Numerical Analysis of Residual Stress and Distortion Use Finite Element Method on Inner Bottom Construction of Geomarin IV Survey Ship with Welding Sequence Variations

N Syahroni ¹, A B W Hartono ¹, and M Murtedjo¹
¹Ocean Engineering Department, Faculty of Marine Technology, Sepuluh Nopember Institute of Technology, Indonesia
²Email: nur.syahroni@gmail.com

Abstract. In the ship fabrication industry, welding is the most critical stage. If the quality of welding on ship fabrication is not good, then it will affect the strength and overall appearance of the structure. One of the factors that affect the quality of welding is residual stress and distortion. In this research welding simulation is performed on the inner bottom construction of Geomarin IV Ship Survey using shell element and has variation to welding sequence. In this study, welding simulations produced peak temperatures at 2490 K at variation 4. While the lowest peak temperature was produced by variation 2 with a temperature of 2339 K. After welding simulation, it continued simulating residual stresses and distortion. The smallest maximum tensile residual stress found in the inner bottom construction is 375.23 MPa, and the maximum tensile pressure is -20.18 MPa. The residual stress is obtained from variation 3. The distortion occurring in the inner bottom construction for X=720 mm is 4.2 mm and for X=-720 mm, the distortion is 4.92 mm. The distortion is obtained from the variation 3. Near the welding area, distortion value reaches its minimum point. This is because the stiffeners in the form of frames serves as anchoring.

Keywords: Finite Element Method, Inner Bottom, Frames, Residual Stress, Distortion

1. Introduction

The fabrication stage of the survey vessel basically has the same principle with other commercial vessels, as in the assembly stage to the build stage. However, in this study, one part of the survey ship block, the inner bottom, receives special attention, especially the welding process. Welding is the process of combining two or more metals based on the principle of diffusion process that has the advantage of being able to make lightweight construction, can withstand high strength, easy in process, and economical [1].

In the construction of a ship, welding has a percentage of about 35% of the total activity. During the welding process, heat transfer occurs along the weld area allowing uneven distribution of heat leading to thermal development. In areas that have cooled down, a stretch barrier arises, which is capable of causing a shape change or is often referred to as residual stress [2]. During heating and cooling cycles, there will be a heat strain between the weld metal and the base metal in the area close to the weld bead, the heat strain is capable of triggering the buckling, bending, and rotation called distortion [3].

The presence of residual stress and distortion can reduce load capacity which contributes to the occurrence of brittle fracture and surface shape change [4]. Distortion can reduce bending strength of the structure to affect reliability [5].

In order to know the magnitude of residual stress and distortion, various experiments and simulations have been performed even with simple objects. In this Study will be implemented simulation to know...
the amount of residual stress and distortion in more complex construction. The study was performed using finite element method with the help of ANSYS Workbench 16.0 software to obtain residual stress and distortion stress on the construction of Geomarin IV inner bottom vessel that varied against the welding sequences.

2. Research Methodology

The systematic work of this research has been described as follows:

2.1 Data Collection

Data collected to support the research needs include:
- Block 1.6 drawing of Geomarin IV survey ship.
- Welding Procedure Specification for Submerged Arc Welding (SAW).

2.2 Geometry Modelling

The modeling in this research is 2D and 3D geometry of inner bottom construction along with frames located in block 1.6. The modeling was carried out using AutoCAD 2016 software support. Block 1.6 has 18 frames with solid floor type, open floor, and watertight floor. However, in this study, the number of frames that will be analyzed only 3, due to the ability of supporting computers that do not meet, and due to the size of the construction is large enough. Three frames that are used as the object of analysis are frames 74, 75, and 76. Frame 74 is solid floor, frame 75 is watertight floor, and frame 76 is open floor. The geometry modeling results are listed in Figures 1 to 3.

![Figure 1. Frames Modelling](image1.png)

![Figure 2. Inner Bottom Modelling](image2.png)
2.3 Geometry Model Input

In this study, the geometry model is modeled using shell elements. This is because the shell element is faster in helping to obtain the desired solution [6]. Geometries that have been modeled in AutoCAD 2016 must be saved with an .iges extension to be readable by ANSYS Workbench 16.0. Geometry modeling that must be entered is a 2D (area) geometry because the element used is shell. Figure 4 shows the geometry modeling that has been inserted into ANSYS.

![Figure 3. Frames for Research Objects](image1)

![Figure 4. Geometry Model in ANSYS](image2)

2.4 Transient Thermal Analysis

Thermal analysis is a basic analysis in the simulation of the welding process to obtain heat distribution. In this study, the type of welding used is Submerged Arc Welding (SAW) with a 2-sided fillet welded T-joint connection type. The welding data used is 575 A electric current, 34 V voltage, and welding speed of 9.31 mm/s. Electrode used is EM13K type. The analyzed construction is composed of ASTM A131 Grade AH36 steel. Based on these parameters, the heat flux used is $179.9 \times 10^6$ W m$^{-2}$.

2.5 Transient Structural Analysis

After transient thermal analysis, the next analysis is transient structural analysis. The purpose of this analysis is to determine residual stress and distortion that occurs in the object of research. At this stage, construction is given boundary condition (fixed boundary condition) in the form of fixed supports (pedestal flops). The pinch is placed on the ends of the research object along with the center that is not covered by the inner bottom. AH36 steel has a yield stress value of 350 MPa.
3. Analysis and Discussion

3.1 Partition of Load Steps

The welding simulation process is numerically performed by dividing the welding area into several hot loading areas. Figures 5 to 7 show the length of the welding area in each frame, and Table 1 shows the details of the welding area division.

![Figure 5. Welding Area Length of Frame 74](image1)

![Figure 6. Welding Area Length of Frame 75](image2)

![Figure 7. Welding Area Length of Frame 76](image3)

| Frame | Welding Area Length (mm) | Partition of Load Steps (mm) | Size of 1 Element (mm) |
|-------|--------------------------|-----------------------------|-----------------------|
| 74    | 3673.39                  | 60                          | 61.22                 |
|       | 2900                     | 45                          | 64.4                  |
| 75    | 6718.04                  | 105                         | 63.98                 |
| 76    | 945.9                    | 15                          | 63.06                 |
|       | 5060                     | 80                          | 63.25                 |
|       | 554                      | 10                          | 55.4                  |

According those partition, one frame has 210 loadsteps, and all of frames which researched have 630 loadsteps.
3.2 Heat Flux Load Calculation

The formulation which used in the determination of heat flux value is:

\[ q_{el} = q_l \frac{A_l}{A_f} \]  \hspace{1cm} \text{(1)}

where,
- \( q_{el} \): Heat flux of element (W/mm²)
- \( q_l \): Heat flux of electrode (W/mm²)
- \( A_l \): Area width under heat load (mm²)
- \( A_f \): Loading area width from welding process (mm²)

3.3 Meshing Sensitivity

Meshing sensitivity has a function to know the number of elements change in numerical analysis. The smaller the divisor element, the smaller the completion value of the analysis. But it affects the duration of completion of the analysis, as it gets longer in giving results. Elements used in this study is a shell element, because it has a running time that tends to be faster than solid elements. Figure 8 illustrates meshing sensitivity in this study.

![Figure 8. Meshing Sensitivity Graphic](image)

Based on the meshing sensitivity that has been done, the type of elements used in this study is 0.2 mm quadrilateral and 0.5 mm triangle. Meshing results can be seen in Figure 9.

![Figure 9. Meshing with 0.2 mm Quadrilateral and 0.5 mm Triangle](image)
3.4. Heat Distribution

The heat distribution validation was performed using research result owned by Chen [7], Wibowo [5], and Wibisono [6]. Analysis on ANSYS with transient thermal analysis type is required to obtain heat distribution.

At this stage, variations in the welding sequence are performed to improve efficiency during the fabrication process. Figure 10 show variations in this research.

![Figure 10. Welding Sequence Variations](image)

**Table 2. Explanation of Welding Sequence Variations**

| Variation | Frame 76       | Frame 75       | Frame 74       |
|-----------|----------------|----------------|----------------|
| 1         | Starboard → Portside | Portside → Starboard | Starboard → Portside |
| 2         | Portside → Starboard | Starboard → Portside | Portside → Starboard |

| Frame 74 | Frame 75 | Frame 76 |
|----------|----------|----------|
| Starboard → Portside | Portside → Starboard | Starboard → Portside |

| Variation | Frame 74 | Frame 75 | Frame 76 |
|-----------|----------|----------|----------|
| 3         | Starboard → Portside | Portside → Starboard | Starboard → Portside |
| 4         | Portside → Starboard | Starboard → Portside | Portside → Starboard |

After the analysis, heat distribution graphs have shown in Figure 11 as sample. Based on the results, peak temperature each welding sequence variations have shown in Table 3.

**Table 3. Peak Temperature Each Variations**

| Variation | Peak Temp. (K) |
|-----------|---------------|
| 1         | 2339.1        |
| 2         | 2339          |
| 3         | 2483.4        |
| 4         | 2490          |
Figure 11. Heat Distribution Graph Variation 1

Figure 12 below shows final condition of heat distribution variation 1 as sample.

Figure 12. Heat Distribution Variation 1

Peak temperature in each variations have passed 1800 K of material’s melting point

3.5 Residual Stress

The residual stress analyzed in this research is longitudinal residual stress. The illustration of the longitudinal residual stress illustrated by a pathline has been shown in Figure 13.

Figure 13. Residual Stress Analysis’ Pathline
Using 4 variations of the predetermined welding sequence, the residual stress analysis results have shown in the graphs in Figure 14 to 17 below.

**Figure 14.** Graph of Residual Stress Variation 1

**Figure 15.** Graph of Residual Stress Variation 2

**Figure 16.** Graph of Residual Stress Variation 3
Based on analysis, 3rd variation is a welding sequence variation which has minimum tensile residual stress and minimum compressive residual stress. Maximum tensile residual stress in 3rd variation is 375.23 MPa, and maximum compressive residual stress is -20.18 MPa. Table 4 gives the summary of residual stresses in all variation.

Table 4. Summary of Residual Stress All Variations

| Variation | Max. Tensile Residual Stress (MPa) | Max. Compressive Residual Stress (MPa) |
|-----------|-----------------------------------|---------------------------------------|
| 1         | 424.81                            | -72.97                                |
| 2         | 434.73                            | -33.76                                |
| 3         | 375.23                            | -20.18                                |
| 4         | 384.85                            | -25.22                                |

From the graph shown in Figure 18 it is clear that the graph of the longitudinal residual stress distribution of this study has the same tendency traits. This property has a maximum residual stress value in the region close to the welding area, and the minimum value is in areas far from the welding.
area. The illustration of the residual stress yield of all variations in the welding sequence has been shown in Figure 19 to 22.

**Figure 19. Illustration of Residual Stress Variation 1**

**Figure 20. Illustration of Residual Stress Variation 2**

**Figure 21. Illustration of Residual Stress Variation 3**

**Figure 22. Illustration of Residual Stress Variation 4**
3.6 Distortion

The results of distortion in this study were conducted on the Z axis in millimeters. In the pathline 1 as in Figure 13, the length of the distortion occurrence begins at the end of the frame 76 to frame 74, or in the direction of the X axis. Figures 23 to 26 have shown graphic distortion after weld simulation in all variations of the welding sequence.

![Figure 23. Graph of Distortion Variation 1](image1)

![Figure 24. Graph of Distortion Variation 2](image2)

![Figure 25. Graph of Distortion Variation 3](image3)
From the graph shown in Figure 23 to 26, it is seen that the maximum distortion value occurs at the free end of the plate (free edge). This is because of the stiffener that serves as a barrier so that no distortion occurs in the area, in this study stiffener is frames. The results of this study, have also been validated by previous studies that show similar things, namely maximum distortion occurs in areas far from welding areas.

Based on the analysis that has been done, it is known that variation 3 has the minimum distortion value, while variation 1 has the greatest distortion value. Table 5. shows the distortion results of all the variations in the welding sequence.

| Variation | The Inner Bottom Edge Near Frame 76 | Location (X = 720 mm) | The Inner Bottom Edge Near Frame 74 | Location (X = 720 mm) |
|-----------|------------------------------------|-----------------------|------------------------------------|-----------------------|
| 1         | 5.74                               | Portside Frame 75     | 13.45                              | Portside Frame 74     |
| 2         | 4.97                               | Starboard Frame 76    | 12.68                              | Starboard Frame 76    |
| 3         | 4.92                               | Portside Frame 75     | 4.20                               | Portside Frame 74     |
| 4         | 7.03                               | Starboard Frame 76    | 5.32                               | Starboard Frame 76    |

Illustration of distortion in all variations have shown in Figure 27 to 30.
4. Conclusion
The following is the conclusion that can be summarized from this research.

a. Heat distribution with the highest peak temperature occurs during welding simulation with the fourth variation with a temperature of 2490 K at the 808.7 second or about 13.5 minutes. While the lowest peak temperature occurs during the welding simulation with a second variation of 2339 K at seconds to the 1243.7 or about 20.7 minutes. This peak temperature has exceeded the material melting point limit of 1800 K.

b. In each variation of the welding sequence, the maximum residual stress with the smallest value is generated by the third welding variation sequence. The maximum tensile residual stress of the third variation is 375.23 MPa, and the maximum compressive stress is -20.18 MPa. Thus, the variation of the welding sequence that can be used to improve the efficiency that is started from the starboard frame 74 to portside frame 76.

c. The distortion with the smallest value that arises after the running of the structural analysis is obtained from the third welding sequence variation as well. The distortion results are reviewed at
X = 720 mm and X = -720 mm. The result of distortion due to variations in the third welding sequence has a large 4.92 mm at X = -720 mm, and 4.2 at X = 720 mm. Thus, the variation of the welding sequence that can be used to improve the efficiency that is started from the starboard frame 74 to portside frame 76.

d. Based on thermal transient analysis and structural transient analysis, the third welding variation has advantages when compared to other variations. Because, the third variation has a welding sequence that leads to inner bottom construction without holes, so that the residual stress distribution and distortion can be spread evenly. As for the welding variations approaching the perforated inner bottom construction, the residual stress distribution and distortion are less evenly distributed, and are concentrated only on certain areas with high residual stress and deformation values.

References

[1] Djamiko, R. D. 2008. Modul Teori Pengelasan Logam. Yogyakarta: Jurusan Pendidikan Teknik Mesin, Universitas Negeri Yogyakarta.

[2] Magga, Ramang. 2009. “Pengaruh Pembentukan Tegangan Sisa pada Hasil Pengelasan”, JIMT. Vol. 6: 54-62.

[3] Putra, Y.A., S. 2007. Analisa Tegangan Sisa dan Distorsi pada Pengelasan Fillet T-Joint dengan Metode Elemen Hingga. Tugas Akhir. Surabaya: Jurusan Teknik Material dan Metalurgi, Institut Teknologi Sepuluh Nopember.

[4] Zaroog, O. S. 2014. “Current and Challenge of Residual Stress Measurement Technique”, International Journal of Science and Research. Vol. 3: 210-216.

[5] Wibowo, B. A. 2016. Analisa Numerik Tegangan Sisa dan Distorsi pada Struktur Plat Berpenegar Jamak dengan Variasi Urutan Pengelasan. Tugas Akhir. Surabaya: Jurusan Teknik Kelautan, Institut Teknologi Sepuluh Nopember.

[6] Wibisono, B. O. B. 2017. Analisa Tegangan Sisa pada Panel Plat dengan Metode Elemen Hingga Menggunakan Jenis Elemen Solid dan Elemen Shell. Tugas Akhir. Surabaya: Jurusan Teknik Kelautan, Institut Teknologi Sepuluh Nopember.

[7] Shen, C. 2013. Low Distortion Welding for Shipbuilding Industry. Thesis Master. Wollongong: School of Mechanical, Materials, & Mechatronic Engineering, University of Wollongong.