International Journal of
Innovation in Mechanical Engineering & Advanced Materials (IJIMEAM)
https://publikasi.mercubuana.ac.id/index.php/ijimeam

Vol.5 (No.1). 2023. pp. 1-8
Published online: April 1, 2023
ISSN: 2477-541X

STRENGTH ANALYSIS OF A WUXI TUNNEL SHAFT USING FINITE ELEMENT METHOD

A. V. Sisliana[1], D. Romahadi[1,2], and M. Imran[3]*

[1]Department of Mechanical Engineering, Universitas Mercu Buana, Jakarta11650, INDONESIA
[2]School of Mechanical Engineering, Beijing Institute of Technology, Beijing 100811, CHINA
[3]School of Material Science and Technology, Beijing Institute of Technology, Beijing 100811, CHINA

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.

Keywords: Wuxi Tunnel, Shaft, Finite Element Method

*Corresponding author: Tel. +62 21 5840815 Ext. 5200
E-mail address: dedik.romahadi@mercubuana.ac.id

DOI: 10.22441/ijimeam.v5i1.18974

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, gearbox, 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 is the tendency of a metal to break when subjected to dynamic stress and repeated (cyclic stress) cases in Fig.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. The safety factor was calculated using Autodesk Inventor simulation as the material's yield strength divided by the material's maximum von Mises stress [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 was made of a ductile material, the maximum shear stress used is the shear stress that occurs due to a combination of bending and twisting moments; a fairly steep stress gradient will arise along the specimen's cross-section, 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

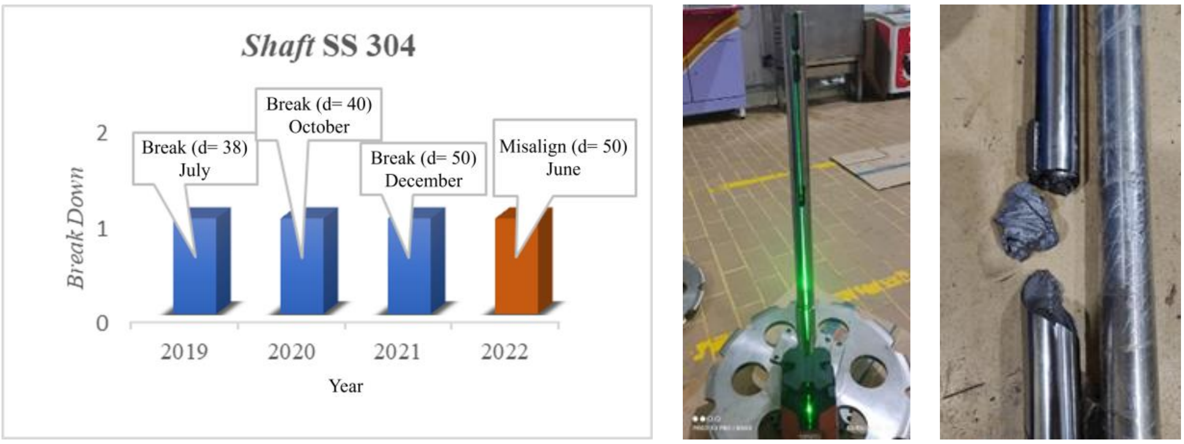


Fig. 1. SS 304 shaft history chart and SS 304 shaft actual

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 essential to have an adequate radius on the edge of the lock path[19].

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 Autodesk Inventor Pro 2019 Software 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.

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 Inventor Software with a maximum difference of 10%, along with the formula and actual operating conditions and manual calculations:

Table 1. Product and material total weight data

Information	Value	Unit	Quantity
Product	45	ml	25 pcs/plate
Plate	2,4	kg	46 pcs/cycle

a. *Calculation of Load and Force*

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

$$F = ma \qquad (1)$$

where F is force in Newtons, m is mass in Kg, and a is acceleration of gravity in 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 Eq. (2):

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

where W is moment resistance in N.mm, and D is diameter of cross section in 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 Eq. (3):

$$Torsion_{actual} = \frac{Power\ (Watt) \times 60}{2\,\pi \times Train\ Speed\ (rpm)} \qquad (3)$$

- *Torsion resistance*

Calculation of torsional resistance calculated on

Vol.5 (No.1). 2023. pp. 1-8
Published online: April 1, 2023
ISSN: 2477-541X

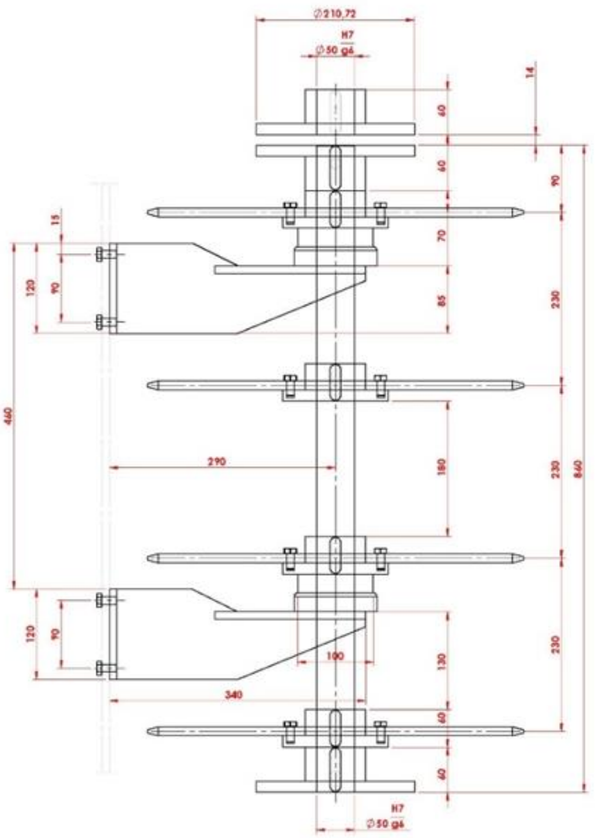


Fig. 2. 3D hub bearing sprocket with AutoCAD 2D 2019

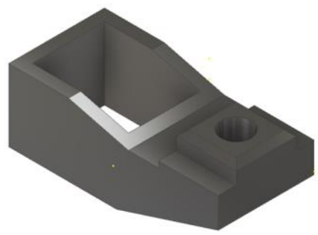


Fig. 3. 3D Hub Bearing Sprocket with Software Autodesk Inventor

Fig. 4. 3D Shaft with Software Autodesk Inventor

the shaft [21], using formula in Eq. (4):

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

Actual operational condition:

V = 139,9 V

I = 13,03 A

Assumption:

V = 150 V

I = 13 A

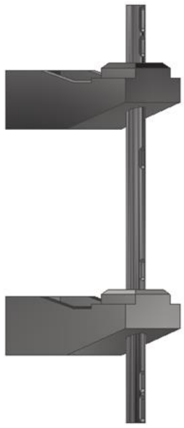


Fig 5. 3D assembly hub bearing sprocket and shaft with Autodesk Inventor Software

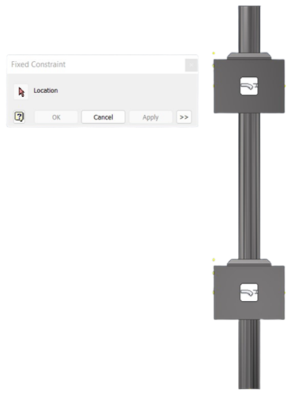


Fig. 6. Determination of constraints in Autodesk Inventor software

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 Eqs. (5) and (6) [17]:

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

where b is width of keyways in mm, h is height or thickness of keyways in mm, l is length of keyways in mm, and d is diameter of shaft in mm.

Vol.5 (No.1). 2023. pp. 1-8
Published online: April 1, 2023
ISSN: 2477-541X

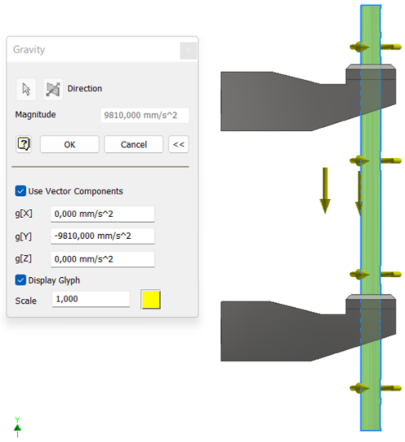


Fig. 7. Giving gravity to Autodesk Inventor software

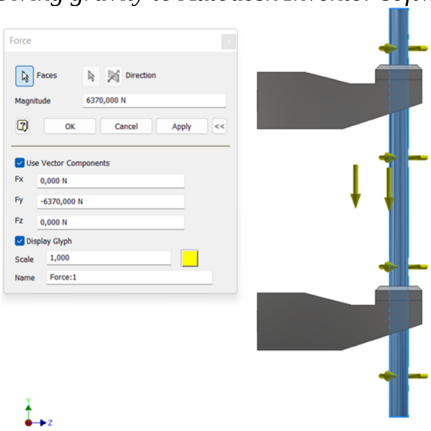


Fig. 8. Giving force to Autodesk Inventor software

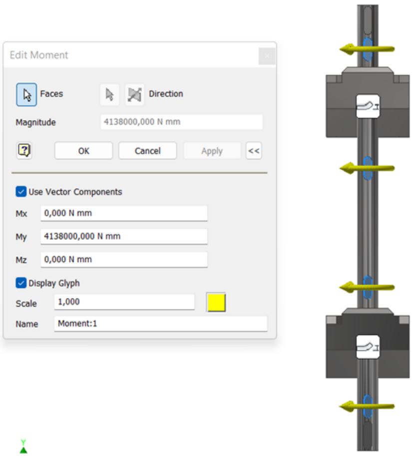


Fig. 9. Giving moment to Autodesk Inventor software

$$\tau = \frac{2M_\tau}{dbl} \tag{6}$$

where τ is shear stress in N.mm, and M_τ is transmitted torque in N.mm.

2.2 Analysis Process in Autodesk Inventor Software

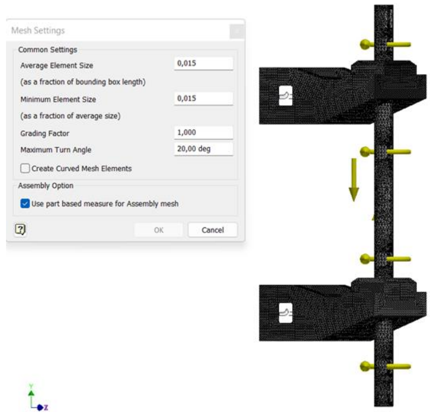


Fig. 10. Meshing view to Autodesk Inventor software

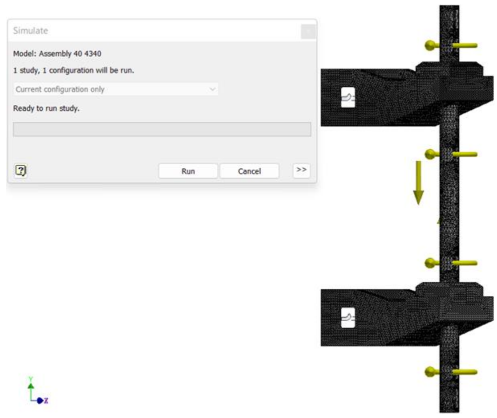


Fig. 11. Running job and simulate to Autodesk Inventor software

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

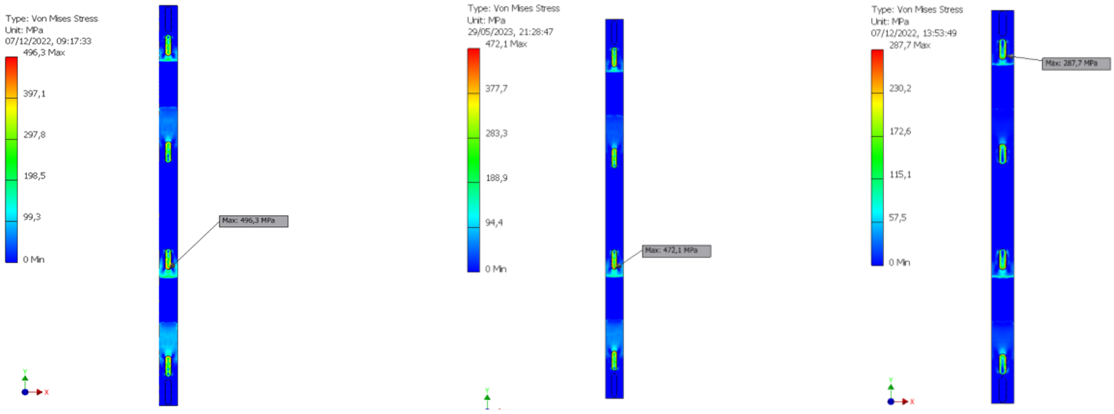
The analysis process is carried out after the geometry data and material properties of the shaft, hub bearing sprocket, and shaft are known. The following shows 2D detail of the shaft and hub bearing sprocket in Fig. 2.

The modeling step in the Autodesk Inventor software is known as 3D design, namely by making a 3D design in the Autodesk Inventor software with the results of the drawing in the IPT format, entering material properties, assembling between parts in the

Table 2. The results of manual calculations

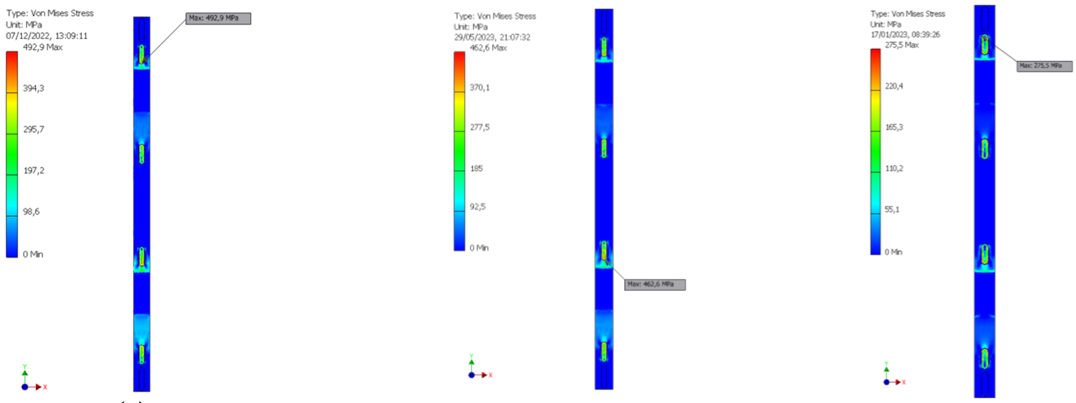
Diameter mm	Prisoner Moment mm^3	Torsion Resistance mm^3	Shear Stress MPa
38	5384.315	10768.63	509.4490
40	6280	12560	459.78
50	12265.625	24531.25	294.2578

Vol.5 (No.1). 2023. pp. 1-8
Published online: April 1, 2023
ISSN: 2477-541X



(a) *(b)* *(c)*

Fig. 12. Finite element analysis results of AISI 4130 at various diameter: (a) 38 mm, (b) 40 mm, (c) 50 mm



(a) *(b)* *(c)*

Fig. 13. Finite Element Analysis Results of AISI 4340 at various diameter: (a) 38 mm, (b) 40 mm, (c) 50 mm

Table 3. Comparison of theory shaft strength with FEA

Diameter	*Von Mises* = *Shear Stress* Theoretical	*Von Mises FEA* 4130	*Von Mises FEA* 4340	*Von Mises FEA* 304 *(Existing)*	Theoretical vs FEA Comparison 4130	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

Autodesk Inventor 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, 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 Fig. 3 and 4.

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, select constrain select which part is this in the assembly select type, mate, after give the required distance, as shown in Fig. 5 following.

Vol.5 (No.1). 2023. pp. 1-8
Published online: April 1, 2023
ISSN: 2477-541X

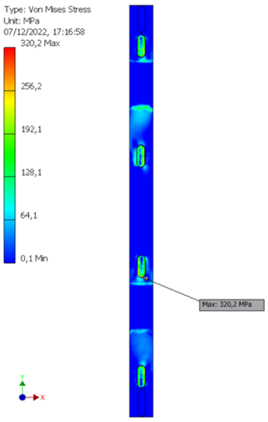


Fig. 14. Finite element analysis results of diameter 50 AISI 304

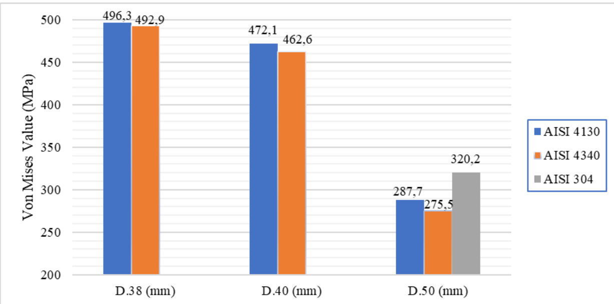


Fig. 15. Graph of shaft strength analysis results

Table 4. Results of the analysis of theory and FEA safety factor

Diameter mm	***Safety Factor* Theoretical *4130***	***Safety Factor* Theoretical *4340***	***Safety Factor FEA 4130***	***Safety Factor FEA 4340***
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

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 selecting constrain-select fix, constrain-specify the surface for the constrain fix.

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

5. Meshing

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

6. Running a job or simulating

Running is done by selecting simulation as shown in Fig. 10. Wait until the simulation process is complete and the analysis status becomes complete, and finally the analysis is complete.

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 Table 2.

3.2 Result of Analysis Using the Finite Element Method

Shaft analysis using the finite element method with Autodesk Inventor 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 AISI 4130 shaft show that the maximum von Mises stress is 496.3 MPa, 459.3 MPa, and 287.7 MPa for a diameter of 38 mm, 40 mm, and 50 mm, respectively.

Thus, the simulation results on the strength of the AISI 4130 shaft show that the maximum von Mises stress is 492,9 MPa, 460,9 MPa, 275,5 MPa for a diameter of 38 mm, 40 mm, and 50 mm, respectively.

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

3.3 Result of Analysis Theoretical 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.

Fig. 15 below shows the results of shaft strength analysis using finite element method calculations with AISI 4130, AISI 4340, and AISI 304 (existing) materials. 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, the 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. 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.

References

[1] Teja, P. K., Moorthy, G., & Kaliappan, S. (2018). Finite element analysis of propeller shaft for automotive and naval application. *International Research Journal of Automotive Technology (IRJAT)*, 1(1), 08-12.

[2] Awali, J., & Asroni, T., (2013). Analisa kegagalan poros dengan pendekatan metode elemen hingga. *Turbo*, 2(2), 39-44.

[3] Sigarlaki, H. H., Tangkuman, S., & Arungpadang, T., (2015). Aplikasi metode elemen hingga pada perancangan poros belakang gokar listrik. *Jurnal Online Poros Teknik Mesin*, 4(2), 104-115.

[4] Fitri, M., (2020). Pengaruh beban lentur pada poros stainless steel terhadap siklus kegagalan fatik. *Jurnal Teknik Mesin*, 9(3), 149-155.

[5] Yulianto, T., & Ariesta, R. C., (2019). Analisis kekuatan shaft propeller kapal rescue 40 meter dengan metode elemen hingga. *Kapal: Jurnal Ilmu Pengetahuan dan Teknologi Kelautan*, 16(3), 100-105.

[6] Saputra, G. J., & Romahadi, D., (2023). Analisis komparasi kekuatan geometri desain rusuk penguat pada kursi plastik menggunakan computer aided engineering. *Jurnal Teknik Mesin*, 12(1), 20-30.

[7] Romahadi, D., Ruhyat, N., & Dorion, L. B. D. (2020). Condensor design analysis with kays and london surface dimensions. *Sinergi*, 24(2), 81.

[8] Walidina, M. F., Kardiman, & Gusniar, I. N. (2022). Analisis tegangan von mises pada poros mesin penggiling sekam padi menggunakan software Ansys. *Mettek*, 8(1), 35-43.

[9] Isworo, H., & Ansyah, P. R. (2018). *Buku ajar metode elemen hingga HMKB654*. Universitas Lambung Mangkurat. https://mesin.ulm.ac.id/assets/dist/bahan/MEH_Buku_Ajar_full.pdf

[10] Afolabi, S. O., Oladapo, B. I., Ijagbemi, C. O., Adeoye, A. O. M., & Kayode, J. F. (2019). Design and finite element analysis of a fatigue life prediction for safe and economical machine shaft. *Journal of Materials Research and Technology*, 8(1), 105–111.

[11] Wibawa, L. N., & Diharjo, K. (2019). Desain, pemilihan material, dan faktor keamanan stasiun pengisian gawai menggunakan metode elemen hingga. *Jurnal Teknologi*, 11(2), 97-102.

[12] Ismail, R., Sugiyanto, Kristianto, H., Saputra, E., & Jamari. (2017). Pemodelan metode elemen hingga kontak femoral head dengan acetabular liner pada sendi panggul buatan dengan variasi diameter celah pada acetabular liner. *Rotasi*, 19(3), 139-146.

[13] Mulyanto, T., & Sapto, A. D. (2017). Analisis tegangan von mises poros mesin pemotong umbi-umbian dengan software solidworks. *Presisi*, 18(2), 24-29.

[14] Huzni, S. (2021). Analisis numerik kekuatan puntir baja karbon rendah menggunakan software (Solidworks). *Jurnal REM (Rekayasa Energi Manufaktur)*, 6(2), 29-36.

[15] Sufiyanto (2006). Kaji empirik kekuatan puntir baja ST 42 dengan variasi jumlah siklus fatigue. *Transmisi*, 2(2), 203-212.

[16] Sahu, M. K. (2014). Optimization of the keyway design with consideration of effect of stress concentration on different materials. *Int. J. Eng. Res*, 3(5), 477-481.

Vol.5 (No.1). 2023. pp. 1-8
Published online: April 1, 2023
ISSN: 2477-541X

[17] Prajapati, H. R., Patel, B. P., & Patel, N. V. (2015). investigation of stress concentration factor for keyway on shaft under different loading conditions: A case study. *University Journal of Research*, 1(1), 45-57.

[18] Suyadi (2016). Perhitungan kekuatan spi poros pada pengujian model kapal. *Wave: Jurnal Ilmiah Teknologi Maritim*, 10(1), 13-18.

[19] Pedersen, N. L. (2010). Stress concentrations in keyways and optimization of keyway design. *The Journal of Strain Analysis for Engineering Design*, 45(8), 593-604.

[20] Chandra, H., Putra, D. P., & Romli (2021). Investigasi tegangan pada poros bertingkat menggunakan metode elemen hingga berbasis computer aided engineering. *Jurnal Austenit*, 13(1), 1-5.

[21] Khurmi, R. S., & Gupta, J. K. (2005). *A textbook of machine design*. S. Chand Publishing.