EFFECT OF WELDING SEQUENCE ON DISTORTION RESULTS IN BRACKET CONNECTIONS

Suheri Suheri[1*], Syamsul Bahri Widodo[1], Zainal Arif[1], Surawan Surawan[1], Sulaiman Ali[2], Nurman Pamungkas[3], Muhamad Syafii[4]

[1] Department of Mechanical Engineering, Faculty of Engineering, Universitas Samudra, Langsa, Aceh
[2] Department of Mechanical Engineering, Faculty of Engineering, Universitas Teuku Umar, Meulaboh, Aceh
[3]Batam State Polytechnic Ship Planning and Construction Engineering, Batam
[4]Department of Mechanical Engineering, Faculty of Engineering and Computer Science, Universitas Sains Alquran, Wonosobo

*Email: suheri@unsam.ac.id

ABSTRACT

The distortion in the shape of the weld specimen is influenced by the uneven heat distribution during the welding process. There is also distortion in the welding modeling performed on the bracket joint. The modeling scenario of the bracket joint forms the basis for measuring the distortion results. Therefore, when we model the bracket joint, we obtain varying distortion results. There are three lines to observe the distortion effects. Lines a, b, and c are located on the X and Y axes (center), while line c is on the X and Y axes (+). Line a is on the X and Y axes (-). The test specimen that has been modeled is the result of numerical modeling, which is then used to calculate the magnitude of the distortion on the longitudinally stiffened plate test specimen. For the numerical modeling of the bracket joint, the result of scenario 1 is distortion. Meanwhile, the distortion results of the bracket joint modeling in scenarios 1 and 2 show that the distortion value obtained in scenario 2 is higher than that obtained in scenario 1. This is influenced by the order of welding lines used in the bracket joint modeling.

Keywords: Distortion, bracket joint, influence of welding sequence

INTRODUCTION

Welding is one of the most common methods of joining materials used in the manufacturing industry. The advantages of welding which include high joint strength and flexibility in joining various types of materials make it the main choice in making metal structures [1]. However, welding also has its own obstacles, one of which is the presence of twisting in the welding system which generally occurs due to slanted (non-uniform) temperature changes. Due to its impact on the functional dependencies of the welded structure, twisting is one type of joint part deformity. Distortion in welding can be recognized or determined using two ways, namely through scientific or mathematical estimation and through exploratory tests [2]. The distribution of heat temperature around the welding area is greatly influenced by the heat produced during the process, which can result in distortion. In addition, heat has the potential to influence the physical and mechanical properties of the weld as well as phase transformations in the microstructure [3].

This phenomenon is actually invisible to the naked eye, because large-scale damage is hampered by the surrounding material not immediately receiving heat treatment. However, welded joints have warpage that causes more pronounced anxiety when exposed to static or cyclic loads [4]. The welded joint is subjected to a load that causes a nominal stress 0.8 times higher than the yield stress of the material. As a result, deformation occurs and distortion is released, resulting in a distortion value of zero. Thermal energy has an impact on the quality of the welding results, it also has an impact on the current, voltage and welding speed [5]. Welding energy, also known as heat input, is generated by the relationship that exists between these three parameters [6]. Welding results are greatly influenced by the amount of current used. If the current used is too low, the arc will be difficult to ignite or the arc flame will become unstable [7]. Welding produces protrusions and shallow weld penetration because the heat produced is not enough to heat the electrode [8]. Precision recreation of bend checking in T-joint fillet welds using limited component strategy [9].

Distortions that occur in welded joints can result in significant structural deformation, which in turn can affect the performance and integrity of the entire structure. In the context of bracket joints, which are often used to support or fasten other components in a system, distortion due to welding can cause mismatches, reduced precision, and even structural damage if not properly controlled [10].

Deciding on proper welding succession is one strategy that can be used to limit distortion [11]. Arranged weld batching can reduce warm pressure accumulation and skew intensity dispersion, thereby limiting twisting in the joint. In recent decades, advances in computing technology have enabled the use of simulation and numerical analysis to model and predict the behavior of welding-induced distortion [12].

Welding is carried out on one side in the modeling with the assumption that the other side is symmetrical. There are variations in flange plate thickness, weld penetration depth, presence of supports, and absence of supports [7]. In conclusion, for transverse distortion, high tensile stresses exist near the fillet weld tip, and the stress approaches zero as the distance from the weld tip increases [13]. For longitudinal bending, compressive stresses appear far from the weld location, whereas very high elastic loads occur near the weld tip [14].

The aim of this research is to investigate and analyze distortions in bracket joints caused by various welding sequences using a numerical approach. By utilizing simulation technology, it is hoped that the optimal welding sequence can be found to minimize distortion, thereby improving the quality and reliability of bracket connections in industrial applications.

RESEARCH METHODOLOGY

Numerical methods called finite element methods are used to solve engineering problems such as geometry, loading and very complex material properties. With solutions from mathematical analysis, this is impossible or difficult to solve. The limited component engineering approach is to utilize data at the focal hub. In the time spent determining the hub focuses, which is called discretization, a framework is broken down into simpler parts, then critical thinking is carried out on these parts and then they are combined again to obtain a comprehensive structure [15].

The CMW (Computational Welding Modeling) method and the FEM (Finite Element Method) approach can now be used to model welding thanks to advances in technology. CMW and display using software are tools to demonstrate circulating and residual stresses in welds with a mathematical arrangement. This demonstration makes it possible to determine the heat and mechanical conduction of the welding system effectively, so that the results are close to real conditions in the field or exploration results [16].

Bracket Connection Geometry

Section association mathematics is a type of association that is traced in one section of the FPSO (Floating Production Storage and Offloading) seaward drift structure seen in Figure 1. Bracket connections are also connected by welding, the same as connections in general. The most common joining method is Welding, which has various advantages and disadvantages. Due to its advantages, welding is often used to connect various steel structures. This is due to the high efficiency and strength resulting from the welding process. However, there are also fundamental flaws in this process. One of them is that welding can cause residual stress and distortion which can reduce the strength and life of the welded joint. Keeping the above conversation in mind, it is important to examine the associations created by welding. To examine all associations within an FPSO requires a significant investment of time and expense. Therefore, to minimize risks, checks are carried out on several parts of the FPSO structure using mathematical techniques. The geometry of the bracket connection is simplified before carrying out welding simulations based on numerical modeling. Numerical modeling focuses on the analysis of residual stresses and distortions produced by the welding process.

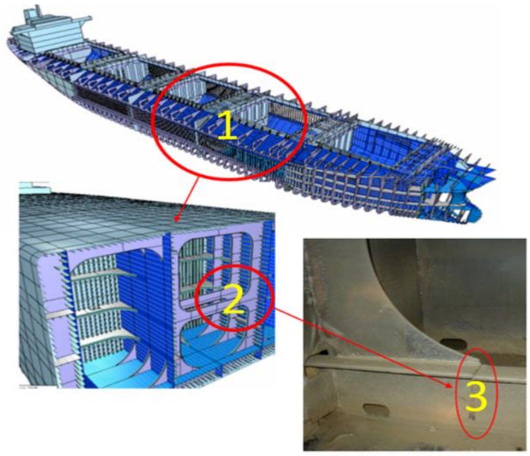

Figure 1**.** Geometric scheme of the bracket connection on the FPSO

Previously we discussed the basics of obtaining bracket connection geometry, with Cross-Section 3 as the original shape seen in one of the FPSO structures in Figure 1. and the geometric shape has been further simplified to make it easier to understand. Rearrangements were also made to work with the demonstration system and it is believed that it can later be applied to real conditions. Based on the dimensions of the longitudinal reinforcement plate, the geometric shape and dimensions of the bracket connection depicted in Figure 2 show the purpose of simulating the geometric shape, which aims to simplify numerical modeling. The results of this modeling can be validated based on research conducted by Syahroni. Some of the information obtained is then used to show part associations.

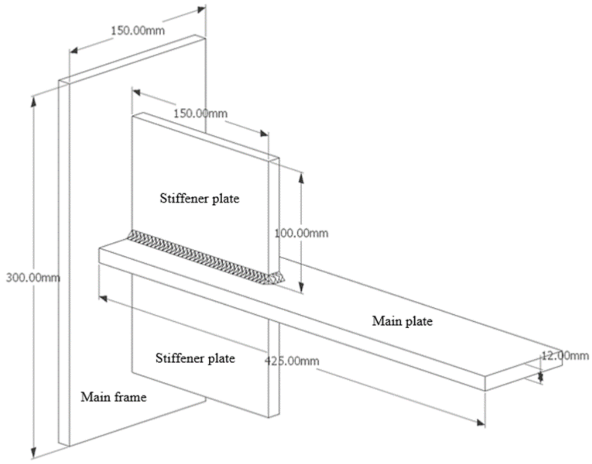

Figure 2. Simplified geometry of bracket connections

The mathematical form broken down in this exploration is a refinement of one of the association parts in the structure that floats towards the sea. The geometric shape taken refers to Syahroni's experimental findings, especially the type of connection in longitudinal reinforced plates. One method of simplifying connections In the context of offshore floating structures, such as FPSOs, a commonly used approach is to model bracket connections. Figure 3 (a) shows a cross-section of the initial geometric model of the bracket before the welding modeling process is carried out, while Figure 3 (b) shows a cross-section of the bracket geometry after the welding modeling process is carried out.

It should be noted that in this discussion, specific details regarding the type and function of FPSO structures are not discussed in depth. Conversations related to welding demonstrations occur in the following areas, relating to the welding display process in general.

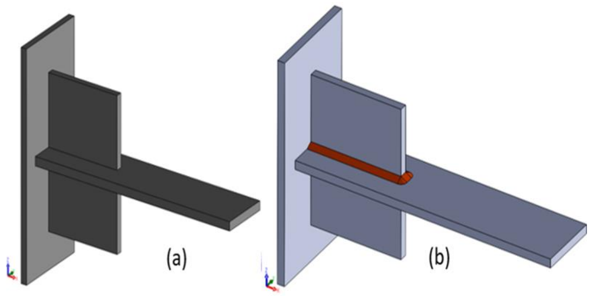

Figure 3. Cross-sectional geometric shape of bracket connection modeling; (a) before the welding modeling process, (b) after the welding modeling process

The first step in numerical modeling is to experiment with several different types of meshing that must be close to the physical research results and meet the modeling criteria. In this case, the approval of the mathematical display for the association of parts in marine structures refers to the consequences of research conducted by Syahroni [17].

Material Data

The demonstration carried out should utilize information. The material chosen for the experiment must be close to or in accordance with the material used in the trial research. The goal is to ensure that the material properties modeled numerically are identical or very similar to actual conditions. Material data for modeling can be obtained from a variety of well-described sources, and should have much in common with what experimental specimens show. Figure 3 depicts the data used to obtain material data for bracket connection modeling. Several parameters that help the modeling process are obtained from this data. Meanwhile, ASTM A131 Steel is often used in the shipbuilding, construction and manufacturing industries because it has good mechanical properties, including a high level of flexibility. The chemical composition data of ASTM A131 steel is very important for understanding the mechanical characteristics and classification of carbon content in steel structures. If you have Table 1 or the chemical composition data, I can help you analyze it further or provide additional information. Steel with a carbon content between 0.05% and 0.30% is considered low carbon steel. Therefore, ASTM A131 steel is known as low carbon steel.

Table 1. Material composition for ASTM A131 steel

Notation	**Element**	**Composition, (wt %)**
C	*Carbon, max*	0.18
M N	*Manganese*	0.90 – 1.60
P	*Phosphorous, max*	0.035
S	*Sulfur, max*	0.04
Si	*Silicon*	0.10 – 0.50
Ni	*Nickel, max*	0.40
Cr	*Chromium, max*	0.25
Mo	*Molybdenum, max*	0.08
Cu	*Copper, max*	0.35
Nb	*Niobium, max*	0.05
V	*Vanadium, max*	0.10
Al	*Aluminum*	0.015 – 0.020

Welding path scenarios in bracket connection modeling

The bracket connection is modeled in two different scenarios. This situation depends on the arrangement of the weld lines and this situation alludes to the results of the exploratory inspection that Syahroni recently carried out. The essence of the welding way of presenting situations is to obtain the mutilation values created due to the welding results of each situation and then break down the results obtained to compare them and the graphic trademark produces an exploratory examination of the finished longitudinally made plate. by Syahroni. The main situation is shown with

The welding demonstration system starts from the middle side, as seen in Figure 4. Welding is arranged by grouping welding lines. In Figure 4-3, the first weld path starts from point A and continues to point B, while the second weld path starts from points C and D. Furthermore, the third and fourth weld paths start from the same points as the first and second paths, but at different positions. Figure 4 depicts the welding position. Figure 4 shows the upper position of the first and second welding lines, while the third and fourth welding lines are in a lower position.

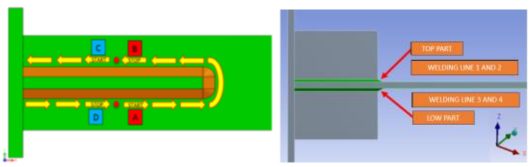

Figure 4. Illustration of welding scenarios based on welding paths

RESULTS AND DISCUSSION

Research result

*Fittings*is the division of an article into several parts or in the limited component technique it is called discretization. Discretization is a method that involves separating a design or object into small components of a certain size, which are then connected as boundaries of the construction or object being studied. Finite element-based modeling studies rely heavily on the process of meshing or forming a network of these small elements. Meshing is very helpful in modeling because it makes calculations easier on the model being analyzed.

Mesh analysis is carried out first by varying various mesh sizes before the modeling stage. The goal is to ensure that the meshing size chosen is appropriate not to influence the specified results, and remains consistent throughout the modeling process. To understand the graphic characteristics of modeling results and Syahroni's experimental research results, a comparison was made between mesh size variations and modeling validation. In this way, agreement on the meshing size can be achieved for subsequent modeling.

Demonstration of cross-sectional variations is divided based on global component size and component size at the weld point. In order to do this changing the mesh size can focus on the weld bead area and save storage space, thereby lowering computing costs. Based on the differences in mesh sizes in modeling variations, experiments on several mesh size modeling scenarios with varying numbers of nodes and elements are presented in Table 2.

The amount of time required to calculate the modeling of mesh size variations is also shown in Table 2. From Table 2, the constraints evaluated to indicate with changes in cross-sectional size include worldwide component size (overall model), component size in the weld point region, number of hubs and components, as well as calculation time.

Table 2. Results of variations in mesh size

Global Element Size (mm)	12	9	8	6	6	6
Weld bead element size (mm)	12	12	8	6	5	4
Number of Node	3421	6420	7868	13003	14320	14353
Number of Elements	1681	3865	4777	8386	9204	9280
Total Computation Time (minute)	30	54	66	102	128	132

Table 2 of the results obtained in shows some interesting differences with respect to the size of the grid that produces the results. The main difference can be seen as a consequence of the number of components and hubs. Modeling with a weld bead size of 4 mm produces almost the same number of elements and nodes as modeling with a weld bead size of 5 mm and 6 mm. This also applies to cross-sectional views with weld dab sizes of 8 mm and 12 mm. Meanwhile, modeling with a weld bead size of 12 mm has the smallest number of components and hubs.

The total computing time for each meshing size shows the second difference. From the results obtained, it tends to be reasonable that the smaller the size of the cross section component, the longer the expected registration time for display. For this situation, the expected term in a logging system with multiple grid sizes is also an extraordinary thought because it concerns productivity in relation to the expected time period to perform a model. The results of variations in network size Figure 5 shows are recorded in Table 2 with their specific sizes. Mesh modeling is carried out on each part of the bracket connection geometry. In Figure 5, all the grids in the weld tip area are tetraedrons, while in other areas the cross section is covered with hexagonal elements. This is because the weld ball region has a more complex mathematical shape compared to the mathematical shape in other areas of the bracket connection geometry.

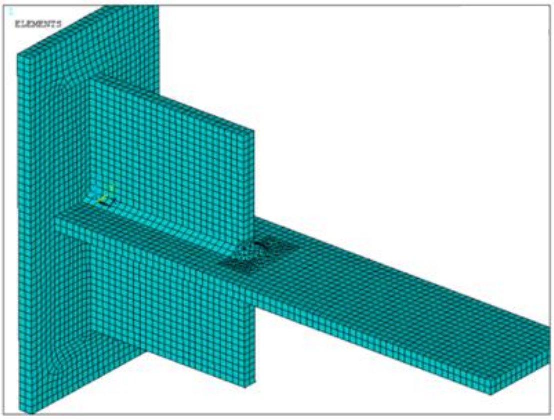

Figure 5. Cross-sectional geometric shape for bracket connections with a mesh size of 6 mm

The effect of varying the network size based on Table 2 with a proficiency value of 70% will be seen in Figure 6. The graphic plot will display several meshing results that meet the criteria for the number of elements for the residual stress results of the Syahroni experiment.

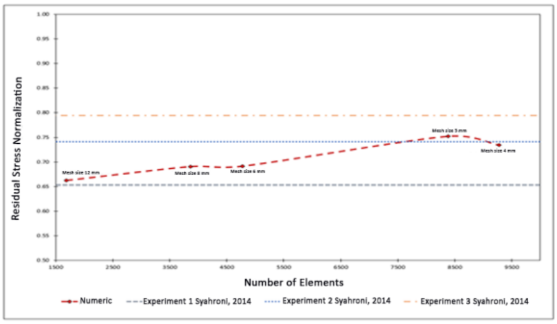

Figure 6. Sensitivity of the number of meshing elements at the centerline at a distance of 6 mm from the weld toe

Figure 6 should show the results of network size variations covering all trajectory foci covered in the test results by Syahroni, including fitting sizes of 12 mm, 8 mm, and 6 mm for global component sizes, and 5 mm and 4 mm for weld ball sizes. According to Table 2, the best results were obtained at a grid size of 6 mm, which corresponds to the global component size and spot weld component size.

Taking into account the previous conversation and the consequences of Table 2, as well as looking at the resulting cross-sectional conditions in Figures 4 to 8, it can be assumed that to display the cross-sectional associations in structures that have been washed out to sea, using a grid size of 6 mm is the right choice. This is based on considering the optimal number of hubs and components, and avoiding undesirable extremities, while keeping important time efficiencies in mind. Thus, the use of a grid size of 6 mm is expected to simplify the modeling process, as well as assist in validating numerical modeling results that are close to the results from physical modeling.

Bracket Connection Distortion Results

The impact of the skewed intensity distribution during the welding system causes twisting as in the welding example. There is also distortion in the welding modeling carried out on the bracket connection. Estimating how much twist the association part displays is done by dividing the three perception areas as lines leading to hub so that when we model the bracket connection, we get different distortion results.

In Figure 7, there are 3 lines used to observe the bending results. Lines a, b, and line c is located in the positive quadrant of the X and Y axes, while line a is located in the negative quadrant of the X and Y axes.

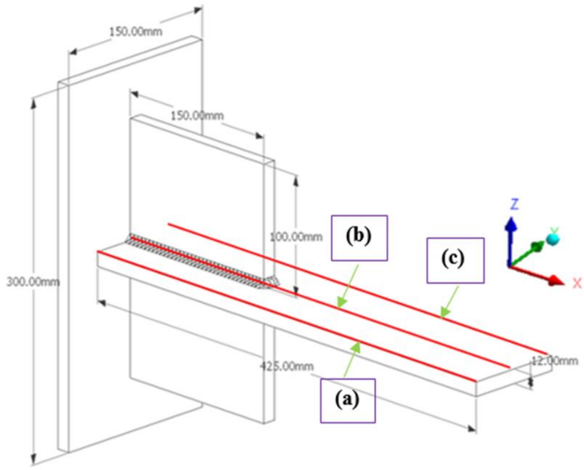

Figure 7. Observation of distortion data collection at bracket connections

Syahroni's experimental research validates observational findings, which are then outlined in graphs for analysis based on each scenario. The agreement is finalized only on the bending effect of situation 1 (welding way from the middle side).

Figure 7-10 shows the methods involved in gathering information to distort part associations in relation to the exploratory examination directed by Syahroni. Figures 8 and 9 show experimental research specimens on the Syahroni longitudinal reinforced plate and the position of the distortion direction.

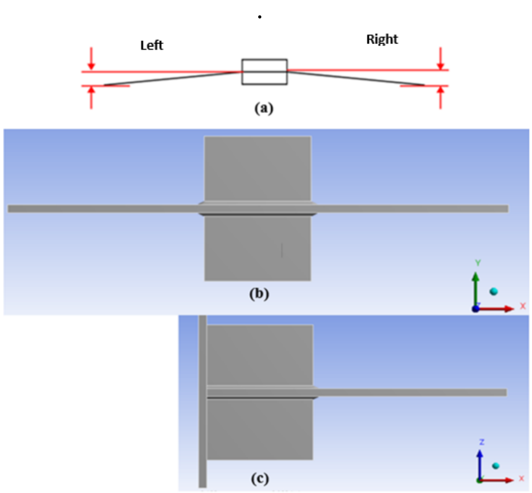

Figure 8. The process of taking distortion data at the bracket connection.

The estimation results are plotted in the graphs introduced in Figure 8 for the bending results of situation 1 and the mathematical results and exploration checks of Syahroni. The graphic results in Figure 8 are based on Figures 8 (a) and (b) taken at each position.

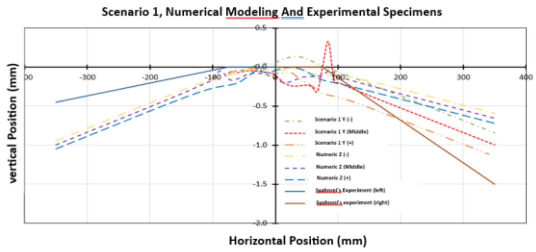

Figure 9. Graph of distortion results for Scenario 1, Numerical results and Experimental Specimen results

The experimental specimen modeled in Figure 9 is the basis for the numerical modeling results, which are then used to calculate the amount of distortion in the longitudinal reinforced plate specimen. Meanwhile, the distortion results from numerical modeling of the bracket connection are depicted in Figure 9 for scenario 1.

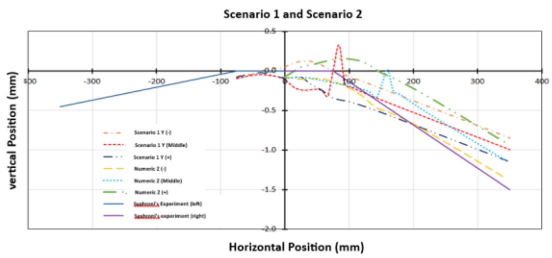

Figure 10. Graph of distortion results for Scenario 1 and Scenario 2

From the graphic characteristics in Figure 10, it is clear that the distortion value in scenario 2 is higher than in scenario 1. This difference can be attributed to the way the welding lines are arranged in modeling the bracket connection.

CONCLUSION

The experimental specimens that have been modeled are the results of numerical modeling which are then used to calculate the amount of distortion in the longitudinal reinforced plate specimens. There is also distortion in the welding modeling carried out on the bracket connection. The scenarios used in bracket connection modeling are the basis for measuring distortion results. In the numerical modeling of the bracket connection, the result of scenario 1 is the observed distortion. The distortion results from modeling the bracket connection in scenarios 1 and 2 show that the distortion value in scenario 2 is higher than in scenario 1. This difference is influenced by the sequence of welding lines applied in modeling the bracket connection.

REFERENCES

[1] Koesgi, Dicky Prasetyo, et al. 2021. Effect of Preheating on the Mechanical Properties of Spot Friction Stir Welding Joints in Installing Aluminum Rivets 2024. *Teknika STTKD: Journal of Engineering, Electronics, Engines*, Vol. 7 (1), pp. 140-153.

[2] Harun, S. and Burhanuddin, Y., 2010. Pemodelan dan Simulasi Urutan Pengelasan Bilah Roda Traktor Berbasis Metoda Elemen Hingga. *Jurnal Teknik Mesin*, Vol. 12 (2), pp.109-118.

[3] Sutekat, A.G., 2018. Studi Eksperimen dan Numerik Pengaruh Variasi Urutan Pengelasan Terhadap Distorsi Sudut Pada Struktur Panel Baja A36 (*Doctoral dissertation, Institut Teknologi Sepuluh Nopember).*

[4] Sochib, M. and Afif, M., 2016. Analisa Perbandingan Pengelasan Smaw Dengan Variasi Ampere Terhadap Sifat Mekanis. *Wahana Teknik*, Vol. 5(1).

[5] Mizhar, S. and Pandiangan, I.H., 2014. Pengaruh Masukan Panas Terhadap Struktur Mikro, Kekerasan Dan Ketangguhan Pada Pengelasan Shield Metal Arc Welding (SMAW) Dari Pipa Baja Diameter 2, 5 Inchi. *Jurnal Dinamis,* Vol. II (14), pp.16-21.

[6] Helanianto, Helanianto; Epriyandi, Epriyandi; Rahmadi, Hairian. 2014. Pengaruh Variasi Arus Pengelasan SMAW Terhadap Kekerasan Logam Induk Dan Logam Las. *Elemen: Jurnal Teknik Mesin*,Vol. 7 (2), pp. 138-147.

[7] Wicaksono, A.P., 2005. Simulasi Distribusi Panas, Tegangan Sisa Dan Distorsi Dengan Metode Elemen Hingga Pada Pengelasan Fillet Tipe T.

[8] Putri, F., 2009. Pengaruh Besar Arus Listrik Dan Panjang Busur Api Terhadap Hasil Pengelasan. *Austenit*, Vol. 1 (02).

[9] Nuryadin, B.W., 2020. Pengantar Fisika Nanomaterial: Teori dan Aplikasi.

[10] Pamungkas, N., Leksonowati, N.F.P., Mufariz, M.F., Hakim, R., Baharudin, B., Dzulfiqar, M.A., Saputra, I., Aryswan, A. and Reynaldi, M.R., 2023. Pengaruh Proses Tacking dan Welding Sequence Terhadap Distorsi Pada Pengelasan Butt Joint Pipa Dengan Flange Berbahan Carbon Steel. *Jurnal Teknologi dan Riset Terapan (JATRA),* Vol. 5 (2), pp.80-86.

[11] Herdiarto, E.F., 2022. Analisa Pengaruh Variasi Urutan Pengelasan Terhadap Tegangan Sisa Dan Distorsi Pada Pengelasan Sambungan Pelat Baja Tahan Karat SS304 dengan Metode Simulasi, *Doctoral dissertation*, Universitas Muhammadiyah Ponorogo.

[12] Fahlevi, R., 2011. Pola Sambungan pada Batang Kuda-kuda Konstruksi Baja (Studi Literature).

[13] Mulyadi, S.T. and Iswanto, I., 2020. Teknologi Pengelasan.

[14] Nurhayati, R.B., Nur Syahroni, S.T. and Handayanu, I., 2016. Analisa Pengaruh Welding Sequence Terhadap Tegangan Sisa Dan Distorsi Pada Sambungan TUBULAR DT-JOINT Menggunakan Metode Elemen Hingga (*Doctoral dissertation, Institut Technology Sepuluh Nopember*).

[15] Akbar, T. and Santosa, B., 2012. Analisa Pengaruh dari Welding Sequence Terhadap Tegangan Sisa dan Deformasi pada Circular Patch Weld Dobule Bevel Butt-Joint Plat ASTM A36 Menggunakan Metode Elemen Hingga. *Jurnal Teknik ITS*, Vol. 1 (1), pp. G351-G356.

[16] Kamelia, L., 2015. Perkembangan teknologi augmented reality sebagai media pembelajaran interaktif pada mata kuliah kimia dasar. *Jurnal Istek*, Vol.9 (1).

[17] Syahroni N, 2014. Fatigue Assessment of Welded Joints Taking into Account Effects of Residual Stress, *Doctoral Thesis*, NTNU, Trondheim.