R.E.M. (Rekayasa Energi Manufaktur) Jurnal | Vol.7 No.1/2022
ISSN online (2528-3723)
http://doi.org/ 10.21070/r.e.m.v7i1.1629

Numerical Analysis of Low Carbon Steel Tensile Strength Using Software (SolidWorks)

Arya Rudi Nasution[1*], Edi Widodo[2]
[*]Email corresponding author: aryarudi@umsu.ac.id

[1]Department of Mechanical Engineering, Muhammadiyah University of Sumatera Utara, Indonesia
[2]Department of Mechanical Engineering, Muhammadiyah University of Sidoarjo, Indonesia

Article history: Received: 18 Oktober 2021 | Revised: 9 April 2022 | Accepted: 18 April 2022

***Abstract**. Tensile testing is the most basic material testing machine. Destructive testing is one of the tests to see the strength and toughness of the material. This test is very simple, inexpensive, and has been standardized all over the world, for example in America with ASTM E-8 and Japan with JIS 2241. This research wants to see the strength of the material using the finite element method. The finite element method is a numerical method used to solve engineering problems. For numerical analysis using the SolidWorks software. The material used is low carbon steel and the geometry is formed according to ASTM E8. To see the composition of the material used optical emission spectroscopy microscope. Experimental tensile testing uses a Universal Test Machine tensile testing machine with a maximum capacity of 50 kN. The results of experimental tests and the results of observations of the composition of the material are entered into the SolidWorks software and then perform simulations to obtain/know the magnitude of the strength value of the material. Before the simulation, each specimen geometry is distinguished/varied based on the length of the mesh. Variation of mesh 2, 4, 6, 8, and 10. From the experimental test results, the stress value obtained is 2.451E+09N/m2. The results of the numerical simulation, the stress value on the material in each variation is different, this indicates that the simulation is running well. The voltage value obtained in the experiment is not that far from the simulation results. The simulation results with 2 and 4 mesh lengths are very close to the experimental values. The larger the mesh will affect the value of the simulation results.*

***Keywords** - Tensile Test, Numeric Simulation, SolidWorks.*

INTRODUCTION

Tensile testing is the most basic materials testing machine. This test is very simple, inexpensive, and has been standardized around the world, for example in America with ASTM E-8 and Japan with JIS 2241. By pulling on a material we will know how the material reacts to tensile forces and know how far the material increases long. Experimental equipment for this tensile test must have a strong grip and high stiffness. Well-known brands for tensile testing equipment include Shimadzu[1], Instron, and Dartec [2].

The properties of carbon steel are highly dependent on carbon content, therefore carbon steels are grouped based on their carbon content. Steel with a carbon content of less than 0.3% is called low carbon steel, steel with a carbon content of 0.3%-0.6% is called medium carbon steel and steel with a carbon content of 0.6%-1.5% is called steel. high carbon [3]. Materials of low carbon steel affect the mechanical properties of materials, such as material toughness, material yield strength and material elongation [4][5]. In addition to experimental tensile testing, now many simulation studies have been carried out using multiple software that supports and investigates the mechanical properties of a metal material. [6].

The finite element method is a numerical method used to solve engineering problems such as geometry[7], loading, and the properties of very complex materials. [8][9]. The process of the finite element method is to divide a complex problem into elements to make it easier to get a solution. The solutions of each element are then combined so that they become the overall problem solution [10]. Based on references from several studies that have been carried out, the finite element method is one solution that can be done to investigate research problems related to the mechanical properties of the material.

METHOD

The material used is ST37 low carbon steel plate profile with dimensions of 1.6mm thickness. the composition of the test material can be seen in table 1. The geometry of the material refers to ASTM E8 which is formed using a machining process.

Tabel 1. Steel Composition ST37

Spessimen	C (%)	Si (%)	S (%)	P (%)	Mn (%)	Ni (%)	Cr (%)	Mo (%)	Cu (%)	Al (%)
Plat	0.189	0.0461	0.0133	0.0188	1.18	1.70	0.0449	0.0514	0.0272	0.0474

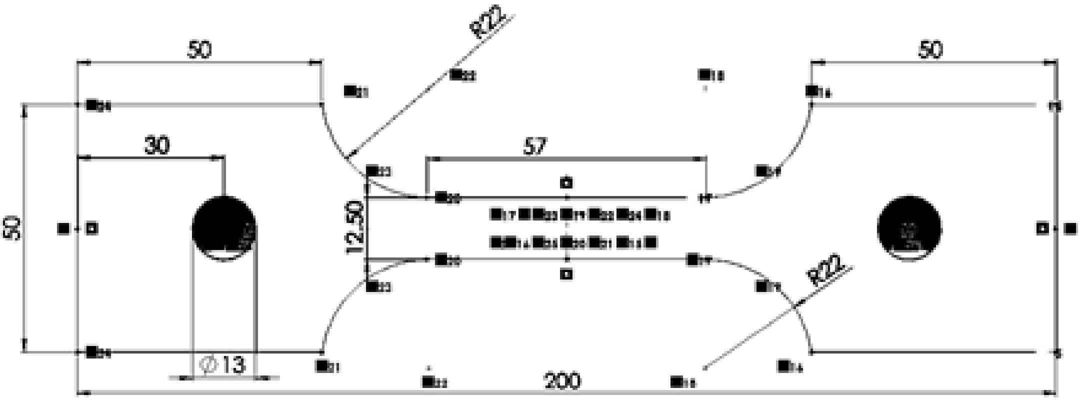

Figure 1. Material modeling geometry [11].

The process of data collection by the experimental method using a universal test machine (UTM). The experimental data results are inputted into the software for numerical investigations of the material. The application used is Solidox 2020, where then the geometric model is distinguished based on mesh variations. The variations of the mesh are shown in table 2. Each meshing of the material. The geometric shape of the plate has meshed into small parts of a triangular shape, which can be seen in Figure 2 [12].

Table 2. Mesh Variations

Element Size	Total Nodes	Total Elements
2 mm	116893	68268
4 mm	50202	27224
6 mm	34955	18746
8 mm	32795	17663
10 mm	30670	16403

The next step is to limit this numerical simulation. This limitation includes the fixed position and the position of applying static force according to the specifications of the UTM 5000N testing machine on the material shown in Figure 2.

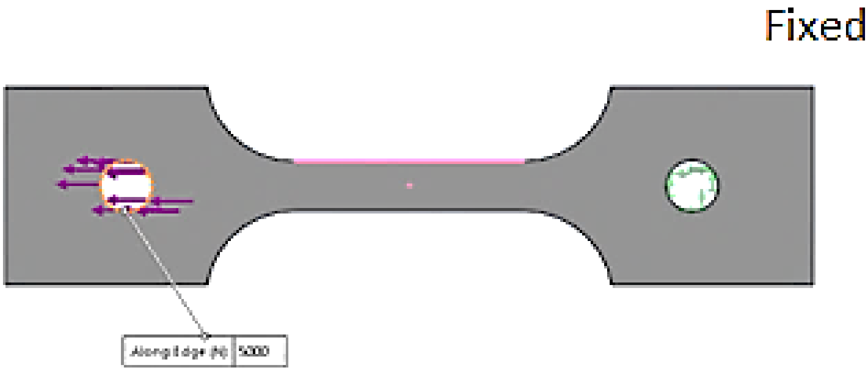

Figure 2. Fixed position and static force.

RESULTS AND DISCUSSION

A. Observation of Material Composition

The material composition was tested using optical emission spectroscopy. The test results have been presented in table 1 above. After getting the results of testing the material composition of ST37 steel, it is included in the category of low carbon steel with carbon content below 0.15%. [13], [14].

B. Hasil Tarik Eksperimental

The experimental tensile test results are 2.451E9 N/m2.

Where the formula :

Stress = $\sigma = \frac{F}{A}$

$$\sigma = \frac{49033,25}{0,00002}$$

$$= 2,451x10^{9}\ N/m^{2}$$

$$= 2,451E+09 N/m2$$

The results of the fracture from the test are shown in Figure 3. The data obtained from the experimental test results are entered into the SolidWorks application to continue the numerical test.

Figure 3. Test specimen testing (after)

C. Numerical Simulation Results

The simulation results showed in Table 3. The increase in the stress value in the SolidWorks numerical simulation in each mesh variation is not so significant.

Table 4.1. Validation of simulation and experimental stress

No	Total Node	Total Elemen	Simulation	Experiment
			Tegangan N/m²	Tegangan N/m²
1	116893	68268	2,450e+09	2,451e+09
2	50202	27224	2,452e+09	
3	34955	18746	2,455e+09	
4	32795	17663	2,458e+09	
5	30670	16403	2,460e+09	

The graph shows the voltage value from the simulation and experimental results, the stress that occurs in the experimental material is 2.451E+09N/m^2. The results of the numerical simulation with the mesh variation of the stress value are not much different from the experimental stress. In the graph, the variation of the mesh is indicated by the total element. At element 68268 the voltage is 2.450E+09N/m^2, at element 27224 the voltage is 2.452E+09N/m^2, at element 18746 the voltage is 2.455E+09N/m^2, at element 17663 the voltage is 2.458E+09N/m^2, at element 16403 the voltage is 2.460 E+09N/m^2.

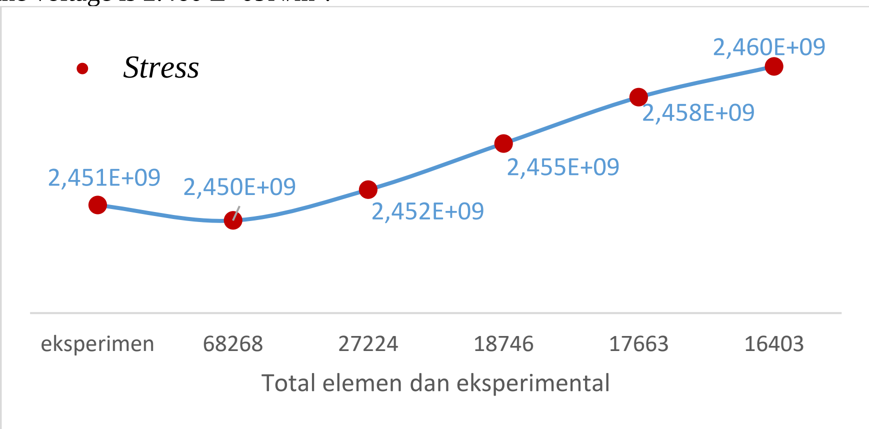

Figure 4. Experimental and simulated stress values.

The following is one of the results of the stress node points that occur in the cross-section in the fixed area. Can be seen in the image below.

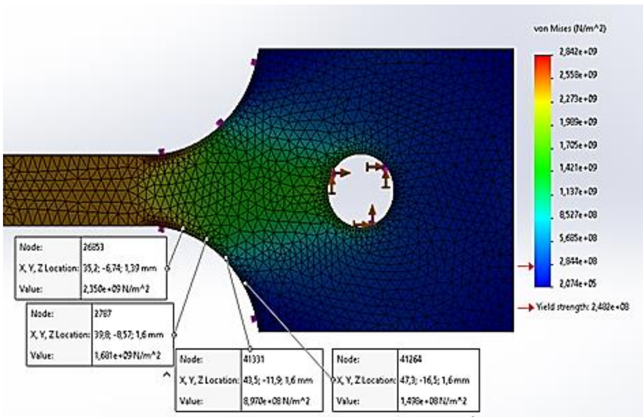

Figure 5. Stress distribution with cross-section 1 (fixed).

CONCLUSION

It can be concluded from the observations in the study that the stress value that occurs in each variation of the element is different, the more the total number of elements, the smaller the stress value. Vice versa, the fewer the total elements, the greater the voltage value. The change in the voltage value that occurs is not so large, it is still in the range of 2.45E+09N/m^2 to 2.46E+09N/m^2. The simulation results are very close to the experimental results.

ACKNOWLEDGMENTS

The authors would like to thanks the LPPM Universitas Muhammadiyah Sumatera Utara, which has funded the 2020-2021 internal grant program. No contract : Number :39/II.3-AU/UMSU-LP2M/C/2021

REFERENSI

[1] A. Y. Elnour *et al.*, "Effect of pyrolysis temperature on biochar microstructural evolution, physicochemical characteristics, and its influence on biochar/polypropylene composites," *Appl. Sci.*, vol. 9, no. 6, pp. 7–9, 2019, doi: 10.3390/app9061149.

[2] E. K. Laksanawati and A. A. Gunawan, "Pengujian Kekuatan Rig Untuk Uji Tarik Baja a36 Diameter 30 Mm Bentuk Standard Dengan Analisa Software Solidwork," *Mot. Bakar J. Tek. Mesin Univ. Muhammadiyah Tangerang*, vol. 2, no. 1, pp. 30–37, 2018.

[3] M.-A. Pourmoosavi and T. Amraee, "Low carbon generation expansion planning with carbon capture technology and coal phase-out under renewable integration," *Int. J. Electr. Power Energy Syst.*, vol. 128, p. 106715, 2021, doi: https://doi.org/10.1016/j.ijepes.2020.106715.

[4] W. Wang and V. Kodur, *Tensile test on steels at elevated temperatures*. 2020.

[5] A. R. Nasution, Affandi., and Z. Fuadi, "Pengaruh Cairan Pendingin Terhadap Kekasaran Permukaan Benda Kerja Pada Proses Face Milling," *J. Rekayasa Mater. Manufaktur dan Energi*, vol. 3, no. 1, pp. 16–22, 2019, doi: doi.org/10.30596/rmme.v3i1.4524.

[6] R. J. Devaney, P. E. O'Donoghue, and S. B. Leen, "Effect of welding on microstructure and mechanical response of X100Q bainitic steel through nanoindentation, tensile, cyclic plasticity and fatigue characterisation," *Mater. Sci. Eng. A*, vol. 804, no. July 2020, pp. 1–19, 2021, doi: 10.1016/j.msea.2020.140728.

[7] T. Long, C. Huang, D. Hu, and M. Liu, "Coupling edge-based smoothed finite element method with smoothed particle hydrodynamics for fluid structure interaction problems," *Ocean Eng.*, vol. 225, p. 108772, 2021, doi: https://doi.org/10.1016/j.oceaneng.2021.108772.

[8] H. Do Kweon, J. W. Kim, O. Song, and D. Oh, "Determination of true stress-strain curve of type 304 and 316 stainless steels using a typical tensile test and finite element analysis," *Nucl. Eng. Technol.*, vol. 53, no. 2, pp. 647–656, 2021, doi: 10.1016/j.net.2020.07.014.

[9] A. H. Ali, A. Gouda, H. M. Mohamed, M. H. Rabie, and B. Benmokrane, "Nonlinear finite elements modeling and experiments of FRP-reinforced concrete piles under shear loads," *Structures*, vol. 28, no. August, pp. 106–119, 2020, doi: 10.1016/j.istruc.2020.08.047.

[10] E. Pratama, Ardi. Premono, Agung. Syarif Syaefudin, "Pengembangan Alat Bantu Media Pembelajaran Metode Elemen Hingga Untuk Kasus Mekanika Benda Padat 2," vol. 2, no. 1, pp. 25–27, 2020.

[11] Astm E8, "ASTM E8/E8M standard test methods for tension testing of metallic materials 1," *Annu. B. ASTM Stand. 4*, no. C, pp. 1–27, 2010, doi: 10.1520/E0008.

[12] K. Umurani and T. Amri, "Desain Dan Simulasi Suspensi Sepeda Motor Dengan Solidwork 2012," *J. Rekayasa Mater. Manufaktur dan Energi*, 2018, doi: 10.30596/rmme.v1i1.2435.

[13] E. L. L. Nunes, Rafael Menezes, *Metallografy and Microstructure*. 1995.

[14] S. H. Affandi, Ahmad Marabdi Siregar, Chandra A Siregar, Arya Rudi Nasution, Iqbal Tanjung, Syarizal Fonna, "ANALISA KOROSI ATMOSFERIK BAJA KARBON RENDAH DI KECAMATAN MEDAN BELAWAN," *Multitek Indones.*, vol. 6223, no. 2, pp. 80–88, 2020.

Halaman ini sengaja dikosongkan