ISOMAse
International Society of Ocean, Mechanical and Aerospace Scientists and Engineers

Proceeding of Ocean, Mechanical and Aerospace -Science and Engineering-
5th October 2018. Vol.5 No.1
© 2012 ISOMAse, All rights reserved

October 4, 2018

Analysis of Weld Strength Through Finite Element Method Using Abaqus Program and Comparing with Experiment Testing

Ade Prihantoro[a,*], and Efi Afrizal[a]

[a)] *Department of Mechanical Engineering, Faculty of Engineering, Universitas Riau, Indonesia*

*Corresponding author: ade.prihantoro@student.unri.ac.id.

Paper History

Received: 15-September-2018
Received in revised form: 25-September-2018
Accepted: 4-October-2018

ABSTRACT

The development of production technology and raw materials metals can't be separated from the use of welding technology. This time the connection with the welding method has been widely applied in the field of construction. But in fact the result of the welding field will experience a tensile stress that will cause damage and leakage at the connection. So to anticipate such damages done a test, wherein the test includes testing destructive testing and non-destructive. In this study, the welding method used is welding SMAW (Shielded Metal Arc Welding) and welding FCAW (Flux Core Arc Welding) as the welding process. After the welding process is then performed a simulation using Abaqus software to determine the tensile strength of the weld results prior to mechanical testing. In addition to tensile testing is conducted as for testing hardness testing and testing of micro structure to get the value of roughness and value of the composition at the weld region, HAZ (Heat Affected Zone), and the area outside of the HAZ.

KEY WORDS: *SMAW, FCAW, Tensile Test, and Abaqus CAE.*

NOMENCLATURE

ASTM	American Standard Testing and Material
BHN	Brinell Hardness Number
E	Modulus of Elasticity
σ	Stress
$\mathcal{E}$	Strain
σ_t	True Stress
ε_t	True Strain

1.0 INTRODUCTION

The development of current industrial welding technology is developing rapidly in line with the needs of industry to produce a product that is good enough in the production process, for welding technology here is very helpful in the work of construction manufacture both simple and constructions that have this level of difficulty and the high requirements [11]. In addition to the welding process, in making good construction also use bolts and rivets as a connector.

In welding technology, other than can be used to connect and cut metal, can also be used to fill holes in castings, make hard coating on tools, thicken worn parts, and various other repairs [10].

But in reality the results of the weld when in the field experience a tensile stress which results in the cracking of the weld and the worse will result in the welding experiencing a leak or break.

From these problems, the welding results must pass a series of tests to anticipate the occurrence of leakage in the weld results in order to find out how large the weld strength is if given a stress before application in the field later.

As for testing conducted to anticipate the occurrence of leakage in the connection by doing destructive testing and non-destructive testing. Damage tests include tensile testing, impact testing, buckling testing, hardness testing and others, while for non-destructive tests include penetration testing, radiographic testing, ultrasonic testing and others.

In this study will use the welding method of SMAW (Shielded Metal Arc Welding) and FCAW (Flux Core Arc Welding) as the welding process. And for the mechanical testing method used is tensile testing to get how much strength the weld results. And hardness testing is used to see how much the value of hardness found in the HAZ area and for microstructure testing is used to see the chemical composition found in the HAZ region, The Heat Affected Zone (HAZ) is a base metal adjacent to the weld metal during the welding process undergoes a rapid thermal heating and cooling cycle.

The purpose of this study is to do a simulation test using Abaqus CAE software which is then tested experimentally. From the experimental test, the results of the test where the specimen is approaching the simulation is then analyzed.

2.0 BASIC THEORY

2.1 Abaqus CAE

Abaqus is a suite of powerful engineering simulation programs, based on the finite element method, that can solve problems ranging from relatively simple linear analyses to the most challenging nonlinear simulations. Abaqus contains an extensive library of elements that can model virtually any geometry. It has an equally extensive list of material models that can simulate the behavior of most typical engineering materials including metals, rubber, polymers, composites, reinforced concrete, crushable and resilient foams, and geotechnical materials such as soils and rock. Designed as a general-purpose simulation tool, Abaqus can be used to study more than just structural (stress/displacement) problems [2].

The Abaqus finite element system includes:

1) Abaqus Standard a general purpose finite element program.
2) Abaqus Explicit an explicit dynamics finite element program.
3) Abaqus CFD a general-purpose computational fluid dynamics program.
4) Abaqus CAE an interactive environment used to create finite element models, submit Abaqus analyses, monitor and diagnose jobs, and evaluate results; and
5) Abaqus Viewer a subset of Abaqus CAE that contains only the post processing capabilities of the Visualization module [3].

Computer Aided Engineering (CAE) is one technology that uses a computer system to analyze the functions of a Computer Aided Design (CAD) design product, which allows designers to simulate and study how the product will behave so that the design can be refined and optimized. From the analysis, it is expected that the resulting design will be optimal and can be used in the next process. CAE uses several analytical methods, one of which is Finite Element Analysis (FEA) or Finite Element Method. This method is one of the popular methods used in conducting analysis.

Finite Element Analysis (FEA) or Finite Element Method is a method used to determine stress, deformation, heat, fluid transfer and other physical effects. This element is used to solve problems that are difficult to solve by other methods. This analysis is used to indicate whether there is a problem in a product, for example, it is broken, worn or the product has been good or not. The output of this analysis is the result of predictions that will occur when the product is processed and used. Software that can be used to use this method is Abaqus [4].

2.2 Finite element formulae for the tensile test

From the results of a tensile test, in the form of the nominal stress–strain curve, can directly obtain the modulus of elasticity, the yield strength, the tensile strength, and the maximum elongation of a material.

Neglecting the volumetric change during plastic deformation, we obtain the relationship between the true stress σ_t and strain ε_t, and the nominal stress σ_e and strain ε_e, as follows:

$$\varepsilon t = ln(1 + \varepsilon e) \tag{1}$$

$$\sigma t = \sigma e(1 + \varepsilon e) \tag{2}$$

Eq. (2) is valid before necking occurs and thus the true stress–strain curve up to the necking point can be obtained from the incompressibility assumption during plastic deformation.

The flow stress in a metal forming simulation is formulated by the following strength coefficient strain hardening exponent model, called Hollomon's constitutive law, or Hollomon's law:

$$\sigma_t = K\varepsilon_t^n \text{ or } \overline{\sigma} = K\varepsilon^{-n} \tag{3}$$

Where and are effective stress and effective strain, respectively, and the strength coefficient K and strain hardening exponent n are determined by minimizing the error between the experimental results and the mathematical model of Hollomon's law.

It should be noted that the finite element simulation of the tensile test with material data found in related literature cannot exactly predict the necking point in the engineering sense. There are two reasons for this. The first results from the finite element formulation and its solution scheme and the second from the inconsistency between the true strain at the necking point and the strain hardening exponent.

It is known that necking takes place when the true strain reaches the strain hardening exponent according to the considere criterion.

The reference strain-hardening exponent denoted as n_N is defined as the true strain at the necking point, as follows:

$$nN = ln(1 + \varepsilon_e^N) \tag{4}$$

Where ε_e^N is the nominal strain at the necking point. In addition, the reference strength coefficient denoted as K_N is defined by making the flow stress curve of Eq. (4) pass through the necking point in the true stress–strain curve. The reference strength coefficient can be found from:

$$K_N = \sigma_e^N(1 + \varepsilon_e^N)/[ln(1 + \varepsilon_e^N)]^{ln(1+\varepsilon_e^N)} \tag{5}$$

Where σ_e^N is the nominal stress at the necking point. Therefore, the following Hollomon's constitutive law is known as the reference stress–strain curve:

$$\overline{\sigma} = K_N\overline{\varepsilon}^{-nN} \tag{6}$$

With the reference stress–strain curve, the necking point should be predicted when the true strain reaches the reference strain-hardening exponent n_N if the correct analysis tools are employed.

Tensile specimen as the analysis domain, sometimes called a full analysis model, it is difficult to trace the deformation between the gauge marks on the tensile specimen, as it is not trivial to construct a structured finite element mesh system with nodes on the gauge marks. An important factor in tensile testing is the deformation between the gauge marks [6].

2.3 Welding

According to the Deutsche Industrie Normen (DIN) welding is a metallurgical bond on the alloy metal joints carried out in a liquid state from the definition can be explained further that welding is a process in which the same type of material is combined into one so that a connection is formed through the chemical bonds produced from the use of heat and pressure [12]. As for the welding method that will be used as follows:

1) Shielded Metal Arc Welding (SMAW)

Shielded Metal Arc Welding is a protected electric arc welding where heat is generated from an electric arc between the tip of the electrode and the welded metal. Electrodes consist of loose electric current conductor metal wire and as a filler material [12]. Which can be seen in Figure 1.

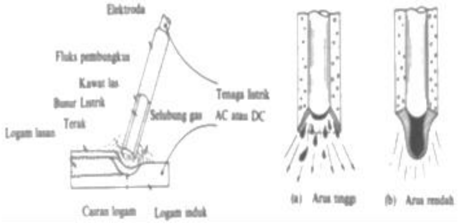


Figure 1: Shielded Metal Arc Welding

The principle of shielded metal arc welding (SMAW) is that it starts when the electric flame touches the tip of the electrode with the work piece. Two conductive metals if electrified with a relatively low voltage will produce electron jumps which cause very high heat, can reach 5000°C which can melt the two metals [9].

2) Flux cored arc welding (FCAW)

Flux cored arc welding (FCAW) is one of the welding techniques using an automatic process that utilizes wire roll electrodes to melt metal.

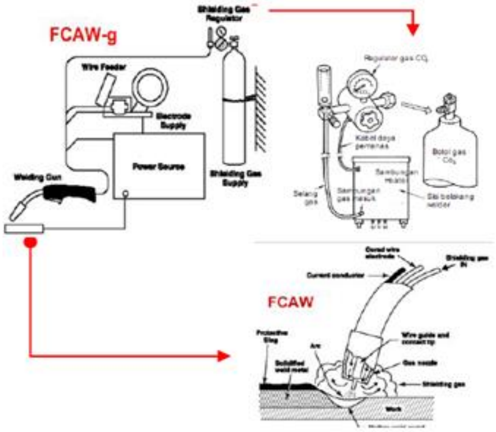


Figure 2: Process Flux Core Arc Welding

This technique has a number of advantages over general welding techniques because this technique has better control and low tensile welding properties [8]. Figure 2 describes the process when welding FCAW:

2.4 Testing Methods

Testing methods on weld results are subject to destructive testing and non-destructive testing. The destructive testing as follows:

1) Tensile test

Tensile testing gives engineers much information about the mechanical properties of materials, including the modulus of elasticity, yield strength, tensile strength, elongation, and the true stress–strain relationship before necking takes place. Therefore, it is important to understand the mechanical and metallurgical phenomena occurring during the tensile testing of materials.

Necking is one factor that characterizes the tensile test. Necking usually occurs at the same elongation for different tensile test specimens, while the necked positions of the pulled specimens differ from specimen to specimen. Thus necking phenomena are bifurcation or instability problems, such as resonance and buckling in structural engineering [6].

According to [1] to calculate Engineering Stress of a material using the following formula:

$$\sigma = \frac{F}{A_o} \tag{7}$$

To calculate Engineering Strains of a material by using the following formula:

$$\in = \frac{l_i - l_o}{l_o} = \frac{\Delta l}{l_o} \tag{8}$$

To calculate the modulus of elasticity of a material by using the following formula:

$$E = \frac{\sigma}{\in} \tag{9}$$

To calculate true stress of a material by using the following formula:

$$\sigma_T = \frac{F}{A_t} \tag{10}$$

To calculate the true strain of a material using the following formula:

$$\in_T = ln \frac{l_i}{l_o} \tag{11}$$

So to calculate true and engineering stress-strains of a material by using the following formula:

$$\sigma_T = \sigma(1{+}\in) \tag{12}$$

$$\in_T = ln(1{+}\in) \tag{13}$$

2) Hardness test
Hardness testing was carried out to find out the changes in violence that occurred after welding both in the weld area, HAZ, as well as the parent metal [12].
The method used in the hardness test is the Brinell method using the ASTM E10 testing standard that in hardness testing uses a 10 mm steel ball indenter with varying loads where at 3000 Kgf load is used for hard metal, 1500 Kgf load is used for intermediate hardness and load 500 Kgf is used for soft material with loading given for 30s which is then measured using a microscope. To calculate the value of brinell hardness by using the following formula:

$$(BHN) = \frac{2P}{\pi \times D\left[D-(D^2-d^2)^{\frac{1}{2}}\right]} \qquad (14)$$

3) Microstructure testing
The properties of metals are strongly influenced by microstructure. The purpose of this test is to obtain a description of a test object about its properties. Forms of certain structures or characteristics in order to analyze other properties possessed by a test object with the standard used by ASTM E3.

3.0 METHODOLOGY

3.1 Research Stages

The steps carried out in this study can be seen in the flow chart as follows:

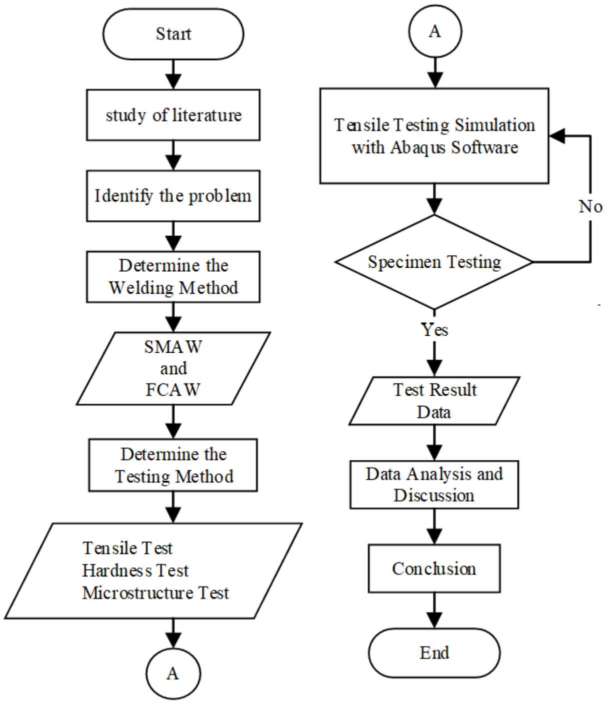


Figure 3: Flow chart

3.2 Prosedure Abaqus CAE

The procedure demonstrates how to generate components, material properties, boundary conditions and forces to create a Finite Element model using Abaqus CAE software:

1. Create the Part Model as per the Dimensions and Shape given above. In the Part Manager dialog box, click on Create. In the Create Part dialog box, give the name of the sample as part 1. Select 3D, Deformable, Solid, Extrusion, and Approximate size as 200. Click Continue. Sketch the Part.
2. Give the properties of the part 1 material in the Material Manager option. The properties that have to be given are density, young's modulus, poisson's ratio, yield stress and plastic strain as shown in table 2 and 3.
3. In the Assembly Module, select the Create Instance tool. In the dialog box, select the part, click the dependent option and then click OK.
4. In the Step Module, different steps have been created. Step Manager, Field Output Manager and History Output Manager are provided.
5. In the Load Module, one end of the sample is made fixed and the axial load is provided in the other end.
6. Proceed to Mesh Module. Click on the Seed Part icon. Accept the defaults in the Global Seeds dialogue box. Click done. Click on the Mesh Part Instance icon, at the bottom of the screen select yes.
7. In the Job Module, click on 'create job' icon. Name the job as Tensile Test. Click Continue.
8. Click on Job manager. Click 'Submit' to submit the job to the solver. Click Monitor to check the analysis progress.
9. Once the job is complete, click results to check the results. Select the icon to display contours. To examine different stresses etc, select Result, Field Output from the main menu bar [5].

3.3 Tensile Test

In tensile testing to be carried out using ASTM E8 testing standards with the following dimensions [5]:

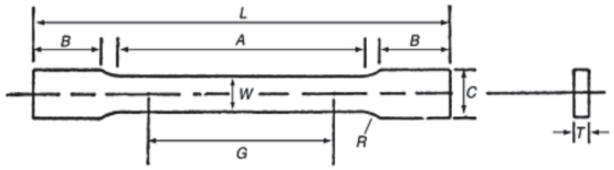


Figure 4: Specimen Tensile Test

Table 1: Dimension of Sample Tensile Test

No.	Deskripsi	Ukuran (mm)
1	Gauge length (G)	203
2	Width (W)	35
3	Thickness (T)	10
4	Radius of fillet (R)	25
5	Overall length (L)	457
6	Length of reduced section (A)	229
7	Length of grip section (B)	76
8	Width of grip section (C)	51

Material property data is assumed as follows:

Table 2: Property of Material plate

Density	7800 kg/m^3
Young’s Modulus	10833 MPa
Poisson’s Ratio	0.3
Displacement at Failure	0.06
Yield Stress	325 MP
Plastic strain	0.03

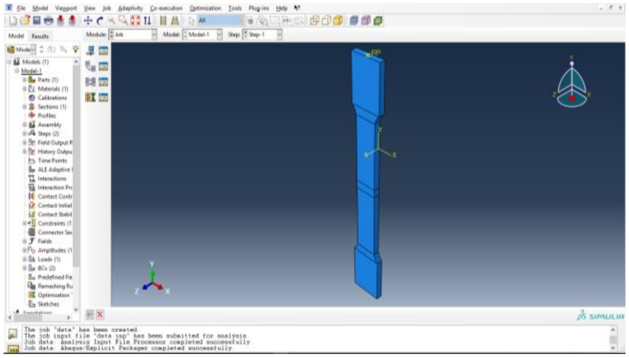
Table 3: Property of Material Weld

Density	7800 kg/m^3
Young’s Modulus	13833 MPa
Poisson’s Ratio	0.3
Displacement at Failure	0.06
Yield Stress	415 MP
Plastic strain	0.03

Where before the tensile testing was carried out experimentally, then the tensile testing simulation was performed using the Abaqus CAE Student software. And after that the hardness testing was carried out by using the brinell method with ASTM E10 standard and microstructure testing to see phase changes in the HAZ and outside the HAZ area using the ASTM E3 standard.

4.0 RESULTS AND DISCUSSION

As before doing the experimental tensile testing, a simulation test was performed using Abaqus CAE software**,** which can be seen in the picture below:

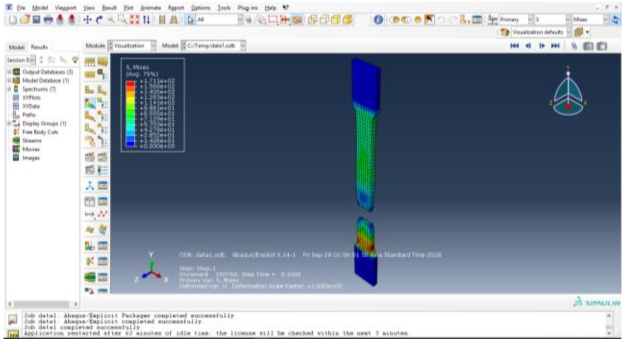


```
The job 'data' has been created.
The job input file 'data.inp' has been submitted for analysis.
Job data: Analysis Input File Processor completed successfully.
Job data: Abaqus/Explicit Packager completed successfully.
```

Figure 5: Specimen Before Being Given a Tensile Force

Figure 5 shows the specimen before being given a tensile force, the specimen is divided into 3 components consisting of plates and welded joints, where the components have different material properties. Usually the welded joint has a greater strength value compared to the base plate.

Figure 6 shows the broken specimen after being given a tensile force of 500 N. It can be seen on the specimen having broken not in the middle part, but on the base plate because the value of the tensile strength of the plate is lower than that of the welded joint and this can occur in the welding process experience uneven cooling and heat treatment.



```
Job data1: Abaqus/Explicit Packager completed successfully.
Job data1: Abaqus/Explicit completed successfully.
Job data1 completed successfully.
Application restarted after 62 minutes of idle time: the license will be checked within the next 3 minutes.
```

Figure 6: Specimen After Being Given a Tensile Force

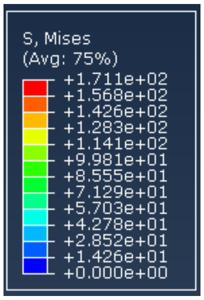


Figure 7: Value of Mises Stress

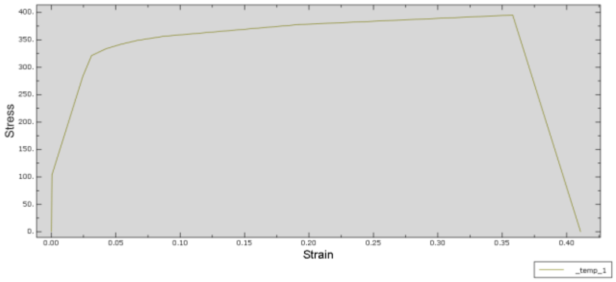


Figure 8: Graph of Stress-Strain

Figure 8 shows a linear stress-strain graph up to a stress value of 356 MPa and a strain value of 0.08, then shows a maximum stress value of 382 MPa and a value of strain 0.23 which then undergoes necking and finally it breaks.

5.0 CONCLUSIONS

Tensile test on the plate which is connected using a welding is broken in the plate, this happens because the strength value is higher than the strength of the base plate. From the simulation test using the Abaqus CAE software that the specimen has a fracture on the base plate. And the simulations carried out follow the law Hooke's where the voltage is directly proportional to the strain which then occurs necking and then breaks.

ACKNOWLEDGEMENTS

The authors would like to convey a great appreciation to Ocean and Aerospace Engineering Research Institute, Japan for supporting this research.

REFERENCE

1. Callister, William D.2007.'Material Science & Engineering an Introction'. Jhon Willey & Sons.
2. Dassault Systèmes.2008.'Getting Started with Abaqus Interactive Edition version 6.8'. Dassault Systèmes Simulia Corp.
3. Dassault Systèmes.2014.'Abaqus 6.14 Example Problems Guide Volume I: Static And Dynamic Analyses'. Dassault Systèmes Simulia Corp.
4. Gunawan, Fransiskus Andre Cahya. 2016. '*Analisis Finite Elemen Pada Material EVA Rubber Dua Layer Pada Aplikasi Insole Sepatu Orthotic*'. Thesis . Program Studi Teknik Industri Universitas Atma Jaya Yogyakarta.
5. Jose, Edgar and Tom Anto.2015.'Analiysis of Tensile Test of Mild Steel using Finite Element Method'. Internasional Journal of Innovations in Engineering and Teknology (IJIET). Vol. 5.
6. Joun, Mansoo, Inso Choi, Jaegen Eom, and Mincheol Lee.2007.'Finite Element Analysis of Tensile Testing with Emphasis on Necking'. Computional Material Science 41, page 63-69.
7. Kurniawan, Ary Setya, Solichin, and Rr. Poppy Puspitasari. 2014.' Analisis Kekuatan Tarik Dan Struktur Mikro Pada Baja St.41 Akibat Perbedaan Ayunan Elektroda Pengelasan SMAW'. Teknik Mesin Universitas Negeri Malang. Jurnal Teknik Mesin, tahun 22, No. 2.
8. Kusumawardani, Meilia, Mustafid, and Hasby Yasin. 2015. ' Optimalisasi Parameter Teknik Pengelasan Flux Cored Arc Welding (FCAW) Menggunakan Metode Taguhchi Multi Respon PCR-Topsis. Jurnal Gaussian, Vol. 4, No. 3, 573-582.
9. M, Imam Pujo, and Sarjito J.S., 2008, 'Analisis Kekuatan Sambungan Las SMAW (Shielded Metal Arc Welding) Pada Marine Plate St 42 Akibat Faktor Cacat Porositas Dan Incomplete Penetration', Universitas Diponigoro. KAPAL, Vol. 5, No.2.
10. Naharuddin, Alimuddin Sam, and Candra Nugraha., 2015.' Kekuatan Tarik Dan Bending Sambungan Las Pada Material Baja Sm 490 Dengan Metode Pengelasan SMAW Dan SAW'. Teknik Mesin Universitas Tadulako Palu. Jurnal Mekanikal, Vol. 6 No. 1.
11. Sisworo, Sarjito Joko. 2012.' Pengaruh Perbedaan Posisi Pengelasan Terhadap Kekuatan Sambungan T-Joint Pengelasan Fillet Dengan Las FCAW Pada Plat Mild Steel'. Teknik Perkapalan Universitas Diponogoro.Suharno.2008.'prinsip-Prinsip Teknologi dan Metalurgi Pengelasan Logam'. Lembaga Pengembangan Pendidik-an (LPP) dan UPT Penerbitan dan Pencetakan UNS (UNS Press) Universitas Sebelas Maret. Surakarta.