JMM: Journal of Mechanical and Manufactur
Vol. 1 No 2, April 2022 , pp. 106-112
DOI:
e-ISSN: 2829-4424

THE TORQUE TEST SPECIMENS ANALYSIS USING FINITE ELEMENT ANALYSIS

Slamet Riyadi [1], Tia Setiawan[2], Engkos Koswara[3]
[1,2]Teknik Mesin Universitas Galuh Ciamis
[3]Teknik Mesin Universitas Majalengka
[1] slametriyadi.cms@gmail.com
[2] tiasetiawan405@gmail.com

Abstract
The industrial component of industrial equipment is required to have good quality. The elements or components of a tool must be light but have strong and corrosion-resistant alloys. Testing of the materials to be used must be appropriate and in accordance with the needs. There are several ways, one of which is by using a torque testing machine and Finite Element Analysis (FEA). Finite Element Method (FEM) or commonly called Finite Element Analysis (FEA), is a numerical procedure that can be used to solve certain engineering problems. Modeling torsion test objects using the FEM with a moment 598.0635 N mm, - 598.0635 N mm with respect to vector X counterclockwise. The material used is ASTM A 36 with a mass of 0.0163569 kg and a volume of 2083.94 mm^3, torsion test analysis using finite element-based software. Based on the simulation results, it can be seen that the largest stress (maximum stress) occurs in the XZ vector with a value of 10.4783 MPa, while the smallest stress (minimum stress) occurs towards the YZ vector with a value of 0.15986 MPa. The largest strain (maximum strain) occurred in vector XZ with a value of 0.000068123 ul, while the smallest strain (minimum strain) occurred in vector XX with a value of 0.00000107672 ul. The safety factor obtained is 0.71, meaning that the value is less than equal to 1 which indicates that the test object is unable to accept the given load.

Keywords: *stress analysis, torsion test specimens, finite element analysis, safety factor*

Submitted: 2022-04-01	Revised: 2022-04-01	Accepted: 2022-04-10

Introduction

Technological advances in the industrial sector of industrial equipment components are required to be of high quality. The elements or components of a tool must be light but have strong and corrosion-resistant alloys. Testing of the materials to be used must appropriate with the materials needed, in which there are several ways, such as, by using a torque testing machine and FEA (Finite Element Analysis). Finite Element Method (FEM) or usually called Finite Element Analysis (FEA), is a numerical procedure that can be used to solve engineering problems.

The advantage of using Finite Element Analysis (FEA) in testing materials as a form of simulation is that this approach is much cheaper than experimentation, with FEA we can reduce the number of experiments and in certain cases we even can eliminate experiments altogether. Autodesk Inventor is a product of CAD after AutoCAD and Autodesk Mechanical Desktop, Autodesk Inventor has several advantages that make it easier for drafters to design and look more attractive and real with the materials provided as similar to the original material.

Torque test tool is a tool designed to measure how much torsional force is, it is commonly used by industry for measuring and obtaining torsional strength data of a material, so that the standard you want to know can be accepted, when testing a specimen by continuously twisting the test rod. continuously until the test rod breaks or reaches the specified amount of torsion.

According to Hayat and Kurniawan, the main loading is torsion that occurs in the machine element or the combined load of the shaft strength must be known using a torsion test tool.

Meanwhile, stress analysis is one of the structural testing tools on the Autodesk inventor which is carried out by applying the FEA concept (Setyono, 2016).

Facts in the field testing of steel specimens using a torsion testing machine have been carried out at the Lab of Universitas Galuh. Mechanically, using a torque testing machine so that further discussion is needed, which for further testing can use FEA where the test is in the form of software.

Research Method

The research method used is specimen testing using software, the material used is ASTM A36 with material properties can be seen in the following table:

Table 1. material specification

Name	Steel ASTM A36	
General	Mass Density	7,84905 g/cm^3
	Yield Strength	248,225 MPa
	Ultimate Tensile Strength	399,9 MPa
Stress	Young's Modulus	199,959 GPa
	Poisson's Ratio	0,3 ul
	Shear Modulus	76,9073 GPa

Table 2 force

Load Type	Moment
Magnitude	598,0635 N mm
Vector X	-598,0635 N mm
Vector Y	0,000 N mm

The applied force is in the opposite direction to the vector x input force of 598,0635 N mm.

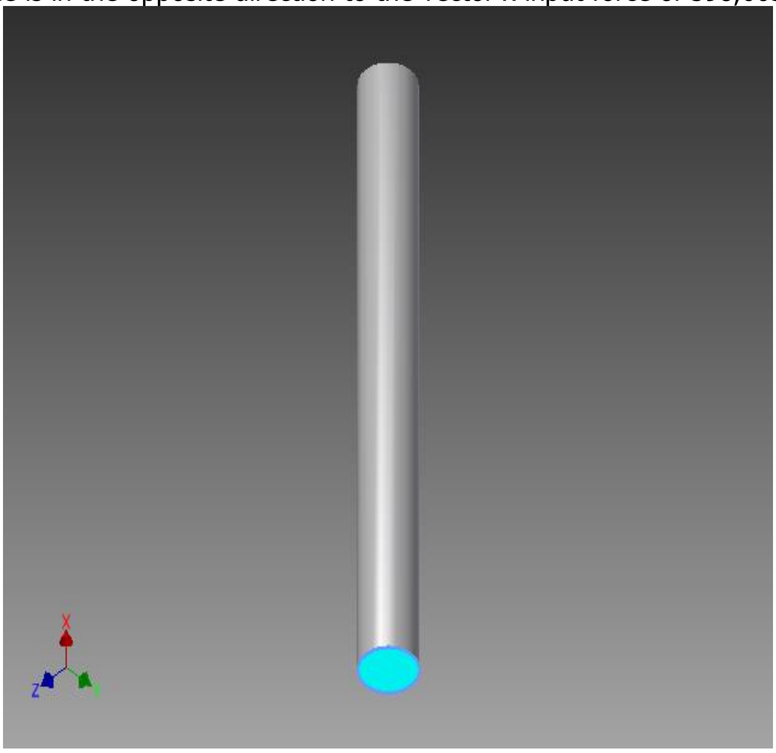

Figure 1 *constrain type*

The fix constrain is where the item locks in the seat so it cannot move.

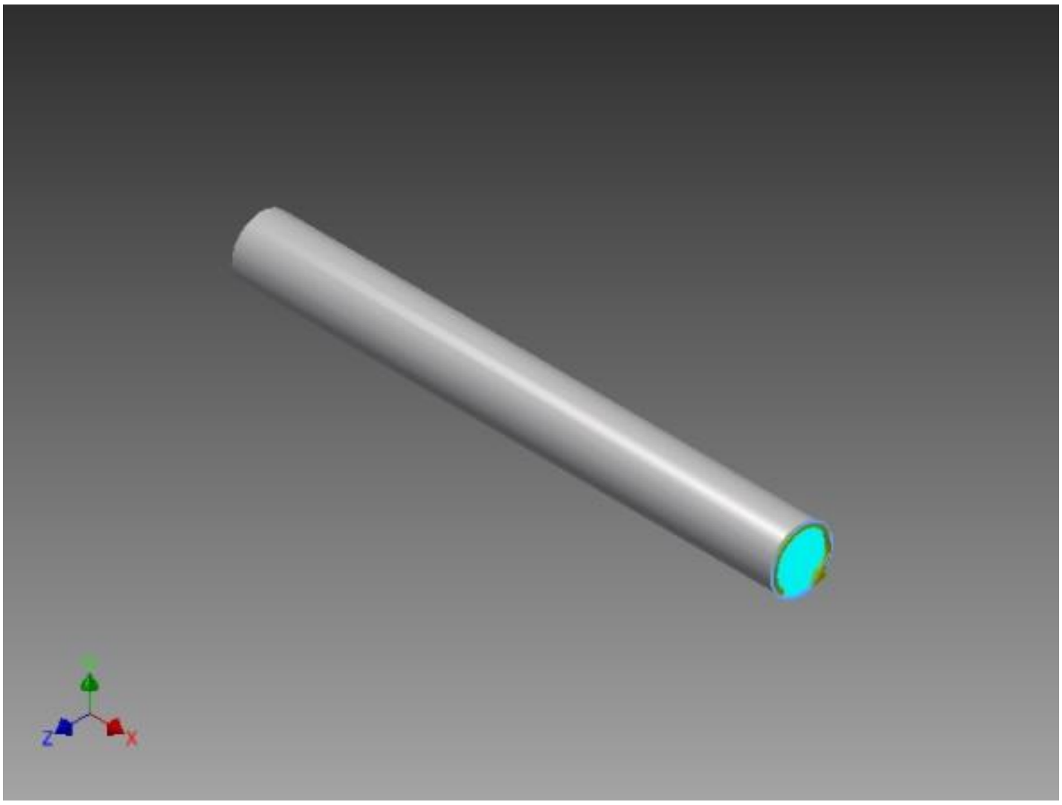

Figure 2 *load type*

Results and Discussion

With *feture generate report* provided, a summary of the simulation results is obtained in the form of tables and images.

Table 3 result summary

Volume	2083,94 mm^3	
Mass	0,0163569 kg	
Von Mises Stress	0,349175 MPa	18,4326 MPa
1st Principal Stress	0,139924 MPa	10,8642 MPa
3rd Principal Stress	-10,6069 MPa	-0,262975 MPa
Displacement	0 mm	0,00808475 mm
Safety Factor	0,0135 ul	0,7109 ul
Stress XX	-0,187931 MPa	0,271168 MPa
Stress XY	-10,591 MPa	10,395 MPa
Stress XZ	-10,406 MPa	10,4783 MPa
Stress YY	-0,753437 MPa	0,781902 MPa
Stress YZ	-0,325131 MPa	0,15986 MPa
Stress ZZ	-0,571581 MPa	0,3333 MPa
X Displacement	-0,000000975621 mm	0,00000053471 mm
Y Displacement	-0,00808093 mm	0,00808184 mm
Z Displacement	-0,00808146 mm	0,00808031 mm
Equivalent Strain	0,00000152487 ul	0,0000798919 ul
1st Principal Strain	0,00000120661 ul	0,0000700844 ul
3rd Principal Strain	-0,0000689209 ul	-0,00000141277 ul
Strain XX	-0,000000848977 ul	0,00000107672 ul
Strain XY	-0,0000688555 ul	0,0000675812 ul
Strain XZ	-0,0000676532 ul	0,000068123 ul
Strain YY	-0,00000338705 ul	0,00000368495 ul
Strain YZ	-0,00000211379 ul	0,0000010393 ul
Strain ZZ	-0,00000242999 ul	0,00000147788 ul

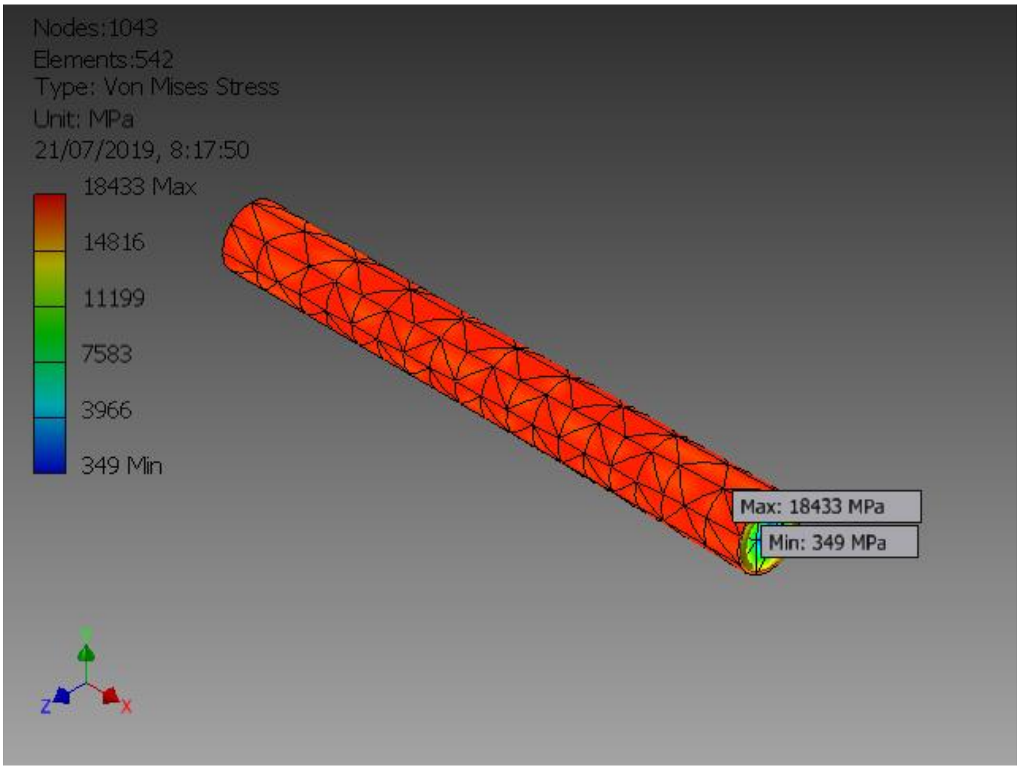

Figure 3 von mises stress

Figure 3 shows that the von Mises stress occurs in the test specimen. Von Mises stress is used to predict the yield level of the material under loading conditions from the test results. The material begins to yield when the von Mises stress reaches a critical value that needs to be known as yield strength.

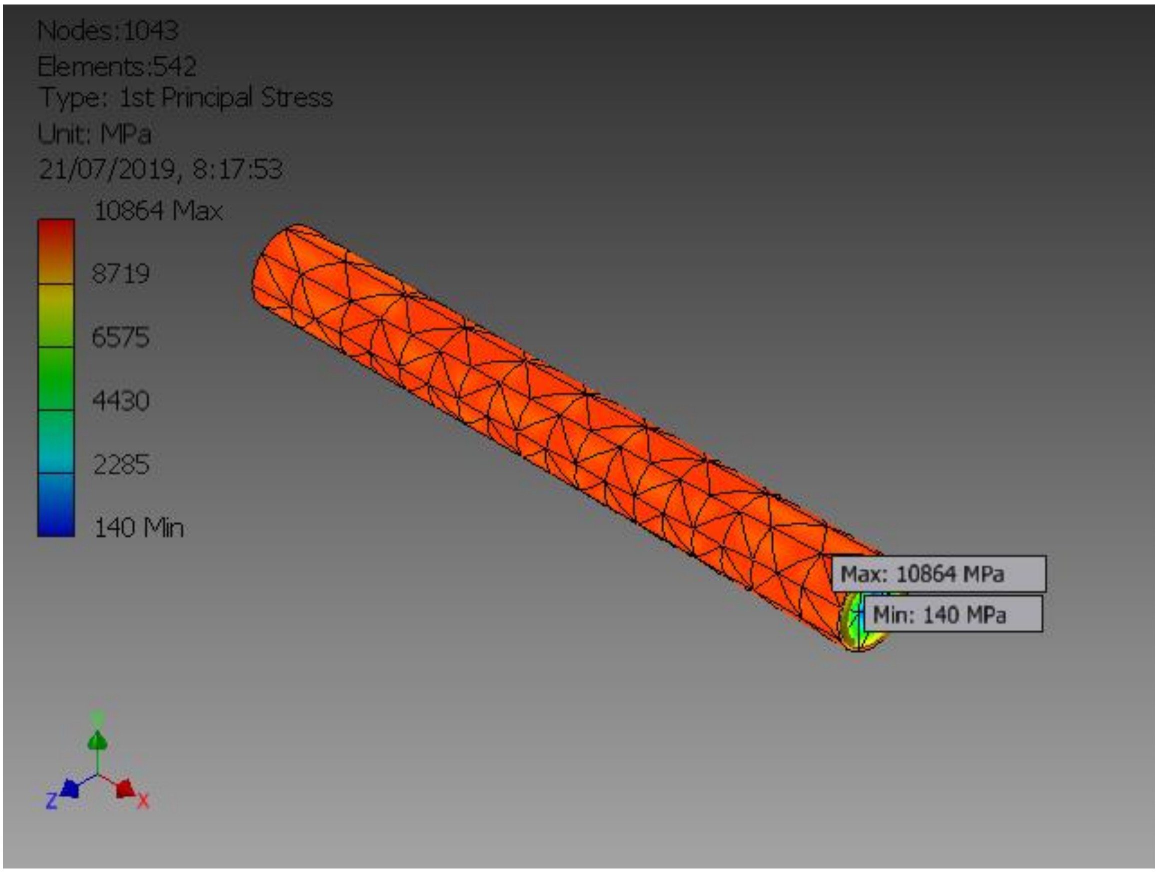

Figure 4 1st Principal Stress

The figure 4 shows a breakdown of the maximum stress that specifically shows the tensest part, which is the part in red.

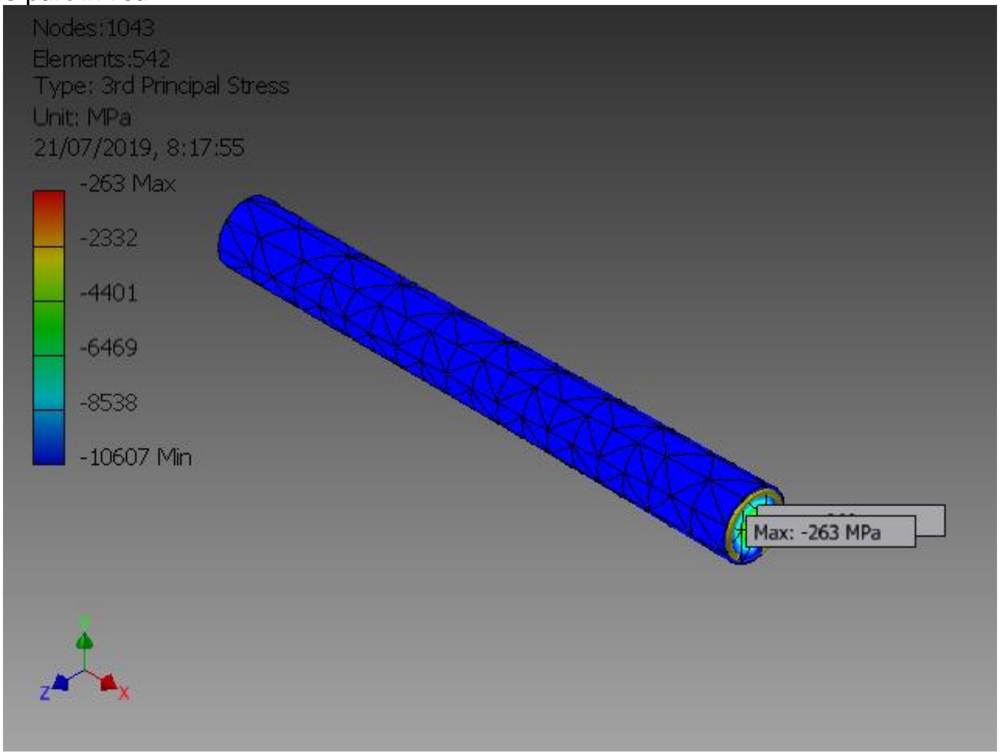

Figure 5 3rd Principal Stress

Figure 5 shows the minimum principal stress which specifically the most relaxed part.

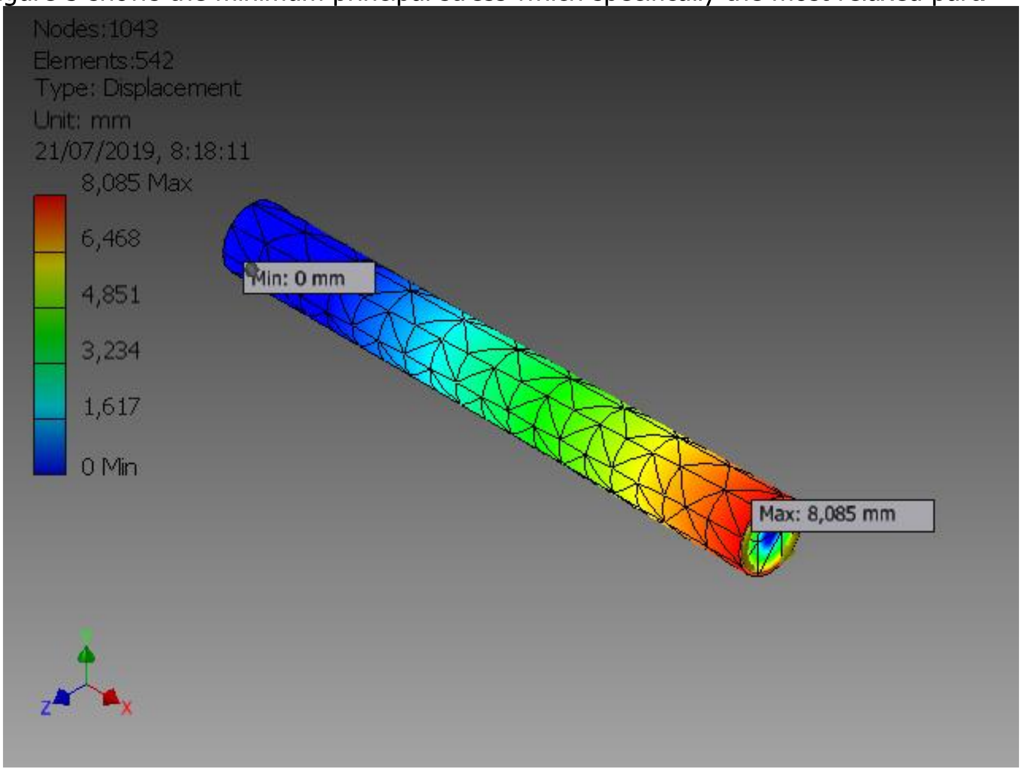

Figure 6 displacement

Figure 6 shows the displacement or movement that occurs in the specimen. The red color on the specimen indicates movement in that direction

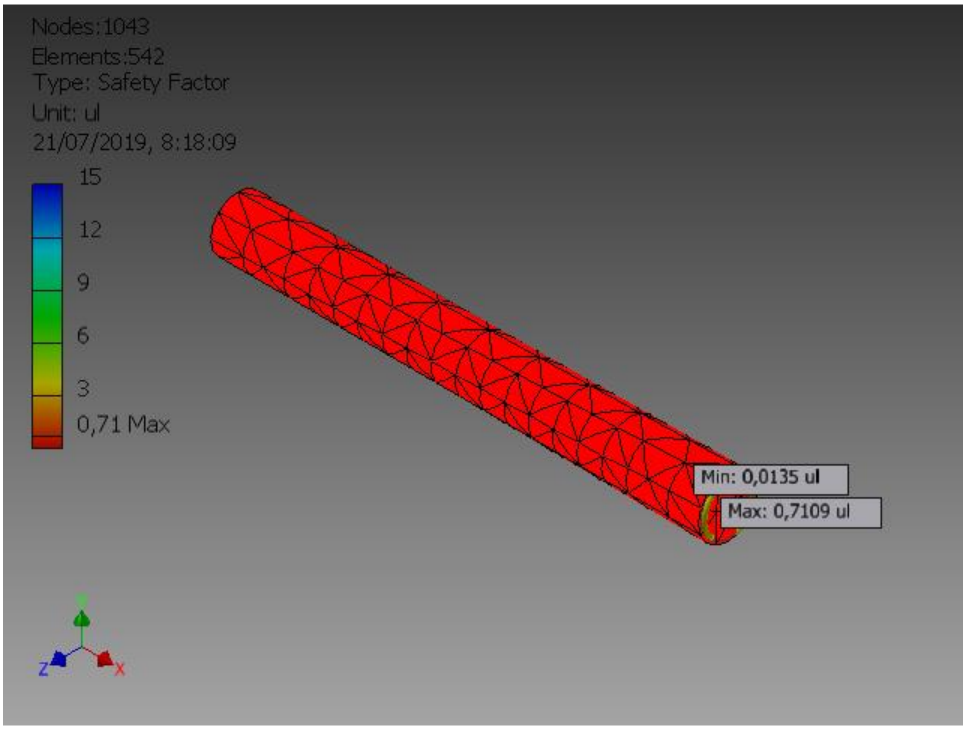

Figure 7 *safety factor*

The safety factor shows the state of the specimen in red. It shows that the specimen is unsafe and the safety factor value ≤ 1 means it is unsafe. In the figure 7 above shows the test specimen is red and the factory safety value is 0.7109 ul, it means that the value ≤ 1, indicating that the specimen is unable to accept the given load.

Conclusion

Based on the results of torsion test using the torsion test tool, the standard specimen for torsion test is ASTM, the specimen changes dimensions after the torsion test is carried out due to improper installation of the specimen, the torque moment (Mt) obtained is 598,0635938 with a twist amount of 8,784, angle torsion (rad) 55,16352 and the modulus of shear elasticity (G) is 202.51384 Gpa.

Further, based on the stress analysis simulation results that have been carried out, it is showed that the greatest stress (maximum stress) occurs towards the XZ vector with a value of 10.4783 MPa, while the smallest stress (minimum stress) occurs towards the YZ vector with a value of 0.15986 MPa.

Then, the largest strain (maximum strain) occurred towards vector XZ with a value of 0.000068123 ul, while the smallest strain (minimum strain) occurred towards vector XX with a value of 0.00000107672 ul.

Meanwhile, the safety factor obtained is 0.71, which means that the value is less than 1, indicating that the specimen is unable to accept the given load.

Reference

Bambang Setyono, Abdul Hamid, Jurnal Teknik Mesin Perancangan Dan Analisis Kekuatan Frrame Sepeda Hibrid "Trisona Menggunakan Software Autodesk Inventor

Ewa Suganda, Rahmawaty,ST, MT, Jurnal Teknik Mesin Uji Kekuatan Puntir Material Stanless Steel Dengan Perlakuan Panas Dan Tanpa Perlakuan Panas

G. Nieman, Anton Budiman, Dipi, Ing. Bambang Priambodo, Elemen mesin jilid I

Randi Achmad Fauzy 2014, Tugas Ahir Pengujian Spesimen Baja Pada Mesin Uji Torsi

Muas A.M.Anzarih,Ahmad. Cara Mudah Menggambar Mesin Melaui Konteks Riset Secara Inklusif Dengan Menggunakan Autodesk Inventor Profesional 2014

Mhd. Daud Pinem. Analisis struktur dengan metode elemen hingga (finite element method)

M. Sulung Hayat, Fadli Achmad Krniawan, ST, MT Jurnal Teknik Mesin Uji Kekuatan Puntir Material Baja St 60 Dengan Perlakuan Panas Dan Tanpa Perlakuan Panas

Pandri pandiatmi, idk okariawan, emi diah salman, jurnal teknik Pembuatan mesin uji tarik kapasitas kecil

Qomarudin jurnal teknik analisa numerik contact stress dan capsule endoscopy di dalam usus kecil
.