Meat Cutting Machine Shaft Design and Analysis

Cipto Cipto*, Klemens A Rahangmetan, Christian Wely Wullur, Farid Sariman, and Hariyanto Hariyanto

Department of Mechanical Engineering, Faculty of Engineering, Universitas Musamus, Merauke 99600, Indonesia

Abstract. This study analyzes the maximum load on the shaft construction with a diameter of 12 mm and a length of 581 mm. The shaft is designed as a shaft for cutting meat with a capacity of 5 kg. The analysis was performed using the finite element analysis method included in the Autodesk software. According to mathematical calculations, the shaft is considered safe because the value of the admissible tension $\tau_a = 7.380$ kg / mm² is greater than the maximum tension $\tau_p 5.62$ kg / mm². Based on the simulation results of the test, the shaft experiences a maximum off-stress of 61.89 MPa, a maximum displacement of 0.07715 mm, and a safety factor of 3.34 μ l so that the shaft is classified as safe for use with a Load capacity of 5 Kg

Keywords: Rice transport vehicle, Strength analyzed, Finite element analysis

1 Introduction

1.1 Shaft and its function

The shaft is a machine element that continues the force and rotation. The shaft usually has a diagonal cylindrical shape in which other motor elements are built, such as e.g., B. B. gears, pulleys, flywheels, cranes, bearings and other items. The shaft has the properties of accepting bending, pulling, threading or twisting, which work alone or in combination with one another [1]. The wave capabilities can be analyzed using many methods, e.g., B. through direct tests (field tests) or with the help of the software. The wave function is very important, requires accuracy in the construction, the shaft strength is adjusted to the maximum planned load, the calculation method allows the analysis of the components, in this case the shaft, for their safety when taking loads or their exhaustion limit [2]. The evaluation of the service life of components (shaft) by analyzing the stress / strain experienced can identify component fatigue [3].

1.2 Calculation of shaft diameter

When calculating the shaft diameter, the correction factor recommended by ASME must be taken into account. Correction factor due to collisions expressed by K_t . The correction factor for subtle loading is 1.0, if a light bump or impact occurs then 1.0-1.5 is chosen. If the load is carried with a shock or a large impact, then 1.5-3.0 is chosen. The correction factor found in this study was 1.5 because the raw material (meat) was previously placed in the grinding machine when the machine was not in operation, so that the shaft of the

grinding machine suddenly received a large impact. Another factor that must be taken into account when determining the shaft diameter is the bending load C_b that the shaft can absorb. Therefore, when determining the shaft diameter, add a correction factor for the bend C_b of 1.2

Then the formula for planning the ds shaft diameter is obtained:

$$d_s = \left[\frac{5.1}{\tau_a} K_t C_b T \right]^{1/3} \quad (1)$$

d_s = Planned shaft diameter (mm)

τ_a = Material tensile strength (kg / mm²) at

K_t = Correction factor for possible collisions

C_b = Correction factor for the possibility of bending load.

1.2 Shaft Power

The energy is obtained from an electric motor, which is transmitted to the meat grinder shaft via the V-belt. At the time of start-up (meat), a lot of effort may be required because the meat is still intact. After a few moments, the meat is cut into small pieces, so it is necessary to correct the average power required using the correction factor in the planning.

The types of correction factors based on the power to be transmitted according to table

* Corresponding author : cipto@unmus.ac.id

Table 1. The types of correction factors are based on the power transmitted [1]

Transmitted power	f_c
Average power needed	1,2 – 2,0
Maximum power needed	0,8 – 1,2
Normal power	1,0 – 1,5

In this shaft calculation, the maximum power is taken as the planned power, with the aim that the actual shaft can absorb a large impact torque. The correction factor used is $f_c = 1.0$.

$$P_d = N \cdot f_c \quad (2)$$

P_d = Planned power (kW)
 f_c = Correction factor
 N = Normal output motor power output (kW)

The determination of the main axis dimensions is calculated based on the torsional load and the possibility of bumps / bumps during the load, when the engine starts. The amount of torque acting on the shaft can be calculated:

$$T = 9,74 \cdot 10^5 \frac{P_d}{n} \quad (3)$$

T = Plan torque (kg.mm)
 P_d = Planned power (kW)
 N = Rotation (rpm)

The shaft uses material of the type S50C, for which a tensile strength of $\sigma_b = 62 \text{ kg} / \text{mm}^2$ is assumed in your planning. Then the torsional stress of the material can be obtained from the formula:

$$\bar{\tau}_a = \frac{\sigma_b}{s_{f1} \cdot s_{f2}} \quad (4)$$

τ_a = Shear stress clearance (kg/mm²)
 σ_b = Tensile strength of materials (kg/mm²)
 S_{f1} = Safety factors that depend on the type of material
 S_{f2} = Safety factors that depend on the shape of the shaft (1,3-3,0)

1.3 Software Autodesk Inventor

Autodesk Inventor is a CAD program (Computer-Aided Design) with solid three-dimensional modeling functions for the visual creation of 3D prototype objects, the simulation and drawing as well as the documentation of the data [4]. In Inventor, a designer can sketch 2D products and model them in 3D to proceed with creating a visual prototype or an even more complex simulation [5–7]. Autodesk Inventor, developed by the US software company Autodesk, is CAD software for mechanical 3D design for creating digital 3D prototypes for product design, visualization and simulation [5].

Structure Analysis on Autodesk Inventor:

a. Stress Analysis

Stress analysis is one of the structural testing tools in Autodesk Inventor that is performed using the concept of finite element analysis (FEA). The way it works is to break an object structure under test into connected finite elements, which are managed by special calculations of the software in order to achieve more precise results [5, 8].

b. Frame Analysis

In addition to stress analysis, Autodesk Inventor has another tool for testing the structure, namely frame analysis. The concept of this test is to apply engineering mechanics related to the structure of the truss, beam and frame. Load and assist data entry while the output stress, strain, and displacement graphs are displayed [5, 8].

c. Prinsip Superposisi

Fig. 1. Displacement on the principle of superposition

For example, an object / structure is subject to three styles P_1 , P_2 and P_3 . In the same place and direction with the three forces there was a shift in the components of q_1 , q_2 and q_3 [9].

2 Material and Method

2.1 Shaft Material

The metal material used to manufacture the shaft is selected according to the standard (JIS G 4501). Castings with a high carbon content with a content of $C > 0.5\%$ (construction machinery made of carbon steel S50C) with a tensile strength of $62 \text{ kg} / \text{mm}^2$.

2.2 Shaft Strength Measurement

The strength measurement in the form of a shear stress test with torque acting on the shaft is carried out in order to determine the maximum rigidity of the shaft. The specified shear stress must not exceed the permissible shear stress so that the shaft does not fail.

The shear stress due to the torque moment acting on the shaft is obtained from:

$$\tau_p = \frac{16T}{\pi d_s^3} \quad (5)$$

2.3 Loading Conditions

The shaft load in the meat grinder is as follows. The shaft is supported by two plain bearings and carries two pulleys of different sizes. The geometric shape of the

shaft and the direction of its loading area in the picture below.

Fig. 2. Meat cutting machine shaft and its components

Fig. 3. Shaft geometry and dimensions.

2.4 The design of the Axis Analysis fund uses the Autodesk Inventor 2015 software.

The design process of sample shaft test design using Autodesk Inventor 2015 software is then carried out in a simulation process with the following steps:

1. Preprocessing, consisting of construction of the motor shaft geometry, determination of the material type, determination of the fixed area, intervention process and determination of the load location and the force magnitude.
2. The process of modeling the shaft geometry with Autodesk Inventor 2015.
3. Create a simulation.
4. Determine the type of material according to the DTA specifications.
5. Static analysis (static analysis), consisting of determining the type of material, determining the hold (fixed), the process of engagement and determining the location of the load and the magnitude of the force.
6. Determination of the boundary conditions and types of loading (boundary conditions).
7. Meshing, in which this geometry is divided into small parts in the form of lines connected by nodes throughout the geometry of objects.
8. Dynamic analysis (modal analysis), which is carried out to find out the personal frequency of the wave in different modes.
9. Solve / execute simulate this process. The data entered in the previous stage is processed to obtain the results of the analysis on the simulation tool.
10. Post-processing is the final process to complete the finite element method. Autodesk Inventor simulation results are displayed in the form of images and total deformation data, as well as

calculation results for the maximum stress in the simulation process.

3 Results and Discussion

The results of the study were obtained from the results of simulation tests using the Autodesk Inventor software. The force exerted on the shaft is the force of gravity that acts on the shaft and was calculated beforehand. The results obtained are of false stresses, displacements and safety factors.

3.1 Von misses stress

Missing Stress is the result of calculating the stress-strain relationship in an object model, the strain obtained from the deformation of the model. The equivalent stress used by the Von Mises method. The representation of the results of the equivalent stress analysis is shown in Figure 4 below.

Fig. 4. Von misses stress

The maximum equivalent stress occurs at the end near the fixed constraints at 115.3 MPa, then the minimum equivalent stress is 0 MPa.

3.2 Displacement

Displacement or deformation is the main result of static structural analysis using FEA. The representation of the results of the analysis of the total deformation in the model is shown in Figure 5.

According to the simulation results, the maximum total deformation occurs in the center of the shaft, which is far from the fixation constraints of 0.0106 mm, while the smallest total deformation occurs in the part close to the fixation constraints, which is 0 mm. The representation of the total deformation is shown in Figure 5.

Fig. 5. Displacement

3.3 Safety Factor

A factor of safety is one of the important parameters in determining whether a design is feasible or not. The safety factor is the relationship between the permissible material stress and the stress that occurs. The shaft is declared safe if the security number is greater than one.

Fig. 6. Safety Factor

Figure 6 shows that when the shaft is loaded with meat weighing 10 kg, the greatest stress occurs at the end of the shaft near the base, with the minimum safety factor = 1.8 ul

Table 1. Result Von misses stress

Load	Von misses stress	
	min	max
W ₁	0 Mpa	8,515 Mpa
W ₂	0 Mpa	61,89 Mpa
W ₃	0 Mpa	115,3 Mpa

Table 2. Result Displacement

Load	Displacement	
	min	max
W ₁	0 mm	0,0106 mm
W ₂	0 mm	0,07715 mm
W ₃	0 mm	0,1437 mm

Table 3. Result Safety Factor

Load	Safety Factor	
	min	max
W ₁	15 ul	15 ul
W ₂	3,34 ul	15 ul
W ₃	1,8 ul	15 ul

W₁ = Empty shaft weight
 W₂ = Shaft weight plus 5kg of meat
 W₃ = Shaft weight plus 10kg of meat

Table 2 shows that the maximum load is the addition of meat weighing 10 kg, where the maximum

safety factor = 15 ul was obtained, while the minimum safety factor = 1.8 ul. Based on the safety factor information from Dobrovolsky (machine element) for static loads: 1.25 - 2; dynamic load: 2-3; Impact load 3 - 5. The shaft belongs to the dynamic load category, so the safety number is at least 2, so the shaft is not safe to take a maximum load of 10 kg. Shafts can be used safely with a maximum load of 5 kg.

4 Conclusions

The conclusions from the above discussion are:

1. The shaft construction of a meat cutting machine made of S50C steel with a maximum tensile strength of 62 kg / mm² can withstand a heavy weight of 5 kg of meat
2. The more the load is given, the more the value of false stresses and deformations occurs.
3. The construction is considered to be safe, as the specified permissible stress of 8.92 kg / mm² is greater than the shear stress of 5.62 kg / mm² according to mathematical calculations.
4. The shaft safety factor for a 5 kg meat load is 3.34 ul. In the safe category, the standard for dynamic loads established by Dobrovolsky (machine element) refers to 2-3

References

1. T.A ISKANDAR CHANDRA, *Simulasi pembebanan gaya berat pada mill shaft roll shell di pabrik gula sei semayang dengan metode elemen hingga*. Medan, (2009).
2. D. C. Lee and C. S. Han, "CAE (computer aided engineering) driven durability model verification for the automotive structure development," *Finite Elem. Anal. Des.*, (2009).
3. F. A. Conle and C. C. Chu, "Fatigue analysis and the local stress-strain approach in complex vehicular structures," *Int. J. Fatigue*, (1997).
4. B. Setyono and S. Gunawan, "Perancangan Dan Analisis Chassis Mobil Listrik " Semut Abang " Menggunakan Software Autodesk Inventor Pro 2013," *Semin. Nas. Sains dan Teknol. Terap. III*, (2015).
5. J. M. Saddam Jahidin, "Rancang Bangun 3D Konstruksi Kapal Berbasis Autodesk Inventor untuk Menganalisa Berat Konstruksi," *J. Tek. Pomits*, vol. 2, no. 1, (2013).
6. A. P. Mahardika, "Introducing Autodesk Inventor," *isometriview* <http://isometriview.wordpress.com/2011/12/08/introducing-to-autodesk-inventor/>, (2011).
7. M. D. Pinem, *Analisis Struktur dengan Metode Elemen Hingga (Finite Element Method)*. Bandung: Rekayasa Sains, (2010).
8. Cipto, K. A. Rahangmetan, F. Sariman, Hariyanto, S. B. Bahar, and M. T. Sapsal, "Stress analysis on the frame of peanut skin peeler using autodesk inventor software," in *IOP Conference Series: Earth and Environmental Science*, (2019).

9. B. Setyono, “Perancangan Dan Analisis Kekuatan Frame Sepeda Hibrid ‘Trisona’ Menggunakan Software Autodesk Inventor,” *J. IPTEK*, vol. **20**, no. 2, p. 37, (2016).