

# 18974

*by* Azara Vigha Sisliana

---

**Submission date:** 17-Jun-2023 10:14PM (UTC+0700)

**Submission ID:** 2117815819

**File name:** 18974\_Turnitin.docx (1.34M)

**Word count:** 2771

**Character count:** 13305

## STRENGTH ANALYSIS OF THE WUXI TUNNEL SHAFT USING THE FINITE ELEMENT METHOD

### Abstract

The Wuxi Tunnel is a machine for producing mochi ice cream from China. One of the most important components in the ongoing production is the shaft. A shaft is a stationary rotating part, usually of a circular cross-section, to which elements such as gears, pulleys, cranks, sprockets, and other rotational transfer elements are attached. The load received by the shaft comes from the product and materials. The load was too heavy and worked continuously resulting in the shaft breaking 3 times and not being straight. The purpose of this research is to analyze the shaft to determine the type of material and recommended dimensions so that the strength of the shaft is maintained, and to determine the stress that occurs on the shaft due to the load from the product and other materials. The research method used in this study is the finite element method using Autodesk Inventor Pro software and manual calculations so that later the results of the type of material and dimensions suitable for the shaft will be used. The analysis results show that the shaft can withstand loads at a diameter of 50 mm on the type of material AISI 4340 Annealed. The von Mises result for manual calculations is 294.2578 MPa and the von Mises result for finite elements is 275.5 MPa. The allowable stress is 470 MPa. So that AISI 4340 material with a recommended large diameter of at least 50 mm is a safe shaft limit that can be used at PT. X because the von Mises value is lower than other types of materials and a safety factor of 1.71 is more than  $>1$ .

### 1. Introduction

The Wuxi Tunnel is one of the machine parts for producing mochi ice cream originating from China. Inside is a tunnel that divides the right and left sides into 2 parts - each part contains Shaft, Sprocket, Gear Box, Gears, pulley, and plate. Cleaning is always carried out every week to keep it hygienic.

A shaft is a rotating member which transmits power [1]. The shaft is one of the most important parts of any machine [2]. The bending moment in the shaft will generate shear stress [3]. Fatigue (Fatigue) is the tendency of a metal to break when subjected to dynamic stress and repeated (Cyclic stress) case in Figure 1 [4].

Three-dimensional modeling using the finite element method can be used to determine the distribution of torsional moment, von Mises, and maximum shear stress that occurs in the shaft [5]. Finite Element Analysis (FEA) involves simulating physically engineered structures using a numerical technique called FEM [6]. The finite element method is a numerical mathematical technique for calculating the structural strength of engineering components by dividing objects into mesh shapes [7] [8]. MEH is one of the most versatile numerical methods for solving problems in the continuum domain [9]. The finite element computation was performed using SolidWorks and Inventor Standard [10].

The safety factor is a factor of uncertainty safe factor using Autodesk Inventor simulation is calculated as the yield strength of the material divided by the maximum von Mises stress of the

material [11].

The von Mises stress is the resultant x, y, and z stresses about the three axes [12]. Materials or materials can be said to yield when the von Mises stress reaches the yield strength or critical value [13]. If the shaft is made of a ductile material, the maximum shear stress used is the shear stress that occurs as a result of a combination of bending and twisting moments a fairly steep stress gradient will arise along the cross-section of the specimen, making measurement difficult [14]. The torsional stress that occurs is directly proportional to the strain [15].

The stress concentration in the keyway when torque is transmitted through the key is the difference in the maximum stress for torsional loading [16]. The value of Shear Stress is the shear stress which results in a twisting moment for deformation [17]. The key material must be softer (have less creep strength) than the shaft and propeller material [18]. It is important to have an adequate radius on the edge of the lock path [19].

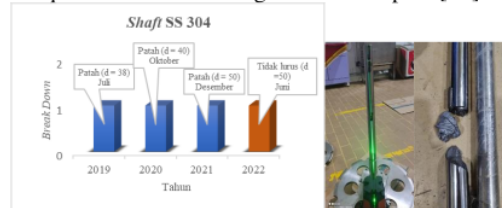


Fig 1. SS 304 Shaft History Chart and SS 304 Shaft Actual

Based on previous research regarding the investigation of stresses on multilevel shafts with the

CAD-based finite element method [20]. The drawback of this research is that the analysis only uses CAD, so the results of the analysis cannot be said to be accurate. To further ensure the accuracy of the CAD results, you can complete manual calculation comparisons according to the data needed.

In this study, the authors wanted to analyze the strength of the shaft using the finite element method based on Software Inventor Pro 2019 with a comparison of manual calculations. In this analysis, the assumptions determine the alternative types of materials and dimensions used.

## 2. Experimental and Procedures

The data in Table 1 needed is data from production material loads, namely the product and the plate that accommodates the product.

Table 1. Product and Material Total Weight Data

Information	Value	Unit	Quantity
Product	45	mL	25 Pcs/Plat
Plat	2,4	Kg	46 Pcs/Cycle

Load Calculation: Number of Plates per cycle x number of products in each plate

### 2.1 Shaft Strength Calculation with Manual Calculation Method

From the survey results, the actual operating conditions are the quantity a machine produces. Analyze using manual calculations based on the actual conditions of PT. X where the results will later be compared with the analysis of the results from using the Autodesk Pro 2019 Inventor Software with a maximum difference of 10%, along with the formula and actual operating conditions and manual calculations:

#### a. Calculation of Load and Force

Based on the data on the force generated from the working load with the (1) [21]:

$$F = m \cdot a \quad (1)$$

Where :

F = Force (Newtons)

m = Mass ( kg )

a = Acceleration of gravity ( m / s )

#### b. Calculation of Resistance Moment

Calculation of the lightening moment on the shaft will later be required to obtain the tensile stress in each section [21], calculated using the following (2):

$$W = \frac{\pi}{32} * d^3 \quad (2)$$

Where:

W = Moment Resistance (N.mm)

D = Diameter of Cross Section (mm)

#### c. Calculation of Torsional Resistance

Calculation of torque and torsional resistance is required to obtain the shaft stress value due to torsion loads [21], with the following calculations,

- *Torque*

Calculation of the torque that occurs is calculated based on the following (3):

$$\text{Torsi}_{actual} = \frac{\text{Daya (Watt)} * 60}{2 \text{ phi} * \text{Speed Train (rpm)}} \quad (3)$$

- *Torsion resistance*

Calculation of torsional resistance calculated on the shaft [21], using the following formula (4) :

$$W = \frac{\pi}{16} * d^3 \quad (4)$$

Actual operational condition:

$$V = 139,9 \text{ V}$$

$$I = 13,03 \text{ A}$$

Assumption:

$$V = 150 \text{ V}$$

$$I = 13 \text{ A}$$

It is known that the maximum power per motor that occurs is 1950 watts, at a shaft speed of 4.5 Rpm.

#### d. Calculation of Shear Stress

The shear stress value is the shear stress caused by the twisting moment for deformation. The load received by the shaft comes from only 1 force of torque, namely the torsional moment where the load received has been calculated in kWh, so the shear stress value is the same as the von Mises value.

- Shear stress due to torsion,

Calculation of shear stress due to torque and keyways is calculated using the following (5) and (6) [17]:

$$h = h = \frac{d}{4} \quad (5)$$

Where:

b = Width of keyways (mm)

h = Height or thickness of keyways (mm)

l = length of keyways (mm)

d = Diameter of Shaft (mm)

$$\tau = \frac{2M_{\tau}}{dbl} \quad (6)$$

Where:

$\tau$  = Shear Stress (N.mm)

$M_t$  = Torque transmitted (N.mm)

## 2.2 Analysis Flowchart with Autodesk Inventor Pro 2019 Software and Process Analysis Software Autodesk Inventor Pro 2019

The research process was carried out to analyze the strength of the shaft using the Autodesk Inventor Pro 2019 Software. Starting from making sprocket and shaft bearing hubs, inputting material properties, assembling parts, and providing pedestals, loads, torque, gravity, and meshing.

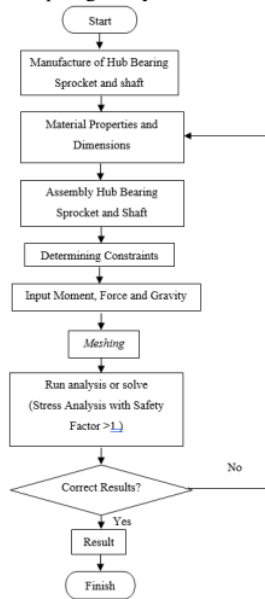


Fig 2. Analysis Flowchart with Autodesk Inventor Pro 2019 Software

The analysis process is carried out after the geometry data and material properties of the shaft, Hub Bearing Sprocket, and Shaft are known. The following shows 2D detail of the shaft and Hub Bearing Sprocket in Figure 3

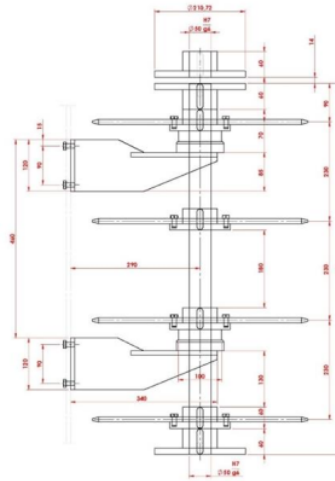


Fig 3. 3D Hub Bearing Sprocket with AutoCAD 2D 2019

The modeling step in the Autodesk Inventor Pro 2019 software is known as 3D design, namely by making a 3D design in the Autodesk Inventor Pro 2019 software with the results of the drawing in the ipt format, entering material properties, assembling between parts in the Autodesk Inventor Pro 2019 software with format results assembly is iam, input constraints, and loading, meshing process, running program. After the 3D design stage, the next stage is the analysis stage, where a running simulation is carried out using simulations. Then the next step is the result, namely the reading of the results in the results section.

The steps for making a 3D design and the analysis process are explained in the following section.

1. Manufacture of Hub Bearing Sprocket and shaft  
Geometry modeling is done using Autodesk Software Inventor Pro 2019, based on detailed drawings in the field. By going to file-new-select the .ipt file type. The results of making a 3D design are shown in the following Figures 4 and 5

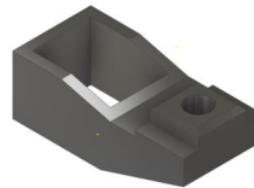


Fig 4. 3D Hub Bearing Sprocket with Software Autodesk Inventor Pro 2019



Fig 5. 3D Shaft with Software Autodesk Inventor Pro 2019

## 2. Input Material Properties and Dimensions

The material properties entered are in accordance with the material specifications of the Shaft.

## 3. Assembly Part

At this stage, the process of merging the Hub Bearing Sprocket and the shaft is carried out. With the stages of entering the file-new assembly-choose the file type iam, click constrain select which part is this in the assembly click type, mate, after give the required distance click ok, as shown in Figure 6 following.



Fig 6. 3D Assembly Hub Bearing Sprocket and Shaft with Autodesk Inventor Pro 2019 Software

## 4. Determining Constraints and Loading

Constrain serves as a fulcrum, where the constraint used is a fixed constraint and is placed at the end of the Sprocket Bearing Hub. By clicking constrain-select fix constrain-specify the surface for the constrain fix.



Fig 7. Determination of Constraints in Autodesk Inventor Pro 2019 Software

As for the loading that is included, namely the loading on the neck of the shaft, and also the torque from the motor, it also includes the provision of gravity with a value of 9.8.

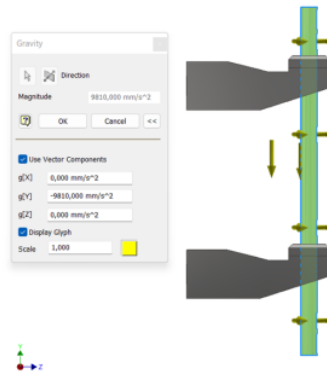


Fig 8. Giving Gravity to Autodesk Inventor Pro 2019 Software

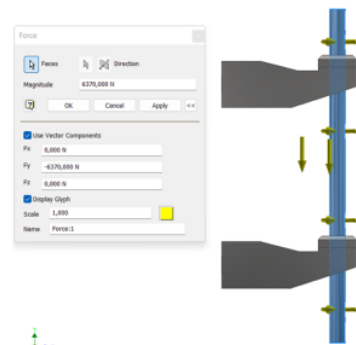


Fig 9. Giving Force to Autodesk Inventor Pro 2019 Software

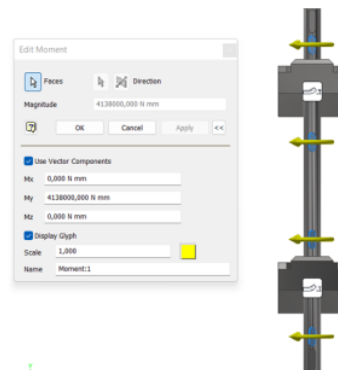


Fig 10. Giving Moment to Autodesk Inventor Pro 2019 Software

## 5. Meshing

After loading and giving constraints, then meshing is done. In this study, the average data element size uses 0.015.

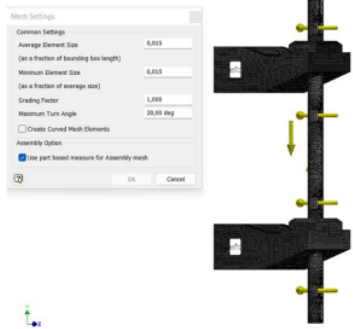


Fig 11. Meshing View to Autodesk Inventor Pro 2019 Software

### 6. Running a Job or simulating

Running is done by going to solve—clicking simulate—run as shown in Figure 11. Wait until the simulation process is complete and the analysis status becomes complete. And the analysis is complete.



Fig 12. Running Job and Simulate to Autodesk Inventor Pro 2019 Software

## 3. Results and Discussion

### 3.1 Result of Analysis Using Manual Calculation

The results of manual calculations with the resulting force of 6370 N with a moment of 4138 Nm and the results of calculating the moment of resistance, torsional resistance, and shear stress on shaft diameters of 38 mm, 40 mm, and 50 mm are in the following table:

Table 2. The results of manual calculations

Diameter	Prisoner Moment	Torsion Resistance	Shear Stress
mm	mm <sup>3</sup>	mm <sup>3</sup>	MPa
38	5384,315	10768,63	509,4490
40	6280	12560	459,78
50	12265,625	24531,25	294,2578

### 3.2 Result of Analysis Using the Finite Element Method

Shaft analysis using the finite element method with Autodesk Inventor Pro 2019 software, the load or force entered is a force value of 6370 N for a diameter of 38 mm, 40 mm, and 50 mm. With a torque value of 4138000 N.mm, accompanied by the force of gravity. In addition, input material properties AISI 304, AISI 4130, and 4340.

The simulation results on the strength of the shaft show that the maximum von Mises stress is 496.3 MPa.

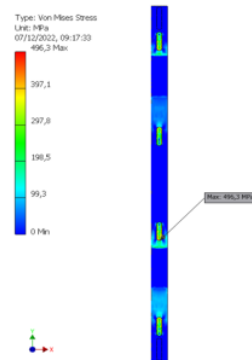


Fig. 13. Finite Element Analysis Results of Diameter 38 AISI 4130

The simulation results on the strength of the shaft show that the maximum von Mises stress is 459.78 MPa.

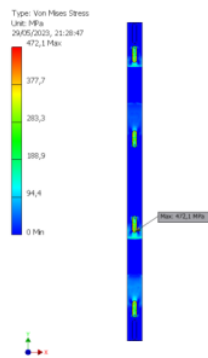


Fig. 14. Finite Element Analysis Results of Diameter 40 AISI 4130

The simulation results on the strength of the shaft show that the maximum von Mises stress is 287.7 MPa.

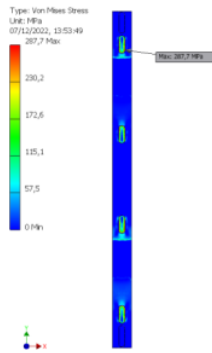


Fig. 15. Finite Element Analysis Results of Diameter 50 AISI 4130

The simulation results on the strength of the shaft show that the maximum von Mises stress is 492,9 MPa.

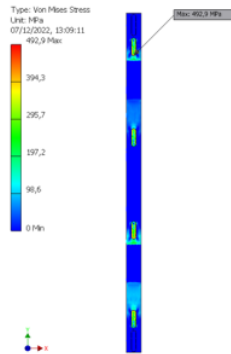


Fig. 16. Finite Element Analysis Results of Diameter 38 AISI 4340

The simulation results on the strength of the shaft show that the maximum von Mises stress is 460,9 MPa.

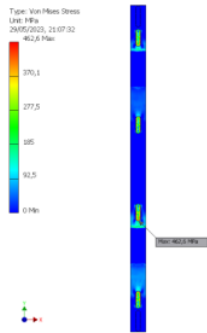


Fig. 17. Finite Element Analysis Results of Diameter 40 AISI 4340

The simulation results on the strength of the shaft show that the maximum von Mises stress is 275,5 MPa.

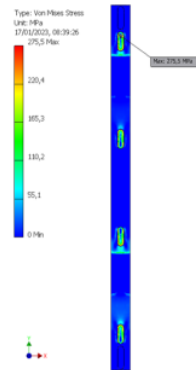


Fig. 18. Finite Element Analysis Results of Diameter 50 AISI 4340

The simulation results on the strength of the existing shaft show that the maximum von Mises stress is 320,2 MPa.

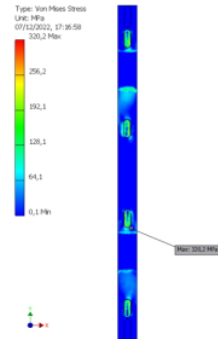


Fig. 19. Finite Element Analysis Results of Diameter 50 AISI 304

### 3.3 Result of Analysis Manual Calculation and Using the Finite Element Method

Shaft strength analysis manually and finite element method were carried out on diameters of 38 mm, 40 mm, and 50 mm with the recommendation of AISI 4130 and AISI 4340 materials and a comparison of shaft strength was carried out to determine the best material.

Table 3. Comparison of Theory Shaft Strength with FEA

Diameter	Von Mises	Von Mises	Von Mises	Von Mises	Precentage	Precentage
	Shear Stress Teori	FEA 4130	FEA 4340	FEA 304 (Existing)	Beda (Teori & FEA) 4130	Beda (Teori & FEA) 4340
mm	MPa	MPa	MPa	MPa	%	%
38	509,4490	496,3	492,9	-	2,58	3,25
40	459,78	472,1	462,6	-	2,6	0,6
50	294,2578	287,7	275,5	320,2	2,23	6,37

Table 4. Results of the Analysis of Safety Factor Theory and FEA

Diameter	Safety Factor	Safety Factor	Safety Factor	Safety Factor
	Teori	Teori	FEA	FEA
	4130	4340	4130	4340
<b>mm</b>				
38	0.87	0.95	0.88	0.95
40	0.94	1.01	0.92	1.02
50	1.51	1.55	1.6	1.71

In Figure 19. Below, the results of shaft strength analysis using finite element method calculations with AISI 4130, AISI 4340, and AISI 304 (Existing) materials are given.

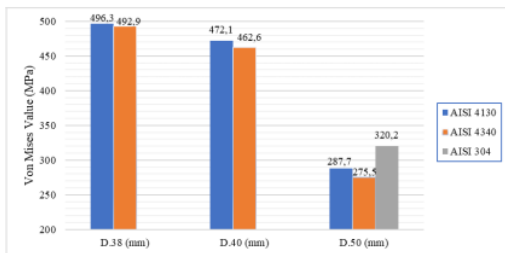


Fig. 20. Graph of Shaft Strength Analysis Results

Results of comparison of manual calculation analysis with the finite element method. The values obtained in the manual calculation of the AISI 4130 material with the lowest safety factors are 0.87 and 0.95 for a diameter of 38 mm with von Mises 509.4490 MPa and 40 mm with von Mises 459.78 MPa. In AISI 4340 material with the lowest safety factor, namely 0.95 in diameter 38 with von Mises 509.4490 MPa. For the results of the analysis using the finite element method, it was found that the AISI 4130 material with the lowest safety factor was 0.88 and 0.92 for a diameter of 38 mm with a von Mises of 496.3 MPa and a diameter of 40 mm with a von Mises of 472.1 MPa and for AISI material 4340 with the lowest safety factor which is 0.95 at 38 mm in diameter with von Mises FEA results is 492.9 MPa. In the Existing Material, the analysis results were obtained with FEA AISI 304 with a von Mises 320.2 MPa. Where in this condition, it can be interpreted that AISI 4130 material in diameters 38 and 40 and AISI 4340 in diameter 40 can cause damage to the strength of the Wuxi tunnel shaft or cannot withstand loads due to von Mises stress exceeding the allowable stress and safety factor less < 1.

So, it can be concluded that the material that can be recommended to PT. X, namely AISI 4340 Annealed at 50 in diameter with von Mises 275.5 MPa with a safety factor of 1.71 more than > 1 where the von

Mises value is smaller than the allowable stress which is 470 MPa and lower in value than the existing material with von Mises 320.2 MPa.

#### 4. Conclusions

The results of the analysis of the FEA calculation on AISI 304 material with a von Mises value of 320.2 MPa, this value exceeds the allowable stress of 215 MPa so that the shaft is broken and not straight because it cannot withstand a load of 650 kg working continuously and safe materials that can last a long time are used, namely AISI 4340 in diameter 50 with a von Mises value of 275.5 MPa below the allowable stress of 470 MPa with a safety factor of 1.71 more than >1. So that the material can be recommended to PT X.

18974

---

ORIGINALITY REPORT

---

5%

SIMILARITY INDEX

4%

INTERNET SOURCES

2%

PUBLICATIONS

2%

STUDENT PAPERS

---

PRIMARY SOURCES

---

1

[ejournal.undiksha.ac.id](http://ejournal.undiksha.ac.id)

Internet Source

3%

2

[karya.brin.go.id](http://karya.brin.go.id)

Internet Source

1%

3

Submitted to Segi University College

Student Paper

1%

---

Exclude quotes Off

Exclude matches < 17 words

Exclude bibliography Off