIGN AND ANALYSIS OF THE KOMATSU PC400 EXCAVATOR'S TOOTH BUCKET THICKNESS USING THE FINITE ELEMENT METHOD

Sutrisna [1] ✉, Angger Bagus Prasetiyo [1], Fatur Rizki Anggun Juniar [1]

[1] **Mechanical Engineering**
Institut Teknologi Nasional Yogyakarta, Indonesia
sutrisna@itny.ac.id.

Abstract

Bucket teeth are an important component of an excavator that functions as a material penetrator or digger. This part is prone to failure because it is in direct contact with the ground. This study aims to determine the value of stress, strain, deformation, and safety factors that occur in the Komatsu PC400 excavator bucket teeth against thickness variations of 2.5 mm, 5 mm, 7.5 mm, and 10 mm. This study uses a computer with specifications: processor: Intel (R) Celeron (R) N4000 CPU @ 1.10 GHz, memory: 8 GB. Windows 10 Home Single Language 64-bit Operating System. This computer is equipped with Autodesk Inventor Professional 2023 and ANSYS Workbench R1 2023 software. The method used in this research is a testing method using ANSYS software with a finite element method approach, namely static structural. The simulation results of bucket teeth show the maximum deformation values are 0.16382 mm, 0.13832 mm, 0.1249 mm, and 0.11619 mm, respectively. Furthermore, the maximum equivalent stress values are 108.6 Mpa, 79.712 Mpa, 80.338 Mpa, and 79.992 Mpa, respectively. For the equivalent elastic strain maximum, 0.00052993, 0.00038899, 0.000392, and 0.00039029 were obtained. Then the safety factor value is obtained 3.8214, 5.2062, 5.1657, and 5.188. This shows that the thicker the thickness variation, the better the strength value.

Keywords: *Bucket Teeth Excavator, Inventor, ANSYS, Finite Element Analysis, Static Structure*.

1. INTRODUCTION

In use, the bucket teeth experience high stress because they are in direct contact with the ground. Therefore, the materials used to make bucket tooth components must have sufficiently high characteristics of wear resistance, strength, and ductility [1]. Bucket tooth damage is usually caused by several factors, namely soil type, excavation speed, excavation edges, and the material used to make the bucket teeth. The design of the bucket tooth geometry must be in accordance with the requirements so that the use time can be extended and costs can be reduced [2].

In line with developments in manufacturing technology, one of which is creating modern designs that can minimize production costs [3]. The next step that needs to be taken before the manufacturing process is to analyze the design, minimize failures, extend the usage time of components [4], and detect product damage. The excavator manufacturing process does not take into account the teeth required for the bucket and the weight of the teeth. Consequently, the production of a tooth does not need design compensation; instead, it uses

Corresponding Author:
✉ **Sutrisna**
sutrisna@itny.ac.id
Received on: 2024-03-28
Revised on: 2024-06-27
Accepted on: 2024-06-27

DOI: 10.21776/MECHTA.2024.005.02.2

the previously completed success data [5]. Product failure can occur due to stresses exceeding the specified yield stress [6]. A structure can be considered a failure because it is unable to withstand the required load [6]. Designers use software that can simulate numerically to help solve engineering problems [7]. Bucket tooth modeling can be simulated using Ansys software which is equipped with a finite element method. When the fatigue point occurs, the strength structure of the bucket teeth undergoes changes, such as stress, strain, freezing, and safety factors[6]. The advantage of using Ansys software is the ability to discretize the model very finely with more elements, which produces high-fidelity output [8].

Based on the problems above, further research on bucket tooth strength is needed to obtain bucket tooth quality that meets excavation needs. In this case, the stress distribution on the bucket teeth is used to improve the existing design using the finite element method, and static structural simulation is used. In this research, we will modify bucket teeth with thickness variations of 2.5 mm, 5 mm, 7.5 mm, and 10 mm on excavator bucket teeth. The aim of this research is to obtain the values of stress, strain, deformation, and safety factors that occur.

2. MATERIALS AND METHODS

2.1. Bucket Teeth Modeling Design

The pre-processing stage is creating a bucket tooth modeling design using Autodesk Inventor Professional 2023 software, after creating a product design, then save it in model form *STEP so it can be included in the program ANSYS. Figure 1 shows the product design to be analyzed.

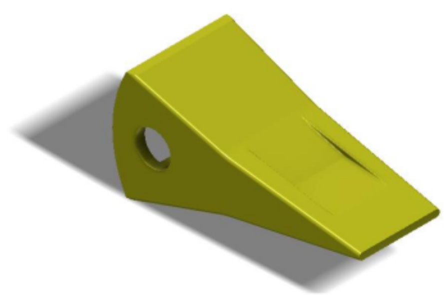

Figure 1. Bucket Teeth Modeling Design

2.2. Computational Domain Division (Meshing)

The next step after the product design has been created, the next stage is meshing, the bucket tooth model design will be discretized into small elements so that the software can simplify the iteration process [9], [10]. The division of the domain into small elements is done for control purposes and produces a more convergent output [11]. when numerical simulations are carried out [7] in engineering design [6]. The meshing results in the bucket tooth product design are tetrahedral type with many nodes and elements as shown in Table 1. Mesh determination is carried out using the finest scale so that the modeling simulation is more optimal [12].

Table 1. Number of nodes and elements

Description	Value
Nodes	115980
Elements	77976

2.3. Initial Condition and Materials Selection

The initial condition that is used is a fixed support, as shown in Gambar 2a. The use of fixed support is marked in the hole that connects the adapter pin, teeth, bucket, and bucket. The load applied is a force of 8285.06 N with a force vector of 32 degrees. The load is placed on the tip of the bucket tooth because it is in direct contact with the ground so it experiences extreme loads. The loadings on the model are shown in Figure 2b.

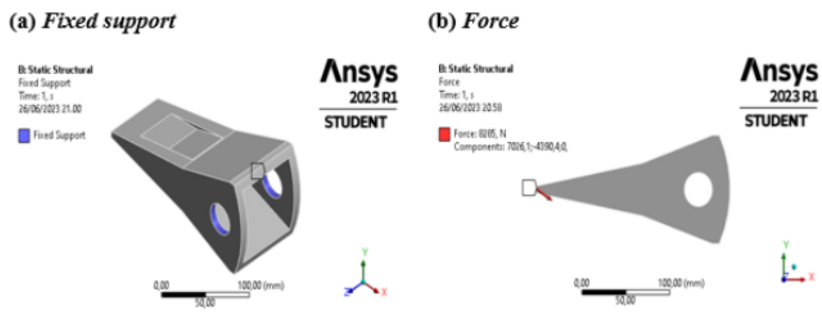

Figure 2. (a) Fixed Support and (b) Force

ANSYS software is facilitated by various types of materials that can be adjusted by the user [13]. Apart from that, it can be done manually to input material properties such as yield strength, tensile strength, density, and shear modulus. The material properties of bucket teeth are shown in Table 2.

Table 2. Material Properties AISI 4140

No	Characteristic data AISI 4140	Value
1	Modulus elastisitas (E)	205 GPa
2	Poisson Ratio	0.29
3	Massa Jenis	7,85 g/cm^3
4	Yield Strength	415 MPa
5	Tensile Strength	655 MPa
5	Bulk Modulus	162 GPa
6	Shear Modulus	79 GPa
7	Thermal expansion coefficient	12,2 μm/m°C
8	Thermal conductivity	42,6 W/mK

3. RESULTS AND DISCUSSION

Structural static analysis is a determinant of the stress of materials and structures that have static and dynamic forces and loads [14]. Equivalence is a determining factor in whether a product design is suitable for use or will fail [15]. The simulation results will be obtained in the form of total deformation, stress, strain, and safety factors.

3.1. Deformation Total

Deformation is a physical or chemical change in shape due to rotational and radial loads experienced by an object. The smaller the number resulting from deformation, the stronger the material. When an object is subjected to a force or load, deformation can occur [6]. Figure 3 shows the results of the total deformation of the bucket teeth.

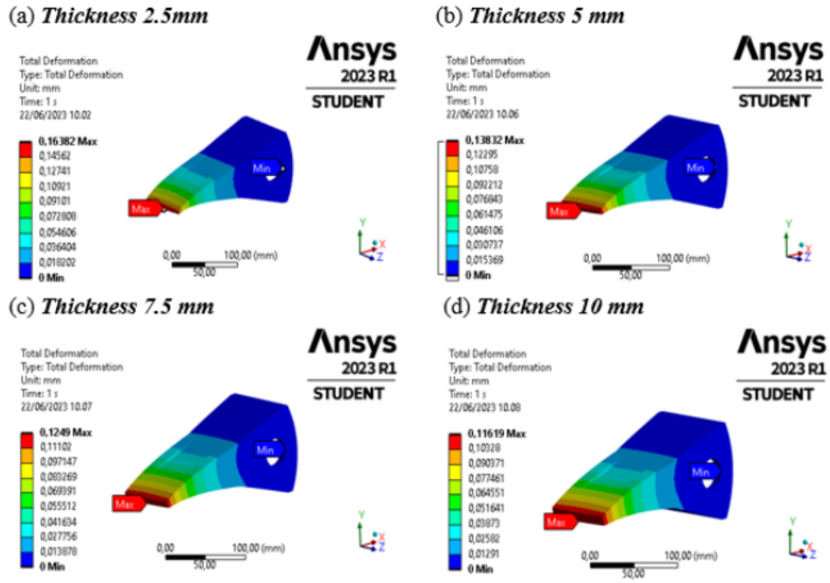

Figure 3. Simulation results of total deformation with a thickness (a) 2.5 mm (b) 5 mm (c) 7.5 mm and (d) 10 mm

In the simulation of bucket teeth with thicknesses of 2.5 mm, 5 mm, 7.5 mm, and 10 mm, the maximum deformation values are 0.16382 mm, 0.13832 mm, 0.1249 mm, and 0.11619 mm, while the minimum deformation value is 0. This shows that the stronger the material, the smaller the deformation that occurs. Based on the simulation results, the part that receives the force load has the highest pressure. This shows that the most critical part is the tip of the tooth because it experiences extreme loads [12], when bucket teeth are used continuously, both ends will be damaged. Failure in the form of wear, bending, cracking, and breaking during use [13].

3.2. Equivalent Stress

Stress is the result of the comparison between the vertical force acting on the cross-sectional area of an object. Equivalent stress on the bucket teeth aims to see how the maximum stress on the bucket teeth occurs when penetrating the ground. This way, you can find out whether any part of the bucket tooth has failed or not. If the maximum stress value of the bucket teeth during the simulation remains below the stress limit permitted by the bucket tooth material, then it can be said that the bucket tooth shape is safe [13].

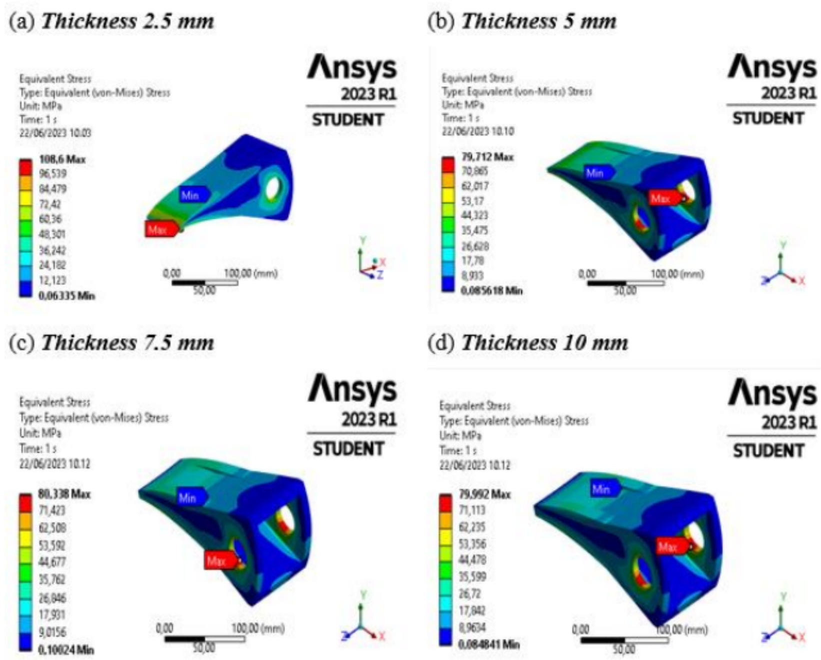

Figure 4. Simulation results of equivalent stress with a thickness (a) 2.5 mm (b) 5 mm (c) 7.5 mm and (d) 10 mm

In the simulation of bucket teeth with varying thicknesses of 2.5 mm, 5 mm, 7.5 mm, and 10 mm, the maximum equivalent stress value was 108.6 Mpa., 79.712 Mpa, 80.338 Mpa, dan 79.992 Mpa respectively. Meanwhile, the minimum equivalent stress is 0.06335 Mpa, 0.85618 Mpa, 0.10024 Mpa and 0.084841 Mpa. The highest equivalent voltage can be achieved when the available load increases because the voltage relationship is inversely proportional to the fundamental parameters observed by a system [16].

3.3. Equivalent Elastic Strain

Strain is the increase in the length of an object relative to its initial length caused by an external force that influences the object. The type of material used will also affect the equivalent elastic strain results [15].

In a bucket simulation with adjustment ranges of 2.5 mm, 5 mm, 7.5 mm, and 10 mm, The equivalent maximum elastic strain is obtained 0.00052993, 0.00038899, 0.000392, dan 0.00039029. Figure 4.5 shows that the modeling simulation results do not affect the results of the equivalent elastic strain contour shape. The largest equivalent elastic strain occurs on the side of the bucket teeth. This is because the distribution of stress and strain is influenced by the location of the specified load [13].

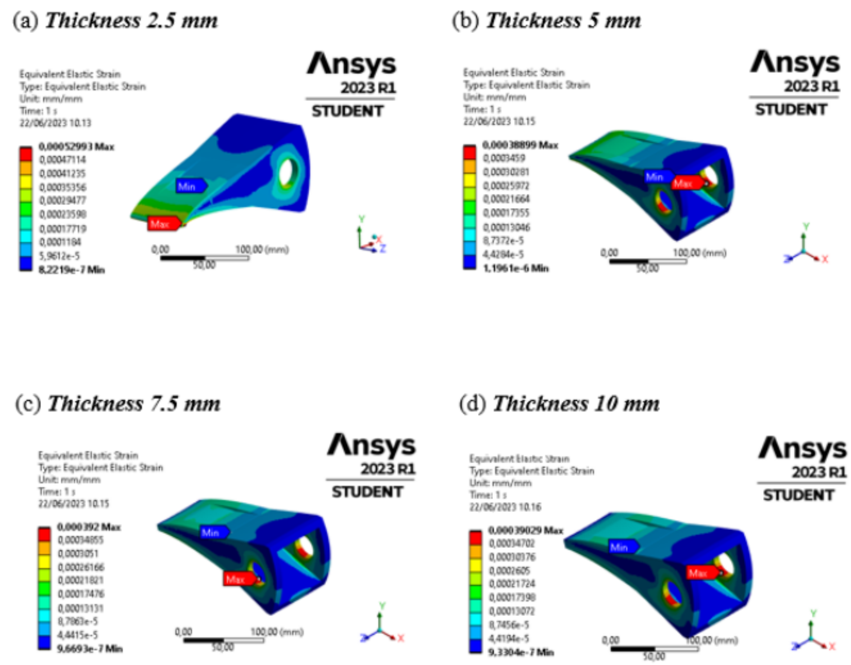

Figure 5. Results of simulating equal elastic strain with a thickness (a) 2.5 mm (b) 5 mm (c) 7.5 mm and (d) 10 mm

3.4. Safety Factor

The lowest value of the design safety factor is a good indicator of how safe it is to use . A safety factor is one of the factors used to assess stress testing in object modeling [6]. Figure 6 shows the results of the bucket tooth simulation. The simulation results show that the minimum value of the safety factor for bucket tooth simulation is 3.8214, 5.2062, and 5.188. This shows that this research design is safe to use and meets the desired safety values [6].

Figure 6 shows the value of the safety factor which changes due to variations in the thickness of the excavator bucket teeth from 2.5 mm, 5 mm, 7.5 mm, dan 10 mm. The graph shows that the value of the safety factor increases due to the influence of thickness variations. Based on the graph above, the thicker the bucket teeth, the better the strength value [17], [18].

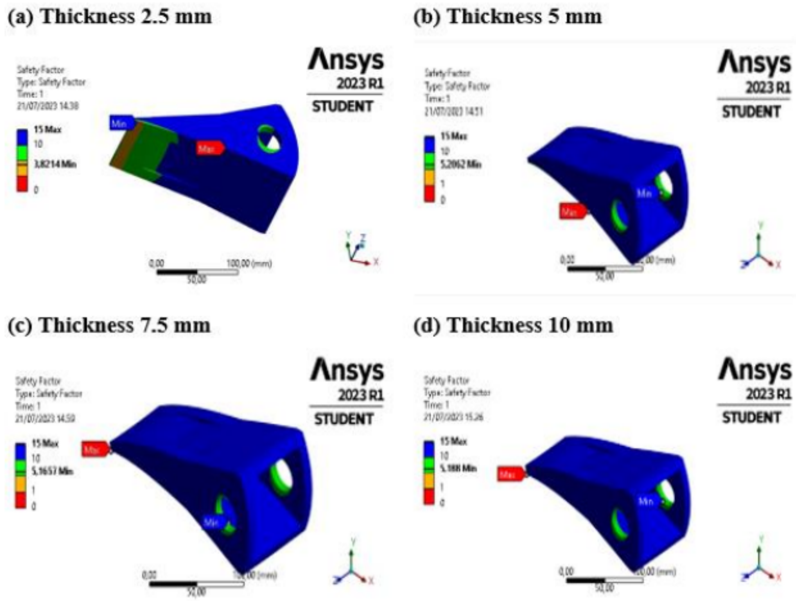

Figure 6. Simulation results of total deformation with a thickness (a) 2.5 mm (b) 5 mm (c) 7.5 mm and (d) 10 mm

4. CONCLUSION

The analysis and discussion presentation findings showed that there is a decrease in the overall deformation value for thickness variations of 2.5 mm, 5 mm, 7.5 mm, and 10 mm. The largest thickness is 2.5 mm with 0.16382 mm indicating the highest degree of deformation. The equivalent stress and equivalent elastic strain values were the smallest at a thickness of 5 mm due to a significant decrease. The smallest equivalent stress value is 79.712 MPa. The smallest equivalent elastic strain value is 0.00038899. The highest safety factor value is at a thickness of 5 mm, The minimum safety factor figure obtained was 5.2062 due to a significant increase.

ACKNOWLEDGMENTS

The author would like to thank all parties who have helped and assisted in completing this research, especially ITNY which has provided funding.

REFERENCES

[1] S. H. Suryo, A. P. Bayuseno, J. Jamari, dan A. I. Wahyudi, "Analysis of Rake Angle Effect to Stress Distribution on Excavator Bucket Teeth Using Finite Element Method," Civ. Eng. J., vol. 3, no. 12, hal. 1222, 2018, doi: 10.28991/cej-030952.

[2] A. Tasevski dan C. Hedlund, "Design of bucket teeth," 2016.

[3] B. Chirende, J. Q. Li, dan W. Vheremu, "Application of finite element analysis in modeling of bionic harrowing discs," Biomimetics, vol. 4, no. 3, hal. 1–11, 2019, doi: 10.3390/biomimetics4030061.

[4] G. M. Vegad dan R. Yadav, "Design analysis and optimization of rotary tiller blades using computer software," AMA, Agric. Mech. Asia, Africa Lat. Am., vol. 49, no. 1, hal. 43–49, 2018.

[5] Bilal Pirmahamad Shaikh dan Abid M. Mulla, "Analysis of Bucket Teeth of Backhoe Excavator Loader and its Weight Optimization," Int. J. Eng. Res., vol. V4, no. 05, hal. 289–295, 2015, doi: 10.17577/ijertv4is050402.

[6] A. B. Prasetiyo dan K. A. Sekarjati, "Analisis Struktur Desain Pisau Pengupas Tempurung Kelapa Menggunakan ANSYS 19.2," Semin. Nas. Ris. Inov. Teknol., vol. 1, no. 1, hal. 417–423, 2022.

[7] S. M. Kalikate, S. R. Patil, dan S. M. Sawant, "Simulation-based estimation of an automotive magnetorheological brake system performance," J. Adv. Res., vol. 14, hal. 43–51, 2018, doi: 10.1016/j.jare.2018.05.011.

[8] A. B. Prasetiyo, K. A. Sekarjati, dan Sutrisna, "Numerical analysis of the influence iron type on Von Mises Stress and safety parameters for compost processing machine frame construction," IOP Conf. Ser. Earth Environ. Sci., vol. 1151, no. 1, 2023, doi: 10.1088/1755-1315/1151/1/012058.

[9] A. B. Prasetiyo dan F. Fauzun, "Numerical study of effect of cooling channel configuration and size on the product cooling effectiveness in the plastic injection molding," MATEC Web Conf., vol. 197, hal. 8–11, 2018, doi: 10.1051/matecconf/201819708019.

[10] A. B. Prasetiyo, A. A. Azmi, D. S. Pamuji, dan R. Yaqin, "Pengaruh Perbedaan Mesh Terstruktur dan Mesh Tidak Terstruktur Pada Simulasi Sistem Pendinginan Mold Injeksi Produk Plastik," Pros. Nas. Rekayasa Teknol. Ind. dan Inf. XIV Tahun 2019, vol. 2019, no. November, hal. 400–406, 2019.

[11] M. Sosnowski, J. Krzywanski, dan R. Scurek, "A fuzzy logic approach for the reduction of mesh-induced error in CFD analysis: A case study of an impinging jet," Entropy, vol. 21, no. 11, 2019, doi: 10.3390/e21111047.

[12] A. B. Prasetiyo *et al*., "Finite Element Analysis (FEA) of blade weed design using Ansys workbench," Sinergi, vol. 26, no. 3, hal. 371, 2022, doi: 10.22441/sinergi.2022.3.012.

[13] A. B. Prasetiyo, K. A. Sekarjati, dan I. P. A. Assagaf, Sutrisna, "Analisis Frekuensi Natural Velg Ring 16 Menggunakan Finite Element Method," in Prosiding Nasional Rekayasa Teknologi Industri dan Informasi XVII Tahun 2022 (ReTII), 2022, vol. 2022, no. November 2021, hal. 354–359.

[14] L. A. N. Wibawa, "Desain dan Analisis Tegangan Alat Pengangkat Roket Kapasitas 10 Ton Menggunakan Metode Elemen Hingga," J. Energi dan Teknol. Manufaktur, vol. 2, no. 01, hal. 23–26, 2019, doi: 10.33795/jetm.v2i01.31.

[15] L. A. N. Wibawa, "Pengaruh Diameter Baut Terhadap Kekuatan Rangka Main Landing Gear Pesawat Uav Menggunakan Metode Elemen Hingga," J. Polimesin, vol. 17, no. 1, hal. 26–32, 2019, doi: 10.30811/jpl.v17i1.828.

[16] S. H. Pranoto dan M. Mahardika, "Design and finite element analysis of micro punch CNC machine modeling for medical devices," AIP Conf. Proc., vol. 1941, no. March, 2018, doi: 10.1063/1.5028079.

[17] L. A. N. Wibawa, Simulasi Kekuatan Komponen Sarana Pengujian Roket Menggunakan Autodesk Inventor Professional 2017, no. June. 2018.

[18] A. B. Prasetiyo dan K. A. Sekarjati, “Finite Element Simulation of Power Weeder Machine Frame,” Indones. J. Comput. Eng. Des., vol. 4, no. 2, hal. 26–34, 2022, doi: https://doi.org/10.35806/ijoced.v4i2.291.

DESIGN AND ANALYSIS OF THE KOMATSU PC400 EXCAVATOR'S TOOTH BUCKET THICKNESS USING THE FINITE ELEMENT METHOD

ORIGINALITY REPORT

14%	13%	4%	6%
SIMILARITY INDEX	INTERNET SOURCES	PUBLICATIONS	STUDENT PAPERS

MATCH ALL SOURCES (ONLY SELECTED SOURCE PRINTED)

4%

★ journals.stmjournals.com

Internet Source

Exclude quotes	Off	Exclude matches	< 1%
Exclude bibliography	On		