Journal of Welding Technology
P-ISSN: 2716-1471 | E-ISSN: 2716-0475
http://e-jurnal.pnl.ac.id/Welding_Technology

Residual Stress Analysis in Pipe Welding ASTM A106 Grade B Using FEM Simulation

Hamdani, Turmizi, Bukhari, Edi Saputra, Ilyas Yusuf
Politeknik Negeri Lhoksuemawe
Lhokseumawe, 24301, Indonesia
Email: hamdani_jtm@pnl.ac.id

Abstract

Fabrication technology has also penetrated the oil and gas industry, such as in the case of joining materials, one of the applications being made is the connection of pipes by welding. In the welding process, the heat distribution is uneven and the cooling rate is fast, resulting in residual stresses. This is indicated by changes in the microstructure in the weld area and the Heat Affecting Zone (HAZ), causing the material to become hard but brittle (brittle). This can cause structural failure in the connection area. This study investigates the residual stresses in pipe welding ASTM A 106 Grade B using the finite element method with ANSYS R17.2 software. The simulation is divided into 2 steps, namely thermal and structural analysis. The variation used in this simulation is the electric current as the welding heat input. The electric current is 110 A with an effective heat input of 2337.5 Watts with a maximum residual voltage of 399.59 MPa, 2762.5 Watts with a current of 130 A of 420.01 MPa, 3400 Watts with a current of 160 A of 454.06 MPa. The simulation shows that an increase in electric current in the range of 110 to 160 A can imply an increase in residual voltage. A welding heat input is obtained with the minimum residual voltage value, namely at a current of 110 A with an effective heat input of 2337.5 watts.

Keywords: residual stress, finite element method, welding simulation

1. Introduction

Technological developments in the oil and gas industry have been very advanced, including in terms of fabrication, one of the applications being made is the joining of materials to support the facilities of the oil and gas industry. In the welding process, heat energy continues which causes differences in temperature distribution in the metal, resulting in uneven expansion and shrinkage [1]. The existence of local heating due to welding and rapid cooling or high-temperature changes causes the stored energy in the weld area to be high, resulting in residual stress and distortion. High residual stress in the weld area decreases bending strength, weld strength, and fatigue life. In circular welded joints, the residual stress distribution is shown in Figure 1 [2]

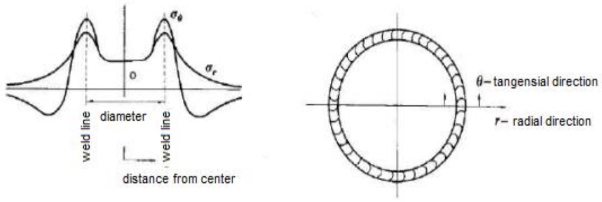

Fig. 1 The residual stress distribution in circular welding

Therefore, in the welding process to improve the welding quality must get the main attention. One of the parameters that can affect the strength of the weld is the welding current, voltage, and welding speed. The relationship between the three parameters produces welding energy which is known as heat input. Appropriate heat input can also minimize thermal contraction, and distortion in the welded product [3], [4]. The investigations and methods have been developed to reduce the residual stress in welding, including heat treating, hammering, preheating and vibration stress relieving. In some constructions, the methods and techniques for reducing residual stress are sometimes difficult, so the factor that needs to be considered to reduce residual stress is by selecting the welding cycle. The prediction of the amount of residual stress in the welding system and welding cycle is important to support increased efficiency and strength of welded joints [5], [6].

Residual stress of low carbon steel pipes in SMAW welding using practical simulations and measurements. The results show that the magnitude of the simulated stress tends to be larger, but there is still a good agreement between the simulated residual stress distribution and practical measurements [7], [8]. Using the finite element method to analyze the residual stress on the structure of jacket PT. Pertamina Hulu Energi West Madura Offshore PHE-24 with Shell Element. The research was conducted by analyzing the post-weld heat treatment which refers to AWS D1.1:2000 to reduce the residual stresses that occur. From the post-weld heat treatment results, the value of the residual stress can be significantly reduced [9].

Analysis of the effect of stopper size on the ship plate joints on residual stress and deformation using the finite element method shows that the longer the stopper used, the smaller the deformation and residual stress generated [10]. [11], [12]. The effect of variation of welding sequence on residual stress and distortion in SMAW welding of tubular joints shows that with this simulation the residual stress and distortion are known for each variation of the welding sequence and the type of welding sequence that produces the smallest residual stress and distortion is known [13], [14].

2. Material and Method

2.1 Material

The material used in this research is ASTM A106 Grade B obtained from the petrochemical industry. The mechanical properties of materila for Grade A and Grade B are shown in Table 1

b = width of the weld area (mm)
v = welding speed (mm/s)
t = welding time (s)

Table 1. Mechanical properties of ASTM A106 Grade A and Grade B

Mechanical Properties		
Strength	Grade A	Grade B
Tensile strength, min, psi [Mpa]	48 000 [330]	60 000 [415]
Yield strength, min, psi [Mpa]	30 000 [205]	35 000 [240]
Elongation in 2 in. or 50 mm	e = 625 000 [1940] $A^{0.2}/U^{0.9}$ or Table X4.2 and X4.2	

The welding performed was SMAW process with single pass filling it's carried out at the T joint circling the diameter of branch pipe. The dimensions used are shown in Figure 2.

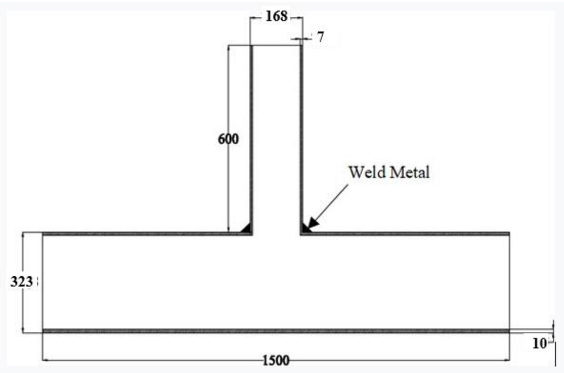

Fig 2. Dimension of welding geometry model

In the welding process, not all energy is used to heat the electrode and the weld metal. Some of the energy produced is absorbed into the environment due to contact with the surrounding air, the arc energy used in welding can be formulated in equation 1 [15].

$$Q = \eta . V . I \quad (1)$$

Q = net heat input (watts)
η = welding efficiency coefficient (-)
V = arc voltage (volt)
I = electric current (amperes)

Meanwhile, the amount of heat flux applied to the material is determined by equation 2 and equation 3

$$qe = Q/Ae \quad (2)$$

$$q1 = qe\, A1/Af \quad (3)$$

A_l = length of element subjected to heat flux load (mm)
A_f = area of the resulting loading electrode (mm^2)
A_e = area of the electrode used (mm^2)

Meanwhile, to find the area of the loading area, equation 4 is used:

$$A_f = b.v.t \quad (4)$$

2.2 Finite Element Method

The finite element method is a numerical method used to solve engineering problems such as geometry, loading, and material properties, which are difficult to solve with mathematical analysis solutions. The finite element method approach is to use the information at nodes. In the process of determining the nodes called discretization, a system is divided into smaller parts, then problem-solving is carried out on these parts and recombined to obtain an overall solution. In this work, ANSYS R 17.2 software is used. This software is a package program that can model finite elements to solve problems related to mechanics, including static, dynamic, structural analysis, heat transfer problems, fluid problems, and also problems related to acoustics and electromagnetics.

Initially the geometry of the pipe components was designed as shows in figure 2. The next process is meshing, which divides the entire system into small, uniform elements with the aim that the analysis becomes more detailed on the entire system. The physical form of the mesh is mechanical, this type of mesh produces 78902 elements and 39049 nodes, the mesh model is shown in Figure 3

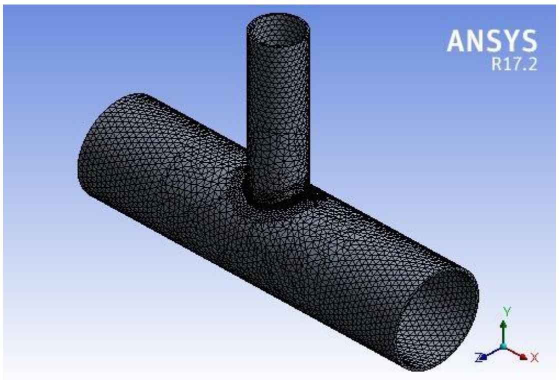

Fig 3. Meshing of component

Determine the welding parameters, where the welding current used is 110 amperes, 130 amperes, and 160 amperes. As for the other welding parameter data which is a fixed variable, namely the welding speed of 2 mm/s, the voltage used is 25 V, the welding efficiency is 85%, and the electrode diameter is 3.2 mm. Determination and provision of heat load in the form of moving heat flux and convection. Convection is a boundary condition in transient thermal simulations.

During the welding process, there is a convection process between the surface of the pipe and the surrounding air. To simulate the convection process that occurs, convection data input is carried out. Convection data obtained from Software Ansys R 17.2. After performing a thermal analysis, then a structural analysis is carried out using the output results from the thermal analysis in the form of temperature distribution.

3. Results and Discussion

The source of welding heat charged in the pipe model is a heat flux load. Calculation of the amount of heat flux load applied to the model can be calculated by equation 2 and equation 3. The results of the calculation of heat flux with variations in electric current as welding heat input are obtained with a current of 110 amperes is 19.47 W/mm2, a current of 130 is 23.02 W/mm^2, and a current of 160 amperes is 28.30 W/mm. Thus it has been known that the amount of heat flux load imposed on the elements through which the welding grooves pass in the model. The heat source in the form of heat flux produced is a connection due to a welding set. However these results are not a result of this study, but a consequence of the design of the welding parameters and the equations used.

The results of the thermal analysis at a current variation of 110 A with a heat flux load on the elements through which the welding groove is 19.47 W/mm2 obtained the results in the form of temperature distribution in the weld area and heat affected areas, the results are as shown in Figure 3. The maximum temperature obtained at a current of 110 A has exceeded the melting point of the material, which is 1500°C. in the first seconds of welding the maximum temperature obtained is 1739.6°C.

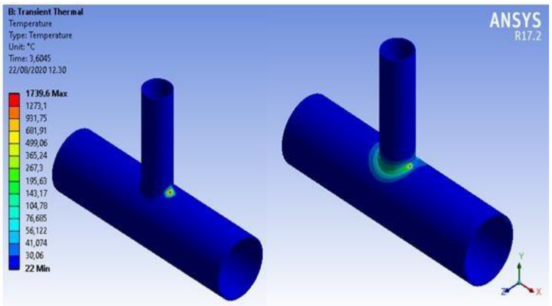

Fig 4. Temperature distribution with a current of 110 amperes

Figure 5 shows the results of running a thermal analysis with variations in welding currents, namely 100 A, 130 A, and 160 A. The results obtained are temperatures at each welding current, namely 1750ºC, 2100ºC, and 2500ºC. The simulation results show that there is an increase in temperature with the greater current used as welding heat input. This shows that there is a correlation between the increase in the given current and the resulting temperature.

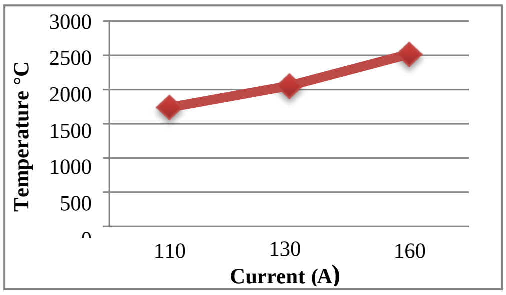

Fig 5. The temperature at each welding current

After running structural analysis with a current of 110 amperes, it is obtained that the maximum von Mises residual stress value is 399.59 MPa, and a minimum of 4.68 MPa, at a current of 130 A, a value of 420.01 MPa is obtained, at a current of 160 A the voltage value increases by 454.06 MPa. as shown in Figure 6.

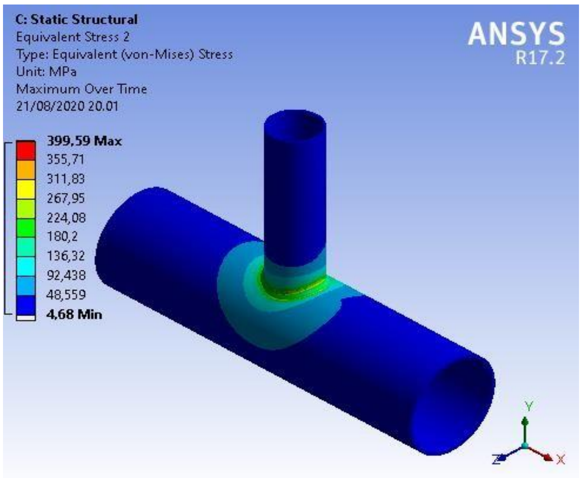

Fig 6. Results of residual stress analysis with a current of 110 A

Table 1. Residual stress with current variation

Current (A)	Residual stress von Mises (MPa)
110	399.59
130	420.01
160	454.06

The results of the structural analysis with variations in welding current in the form of von Mises residual stress at each welding current are shown in Table 1. From these results, it can be concluded that by increasing the value of the electric current, the resulting voltage will be even greater. The residual stress is higher with increasing welding current, this indicates contraction due to the influence of heat on the weld area.

4. Conclusion

Based on the simulation results using the finite element method software, several conclusions can be summarized. The maximum temperature that occurs has exceeded the material's melting point limit, which is 1500°C. Too high a welding temperature will cause the HAZ area to widen. Residual stress in SMAW single pass pipe welding on ASTM A 106 Grade B pipe material. With numerical simulation, the maximum residual stress value of von Mises at a current of 110 A is 399.59 MPa. The value current of 130 A produced a residual stress of 420.01 MPa. At a current of 160 A, the voltage value increases by 454.06 MPa. Temperature differences due to welding can also affect the distribution of heat-induced residual stress. It's causing the material to become hard but brittle (brittle) so that it can cause structural failure in the joining area.

Reference

[1] W. Ma, H. Zhang, W. Zhu, F. Xu, and C. Yang, “Study on residual stress of welded hoop structure,” *Appl. Sci.*, vol. 10, no. 8, p. 2838, 2020.

[2] H. Wiryosumarto and T. Okumura, “Teknologi pengelasan logam, PT,” *Pradnya Paramita, Jakarta*, 2000.

[3] H. Hamdani, “Pengaruh Masukan Panas Proses Pengelasan terhadap Sifat Mekanik Baja AISI 1045 suatu Kajian Analitis dan Analisa Numerik,” *J. Polimesin*, vol. 17, no. 1, pp. 1–8.

[4] S. Das Banik, S. Kumar, P. K. Singh, S. Bhattacharya, and M. M. Mahapatra, “Distortion and residual stresses in thick plate weld joint of austenitic stainless steel: Experiments and analysis,” *J. Mater. Process. Technol.*, vol. 289, p. 116944, 2021.

[5] B. Kumar and M. Manikandan, “Assessment of process, parameters, residual stress mitigation, post treatments and finite element analysis simulations of wire arc additive manufacturing technique,” *Met. Mater. Int.*, vol. 28, no. 1, pp. 54–111, 2022.

[6] S. S. Miftin, H. M. Mohammed, and A. A. Nassar, “Measurement and Prediction of Residual Stresses in Low Carbon Steel Pipes Welded Shielded Metal Arc Welding,” *Basrah J. Eng. Sci.*, vol. 20, no. 2, pp. 60–65, 2020.

[7] A. Yaghi, T. H. Hyde, A. A. Becker, W. Sun, and J. A. Williams, “Residual stress simulation in thin and thick-walled stainless steel pipe welds including pipe diameter effects,” *Int. J. Press. Vessel. Pip.*, vol. 83, no. 11–12, pp. 864–874, 2006.

[8] H. Alipooramirabad, A. Paradowska, R. Ghomashchi, and M. Reid, “Investigating the effects of welding process on residual stresses, microstructure and mechanical properties in HSLA steel welds,” *J. Manuf. Process.*, vol. 28, pp. 70–81, 2017.

[9] F. N. Lazawardy, “Analisa Tegangan Sisa dan Distorsi Menggunakan Metode Elemen Hingga Pada Struktur Jacket PT. Pertamina Hulu Energi West Madura Offshore PHE-24 Dengan Dengan Shell Element.” Institut Teknologi Sepuluh Nopember, 2017.

[10] R. B. Putra, “Analisis pengaruh ukuran stopper pada sambungan pelat kapal terhadap tegangan sisa dan deformasi menggunakan metode elemen hingga.” Institut Teknologi Sepuluh Nopember, 2017.

[11] D. Jin, C. Hou, and L. Shen, "Effect of welding residual stress on the performance of CFST tubular joints," *J. Constr. Steel Res.*, vol. 184, p. 106827, 2021.
[12] P. Biswas, N. R. Mandal, P. Vasu, and S. B. Padasalag, "Analysis of welding distortion due to narrow-gap welding of upper port plug," *Fusion Eng. Des.*, vol. 85, no. 5, pp. 780–788, 2010.
[13] K. Ogawa, D. Deng, S. Kiyoshima, N. Yanagida, and K. Saito, "Investigations on welding residual stresses in penetration nozzles by means of 3D thermal elastic plastic FEM and experiment," *Comput. Mater. Sci.*, vol. 45, no. 4, pp. 1031–1042, 2009.
[14] K. Masubuchi, *Analysis of welded structures: residual stresses, distortion, and their consequences*, vol. 33. Elsevier, 2013.
[15] A. Pilipenko, "Computer simulation of residual stress and distortion of thick plates in multielectrode submerged arc welding: Their mitigation techniques," 2001.