eISSN: 2721-5717; pISSN: 2747-2124 https://journal.unhas.ac.id/index.php/zonalaut

ZONA LAUT

JOURNAL OF OCEAN SCIENCE AND TECHNOLOGY INNOVATION

Analysis of Tensile Strength on the Welding SS400 Steel with Varying Thickness

Christoper Siburian, Destyariani Liana Putri*
Program Studi Teknik Kelautan, Institut Teknologi Kalimantan
*putridestyariani@lecturer.itk.ac.id

Abstract

Technology in the construction sector is increasingly developing and advancing at this time. The construction that we often encounter is steel construction. In steel construction, we most often encounter metal joining or welding processes. Shield Metal Arc Welding (SMAW) is a metal joining process that uses electrical energy as a source of heat and is assisted by electrodes. In this journal, I modeled the welding results using Ansys Workbench. The modeling aims to carry out a tensile test simulation to determine the tensile strength of SS400 Steel with a plate thickness of 8 mm, 10 mm, and 12 mm. This research began with creating a model in ANSYS by referring to the SNI 8389:2017 standard. After the modeling is complete, then input the materials to be used, mesh the model with a size of 2 mm, and determine the fixed support and force of 100,000 (N/m2). Then we carried out a tensile test simulation to get the normal stress and total deformation values. The largest normal stress value was obtained on the 8 mm plate, which was 3845.4 MPa and the largest total permit value was obtained on the 8 mm plate, which was 0.28271 mm. The conclusion from this test is that the smaller the cross-sectional area of a model, the higher the pressure it will experience, so it will experience higher stress and will experience cooling more easily.

Keyword: SS400 steel, welding, plate

1. INTRODUCTION

Nowadays, technology in the construction sector is increasingly developing and advancing, especially in product design. In structural steel applications, the metal joining process cannot be avoided or is often called welding. The development of welding technology provides various ways to join machine parts or structures. SMAW welding (Shield Metal Arc Welding) is a metal joining process using electrical energy as a heat source and adding electrodes as additional material.

Tensile testing is a method used to test the strength (tensile strength) of a material by applying a load along the same axis and applied slowly or quickly. The results of this test show the mechanical properties in the form of strength and elasticity of the material. Where this research aims to determine the differences in tensile strength of samples with different thicknesses. In Sofyan's research entitled analysis of the influence of mesh on the distribution of tensile stress and bending of low carbon steel plates using Solidworks software. Analysis was carried out to determine the effect of mesh on electrical stress distribution and bending using Solidworks software. Based on the results obtained after analyzing the average tensile stress depreciation value of 0.066% and the average bending stress depreciation value of 0.515%.

Therefore, in this research, an update was carried out where tensile testing was carried out on the welding results using the Ansys workbench. Where in this modeling different plate thicknesses are used, starting from 8 mm, 10 mm, and 12 mm. After modeling, we carry out tensile test simulations to obtain normal stress and total deformation values from each model to determine the differences between different thicknesses.

2. METHODOLOGY

2.1. Variable

Tabel 1. Independent Variable

Variable	Variation
Material Thickness	8 mm, 10 mm, 12 mm

Tabel 2. Dependent Variable

Variable	Variation
Material	SS400

In this research there are two variables used, namely fixed variables and independent variables. Independent variables are variables that are unrelated in the sense that they are not influenced by other variables but will influence the dependent variable. The dependent variable is a variable that is influenced by the presence of an independent variable. The following is a Independent Variable (in Table 1) and Dependent Variable (in Table 2) in this study.

2.2. Research Process

In this research, there are stages carried out, namely as follows:

1. Open the geometry module first to model the shape of the Tensile test specimen.
2. Model the geometry according to the dimensions specified in the SNI 8389:2017 standard. With a parallel length of 80 mm, transition radius of 15 mm, width of 12.5 mm, grip length of 50 mm.
3. After the geometry has been formed, then connect the geometry module with the static structural module. Then enter the required engineering data, namely structural steel and carbon.
4. The next step is to open the model in the static structural option, select edit. Then there is a geometry option to determine what materials will be used in each part of the geometry.
5. Next select the mesh and enter an element size of 2 mm. Right click on the mesh and select generate mesh to perform meshing.
6. Then right click on the static structural menu, select fixed support and click on one side of the modeling, which will be the side that will be held down during tensile testing.
7. Then in the same menu select force and click on the opposite side. Enter the tensile strength used as 10,000 N, which is where this side will be pulled during the simulation.
8. Then right click on the solution menu, select normal stress and total deformation. Next, click the solve menu to carry out a tensile test simulation and get the values for normal elastic strain and total deformation.

2.3 Meshing

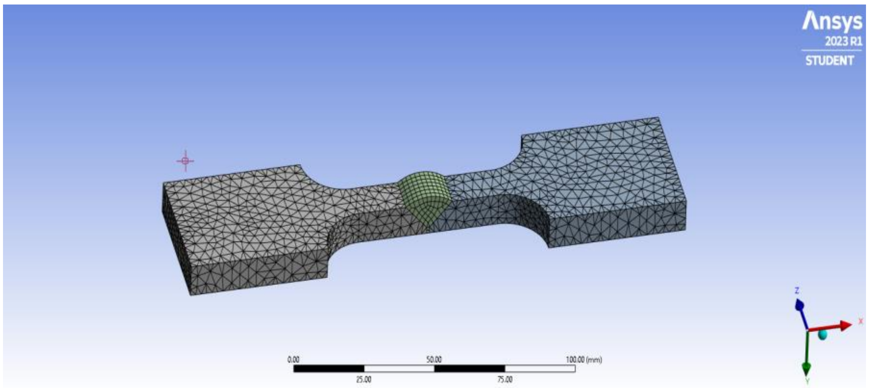

Figure 1. Meshing

The meshing stage is an important stage in an analysis. This is because the size of the meshing element will affect the results obtained in the analysis. The mesh size used for the entire model in all analyzes is 2 mm. Below (in Figure 1) is the result of meshing with a size of 2 mm.

2.4 Boundary Condition

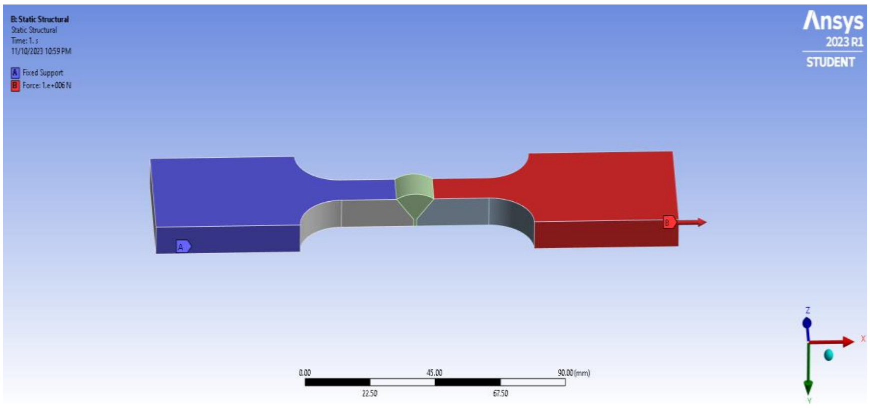

Figure 2. Boundary Condition

The analysis carried out in ANSYS also uses predetermined boundary conditions and load distribution. In this simulation, SS400 steel material is used with thickness variations of 8 mm, 10 mm, 12 mm. Before the simulation is carried out there is a meshing stage with a size of 2 mm. In this test, 2 parts are used, the first is a fixed support on the left, which is then the part that will be supported and will not change or move. Next, for the second part, namely the force on the right side, this side will change or move due to a load of 10,000 N. Figure 2 is the boundary condition in the tensile test simulation.

3. RESULT AND DISCUSSION

3.1. Tensile Test

3.1.1. Modeling Results Thickness 8 mm

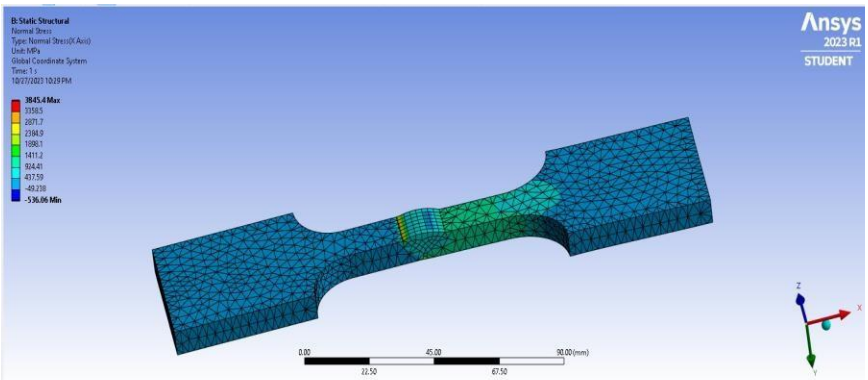

Figure 3. Normal Stress Thickness 8 mm

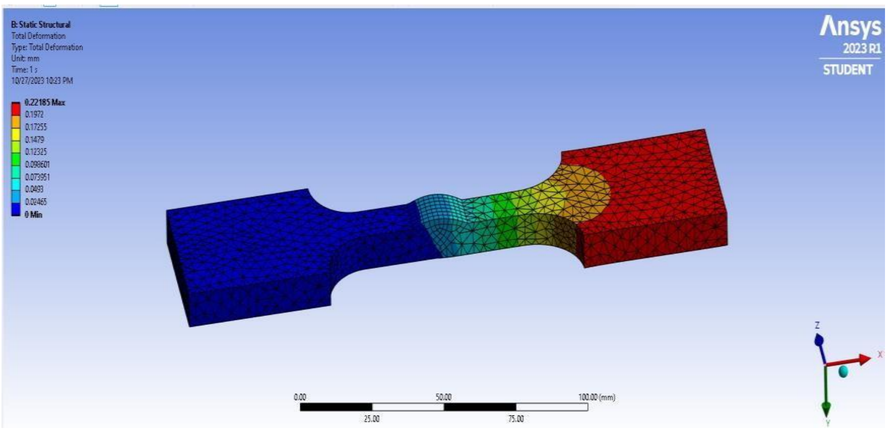

Figure 4. Total Deformation Thickness 8 mm

The tensile stress simulation results on the SS400 steel plate use a tensile strength of 10,000 (N/m2) (in Figure 3). In this way, the normal stress obtained from this tensile stress is 3845.4 Mpa. Where the initial tensile stress starts from -536.06 and ends at a stress of 3845.4 MPa. When testing the test object, a load will be applied until the test object experiences a plastic area. The shape changes that occur are caused by a combination of elastic deformation and plastic deformation and result in total deformation. The total deformation obtained in this tensile test was 0.28271 mm. (in Figure 4).

3.1.2. Modeling Results Thickness 10 mm

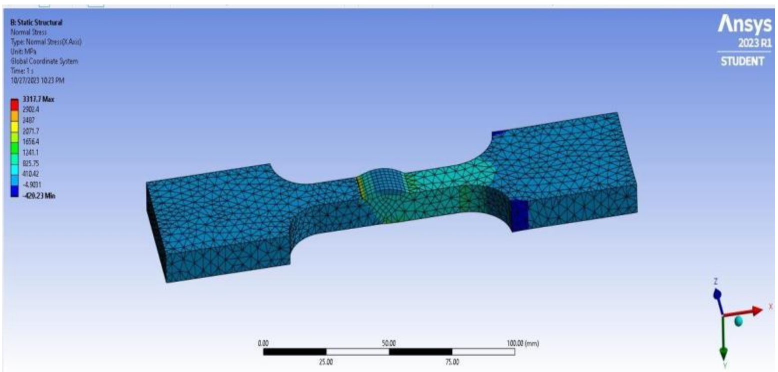

Figure 5. Normal Stress Thickness 10 mm

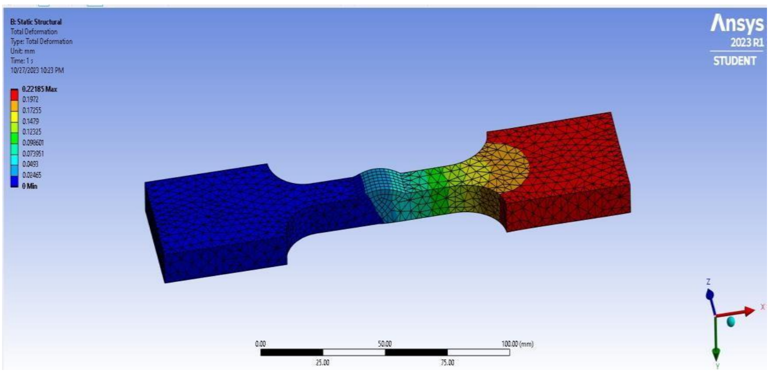

Figure 6. Total Deformation Thickness 10 mm

The tensile stress simulation results on SS400 steel plate use a tensile strength of 10,000 (N/m2) (in Figure 5). In this way, the normal stress obtained from this tensile stress is 3317.7 Mpa. Where the initial tensile stress starts from -420.23 and ends at a stress of 3317.7 MPa. When testing the test object, a load will be applied until the test object experiences a plastic area. The shape changes that occur are caused by a combination of elastic deformation and plastic deformation and result in total deformation. The total deformation obtained in this tensile test was 0.22185 mm. (in Figure 6).

3.1.3. Modeling Results Thickness 12 mm

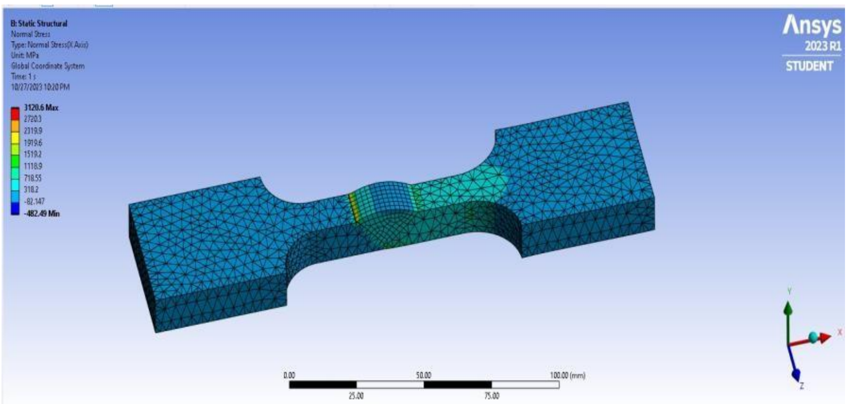

Figure 7. Normal Stress Thickness 12 mm

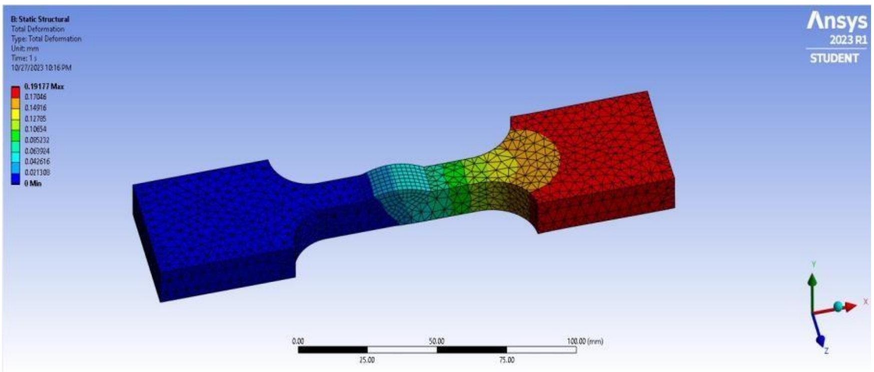

Figure 8. Total Deformation Thickness 10 mm

The tensile stress simulation results on the SS400 steel plate use a tensile strength of 10,000 (N/m2) (in Figure 7). In this way, the normal stress obtained from this tensile stress is 3120.6 MPa. When testing the test object, a load will be applied until the test object experiences a plastic area. The shape changes that occur are caused by a combination of elastic deformation and plastic deformation and result in total deformation. The total deformation obtained in this tensile test was 0.19177 mm. (in Figure 8).
Based on the results obtained from the tensile test simulation in the Ansys software, normal stress and total deformation values were obtained. For the 8 mm plate simulation, the normal stress value was 3845.4 MPa, for the 10 mm plate the normal stress value was 3317.7 MPa, and for the 12 mm plate the normal stress value was 3120.6 MPa. The largest normal stress value obtained in the 8 mm plate simulation was 3845.4 MPa because the 8 mm plate has a small cross-sectional area, so the tensile force exerted on the 8 mm plate produces high pressure which can produce higher stress. The load distribution given to the 8 mm plate can also influence high stress because it has a smaller thickness and the load distribution is more even, which causes a higher stress value compared to the 10 mm and 12 mm plates.

4. CONCLUSION

In tensile testing, the sample changes shape or experiences deformation due to the load. So in the simulation the total deformation value for an 8 mm plate is 0.28271 mm, for a 10 mm plate it is 0.22185 mm and for a 12 mm plate it is 0.19177 mm. The plate that experienced the greatest deformation occurred in the 8 mm plate at 0.28271 because this plate was thinner so it was easier to experience deformation and had a higher stress value. So it can be concluded from the tensile test simulation using Ansys that the smaller the cross-sectional area of the material, the greater the stress value obtained and the easier it is for a material to change shape or elongate.

REFERENCES

[1] Adi Nugroho, Ekos Setiawan. "Pengaruh Variasi Arus Kuat Arus Pengelasan Terhadap Kekuatan Tarik Dan Kekerasan Sambungan Las Plate Carboon Steel ASTM", Volume 3.No.2, 2018.
[2] Azwinur, Muhazir. "Pegaruh Jenis Elektroda Pengelasan SMAW Terhadap Sifat Mekanik Material SS400", Volume 17.No.2, 2019.
[3] Helanianto, Epriyandi, Hairian Rahmadi, "Pengaruh Variasi arus Pengelasan SMAW Terhadap Kekerasan Logam Induk dan Logam Las", Vol.7 No.2, 2020.
[4] Iqbal Hamdi, Taufiqullah, Herry Oktadinata, "Pengaruh Variasi Posisi Pengelasan Terhadap Distorsi dan Sifat Mekanik Hasil Pengelasan Baja SS400 Menggunakan Metode", Vol. 8, No. 1, 2020.
[5] Minto, Andhika Mayasari, Basuki, "Analisa Daerah Haz Hollow Terhadap Variasi Elektroda". Volume XXVI.No.1, 2021.
[6] Ahmad Sofyan, "Analisis pengaruh mesh pada distribusi tegangan Tarik dan bending plat baja karbon rendah dengan software solidworks", 2021.
[7] Nanda Julian, Untung Budiarto, " Analisa Perbandingan Kekuatan Tarik pada Sambungan Las Baja SS400 Pengelasan MAG Dengan Variasi Arus Pengelasan dan Media Pendingin Sebagai Material Lambung Kapal", Vol. 7, No. 4, 2019.
[8] Naharuddin, Alimuddub Sam, Candra Nugraha, " Kekuatan Tarik dan Bending Sambungan Las Pada Material Baja SM 490 Dengan Metode Pengelasan SMAW dan SAW" Vol. 6 No. 1, 2015.
[9] Vuri Ayu Setyawati, Suheni, " Variasi Arus dan Sudut Pengelasan Pada Material *Austentic Stainless Stell* 304 Terhadap Kekuatan Tarik dan Struktur Mikro " Vol. 20 No. 2, 2016.
[10] Awal Syahrani, Alimuddin Sam, Chairulnas, "Variasi Arus Terhadap Kekuatan Tarik dan Bending Pada Hasil Pengelasan SM490" Vol. 4 No. 2, 2013.