FEM analysis of the HAZ temperature by heat source modeling on butt-joint process using msc marc mentat

Charles SP Manurung[1*], Himsar Ambarita[1*], Yupiter H.P. Manurung[2], Farel Napitupulu[1], Richard AM Napitupulu[3].

[1]Department of Mechanical Engineering, University of North Sumatera, Medan
[2]Faculty of Mechanical Engineering, UiTM Shah Alam, Malaysia.
[3]Department of Mechanical Engineering, Nommensen HKBP University, Medan

*charles.manurung@uhn.ac.id

Abstract. This paper uses the Finite Element Method software to simulate the welding method, namely MSC Marc. The selected material is structural steel type C15 categorized as low carbon steel. The material has a thickness of 2 mm and butt-joint welding will be performed, which needs additional attention because welding is highly susceptible to distortion on thin plates. It can be seen that the simulation results against the sample SM 4PL Low have a lower temperature from the heat distribution data taken about 1 mm from the fusion line. Whether or not distortion happens on the plate is significantly affected. This information can be used to predict the real parameter of welding. However, it is better to ensure the best parameters; this data must be calibrated with the results of thermal distribution measurements through experimental methods.

1. Introduction

The success of a welding process is the occurrence of good and strong weld joints. This is inseparable from the influence of the heat source for the process of melting the parent metal for the formation of welded joints. The problem that often occurs when welding is distortion. Distortion in addition to reducing form perfection, also decreases the strength of welded joints and can be the initial formation of cracks.

The distortion in the welding process is produced from thermally induced thermal stresses that exist in welded joints that are not retained. The three main forms of distortion that are usually found in the welding process are: (1) longitudinal shrinkage that occurs in parallel directions with the weld line, (2) the transverse shrinkage that develops in a direction perpendicular to the weld line and (3) the angle changes due to rotation around weld line. The configuration of the weld connection, heat input during transfer, and welding sequence are factors that can affect the end of the distortion [1-2]. Heat-Affected Zone (HAZ) is an area where the micro structure changes due to heat fusion during welding. This also causes distortion due to expansion and shrinkage that occurs in a short time.

The welding process has a very broad classification starting from the type of material being welded, the welding conditions whether liquid or solid, the method, the welding process, the filler material and others [3]. The welding process requires quite high costs, starting from the availability of materials, welding equipment, metal fillers, shielding gases and others. If the welding process results are not perfect, it will certainly be a lot of wasted costs. By conducting repair welding process, it will lead into a quite sums of schedule delay and additional cost. By predicting the degree od welding distortion beforehand, engineer also could reduce the number of reworks by means of reverse design and distortion margin.

That is why with the development of technology today it is possible to carry out simulations based on analytical solutions that are implemented through the use of software. Therefore, the modeling process is designed to be able to obtain an approach to the actual welding process based on the related analytical solutions.

2. Methodology

Simulation Procedure using Nonlinear FEM

Welding simulation making should consider the relationship between technologies in terms of the resistance used, thermal, mechanical and metallurgical aspects in one numerical model. This is intended to obtain simulation results that are close to the original state of the welding process. In general, making simulations using the Marc Mentat MSC can be seen in the following process :

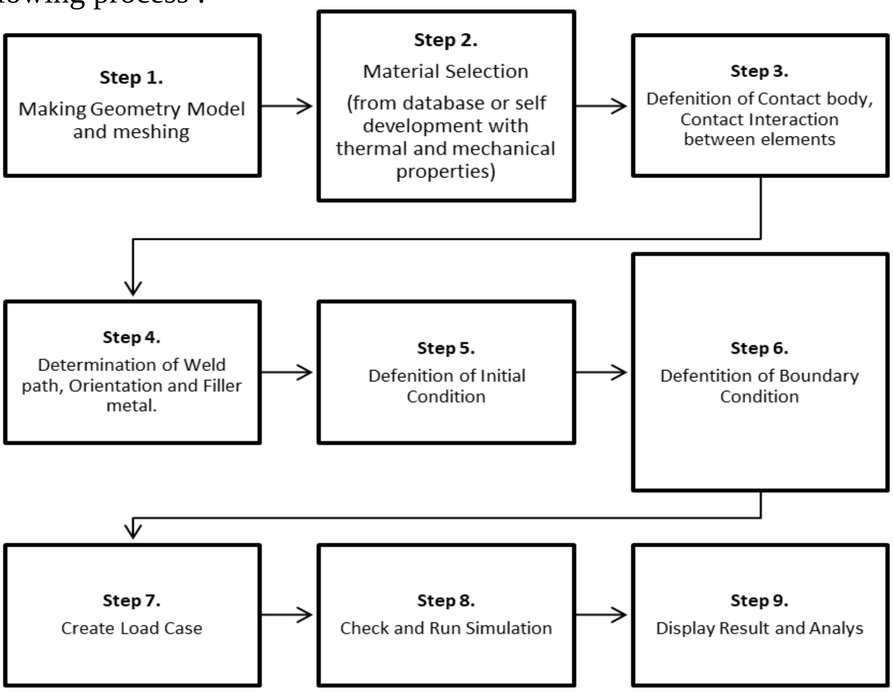

Figure 1. General Flowchart of Simulation Procedure using

Geometrical and Material Description

Figure 2 shows the schematic illustration of the geometrical model, which have the dimension of 50 x 55 mm x 2 plate with V Groove, the thickness of plates is 2 mm, which was assigned to produce single-sided weld. In this simulation, the geometry is meshed using single-passed welding bead.

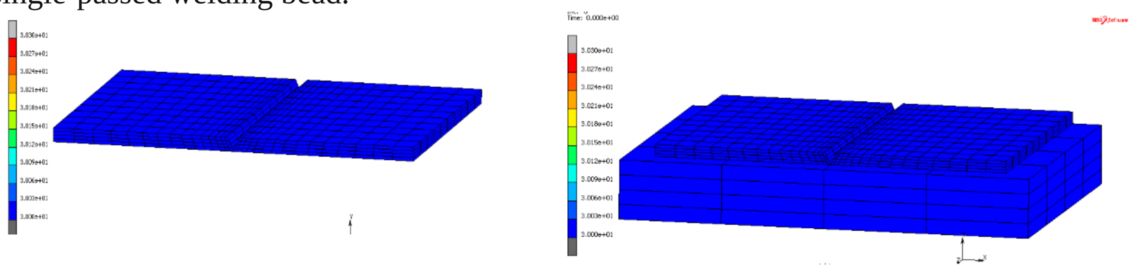

Figure 2. Finite element models of 3D-Solid V Groove Butt-joint without table (left) and 3D-Solid V Groove Butt-joint with table (right)

In this simulation, the C15 steel is selected as material for the T-joint plates and filler material. The physical properties of the material are shown in Figure 3.This figure shows that the material assigned for the FEM simulation has temperature-dependent variables.

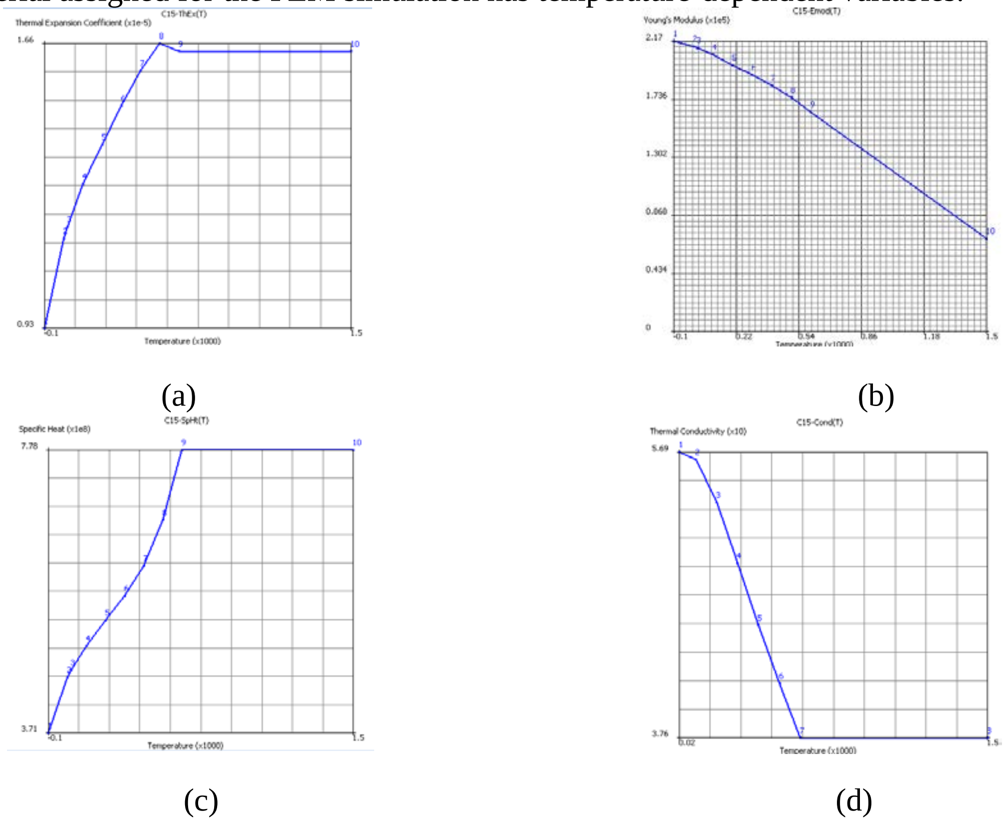

Figure 3. Temperature-dependant thermo-physical properties of C15: (a) Thermal Expansion Coefficient, (b) Thermal Conductivity, (c) Young´s Modulus and (d) Specific Heat.

Table 1. Chemical Composition of Material

Elements	C15 (Material Data Sheet)
C	0.12 – 0.18
Mn	0.3 – 0.6
Si	< 0.4
S	< 0.045
P	< 0.045

Assigning Contact Body and Interaction

In this FEM Simulation, Contact Bodies are set prior determining the contact interaction of all the elements. Elements in this simulation are 2 Plates, Weld Filler and Table. Deformable type of contact is assigned to both weldment and filler while rigid type of contact to table. This applies for both Shell and Solid model. In this FEM simulation, the deformable contact type influenced by the result of the simulation after the job has been set, the alteration could be seen implicitly during thermomechanical analysis where the surface transformation occurs while running the results. While rigid contacttends to have more static surface during the executionbecause rigid body mode is defined as the free translation or rotation of a body without undergoing any significant internal deformation. There are two types of contact interaction namely “Touching/T” and “Glued/G”. These two contacts are available in MSC Marc/Mentat located on contact bodies section.

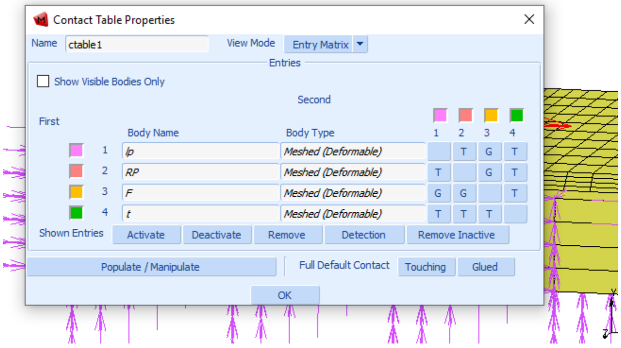

Figure 4. Contact interactions of elements (G: Glued, T:Touching)

Initial and Boundary Conditions

The initial and boundary conditions of a finite element model includes thermal and mechanical boundary conditions, in which the main consideration of the thermal boundary conditions is heat radiation, heat conduction and convection thermal cooling. In this simulation, the analysis of bondaries are divided into Thermal and Structural analysis. The thermomechanical parameters are defined in each sections of boundary condition on both shell and solid model.

The structural analysis for this bounndary consits of fixed displacement and structural point load. Both are the features of structural analysis provided by MSC Marc/Mentat.

The thermal boundary condition in this simulations are Thermal Face Film and Thermal Volume Weld Flux. The Thermal Face Film is implemented in the same procedure for both Shell and Solid model of V Groove Butt-Joint while Thermal Volume Flux has one of the two models differs from each other. In this case, the heat to the surrounding of weld filler (vicinity face weld flux) was assumed as nothing.The amount of heat input that being used was 1.5 $x10^6$ and $1.3x10^6$ N*mm/s and the amount of efficiency being used is 80%. The velocity of weld speed matches with the experimental parameter which is 5 mm/s which would be later accompanied by weld path and filler assignation. The Current (I); and the Voltage (V) are considered under the equation of power. The assigned travel speed (v) was based on the ideal welding speed for this material using one-pass V-Groove Butt-joint.

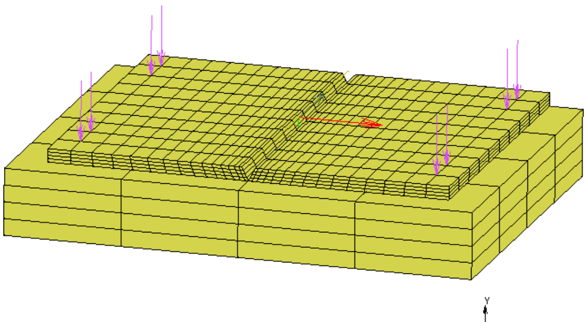

Figure 5. Clamping position of structural boundary condition

In MSC Marc/Mentat, heat source generation is set while assigning the Volume Weld Flux boundary condition in thermal analysis. The Goldak’s Double Elipsoid Model is choosen as the heat source model for this simulation. The Double Ellipsoid Model that mostly used to represent the heat which is made by the torch in GMAW welding. Double Ellipsoid means that the heat source is consisted of two elliptic regions, one in front of the arc and one is in the centre.Z>0 and the other behind the arc centre is Z < 0[4]. The Figure 6 illustrates the Goldak’s Double Ellipsoid model.

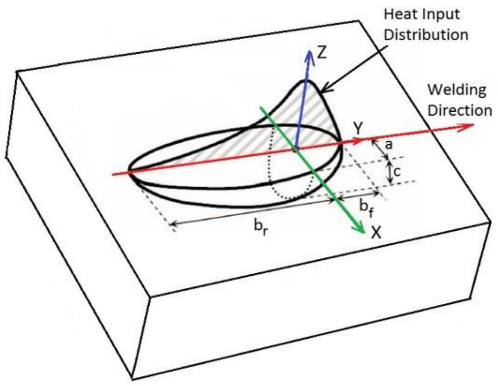

Figure 6. Illustration of Goldak’s Double Ellipsoid Heat Source Model

The power density of the heat flux in front section (Qvf) of heat source can be determined by following formula (Eq 1):

$$q_{vf}(x,y,z) = \frac{6\sqrt{3}.f_f Q}{a c_{bf}.\pi\sqrt{\pi}} \; .e^{-3\frac{x^2}{a^2}} \; .e^{-3\frac{y^2}{c^2}} \; .e^{-3\frac{z^2}{b_f^2}} \qquad (1)$$

And the power density of the heat flux in rear section (qvr) can be determined by (Eq 2):

$$q_{vr}(x,y,z) = \frac{6\sqrt{3}.f_r Q}{a c_{br}.\pi\sqrt{\pi}} \; .e^{-3\frac{x^2}{a^2}} \; .e^{-3\frac{y^2}{c^2}} \; .e^{-3\frac{z^2}{b_r^2}}$$

Where f_f and f_r are the heat deposited fractional factors in the front and rear quadrant respectively and its sum is equal to 2. The distribution of fluxes in the double ellipsoid model is determined by 4 directions: Width (b), Depth (d), Rear Length (ar) and Front Length (af). In this simulation, there are several value for Heat source direction, but in this paper only the two best variable categories are displayed as shown in Table 2.

Table 2. Heat Source Dimension in FEM Simulation

Heat Source Direction	Value	
	SM_4PL	SM_4PL_Low
Width (mm)	2	1.8
Depth (mm)	1.5	1.4
Rear Length (mm)	0.8	0.7
Front Length (mm)	0.8	0.8

3. Result and Discusion

From the simulation results, it can be known that better parameters to be done on the experimental. This is concluded based on the comparison seen between the 2 best parameters of the simulated parameters.

The picture shown below is a comparison of the Temperature, Displacement and Effective Stress variables on the Increment observed for each of the same nodes.

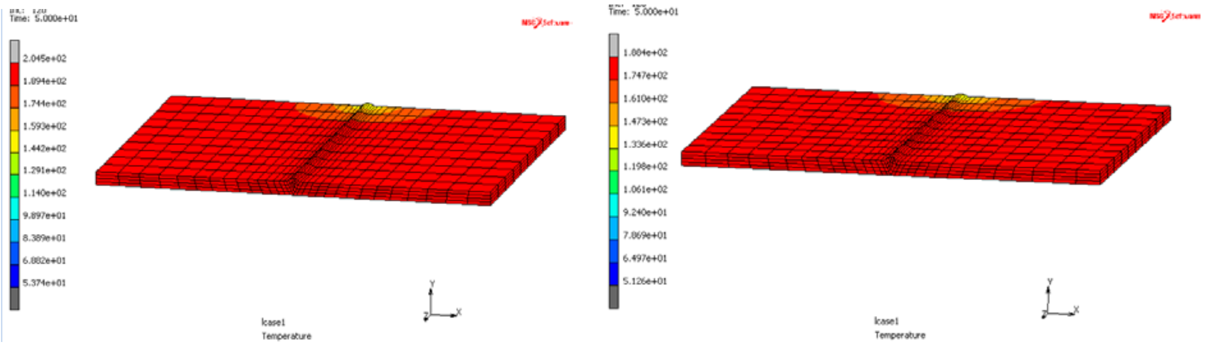

Figure 7. Comparison of Temperature to Increment from 2 parameters SM_4PL (left) and SM_4PL_Low (right)

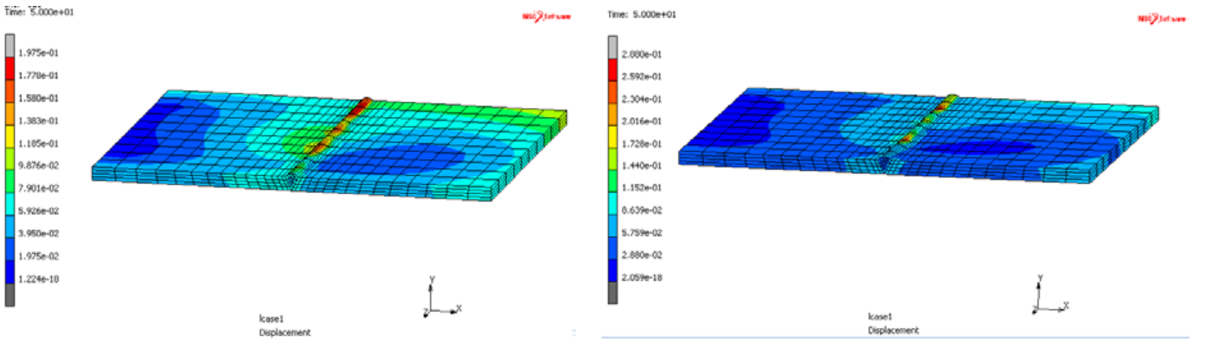

Figure 8. Comparison of Displacement to Increment from 2 parameters SM_4PL (left) and SM_4PL_Low (right)

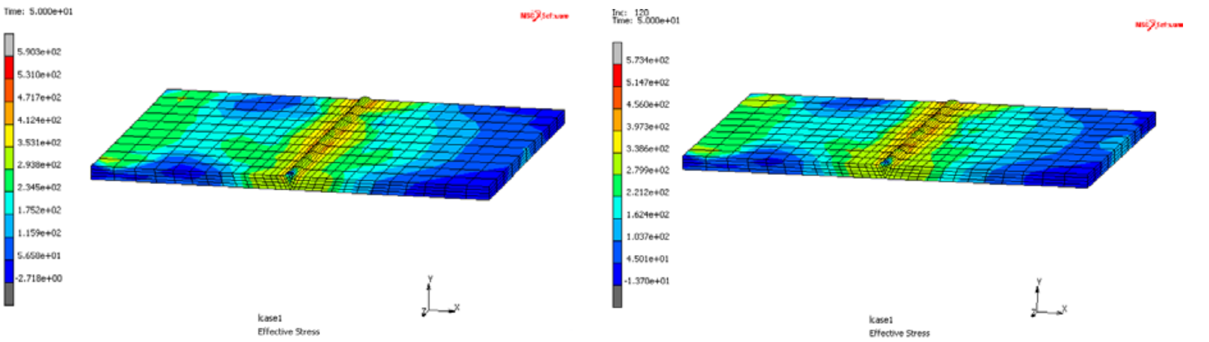

Figure 9. Comparison of Effective Stress to Increment from 2 parameters SM_4PL (left) and SM_4PL_Low (right)

From Figures 7 to 9, we can see the difference in conditions of temperature, displacement and effective stress taken on the same node in the HAZ region.
The comparison of the simulation results from the image above can also be seen in the following curves:

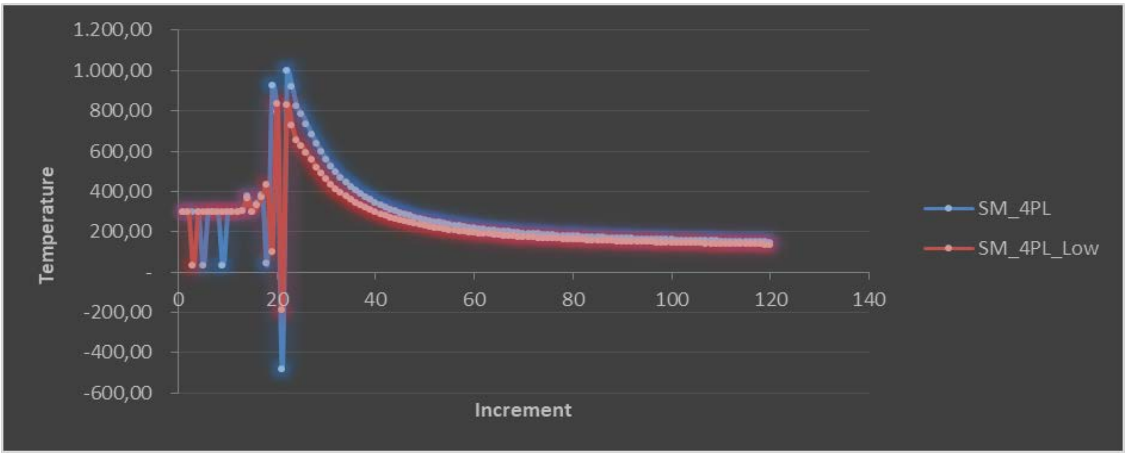

Figure 10. Comparison of Temperature to Increment from 2 parameters

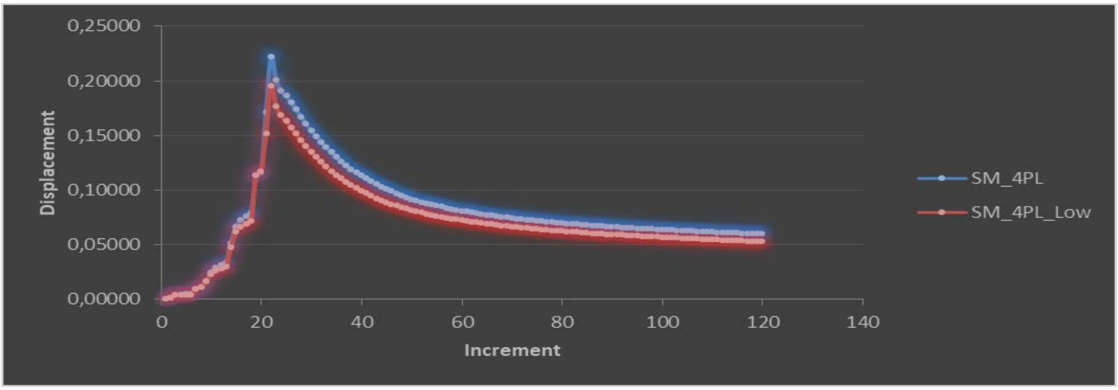

Figure 11. Comparison of Displacement to Increment from 2 parameters

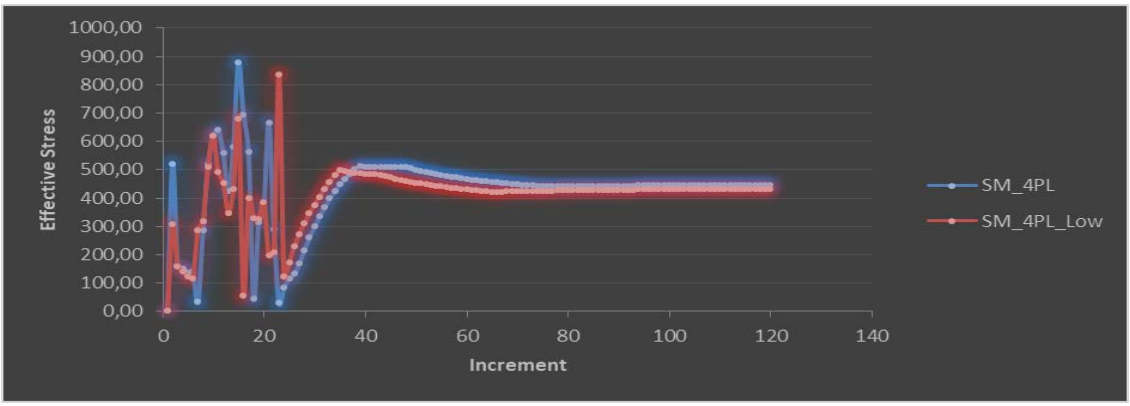

Figure 12. Comparison of Effective Stress to Increment from 2 parameters

From the pictures above it can be seen that the heat source is very determining the occurrence of distortion and stress on the surface. This is caused by the phenomenon of melting and shrinkage that occurs in a short time. In the main areas of welding such as weld metal, HAZ and parent metal areas, microstructure changes occur. In the weld metal area, melting and solidification occur where pure recrystallization occurs in the process, whereas in the HAZ area micro-structure changes occur due to high temperatures which also cause grain growth without recrystallization. This phenomenon will cause distortion in the HAZ area.

4. Conclusion

Through the Finite Element Methods, it can be concluded that the modeling and simulation has been successfully carried out and the following things can be conveyed:

1. The use of Finite Element software is very helpful in resolving to get better welding parameters compared to the trial and error method in the actual welding process.
2. By using software, the efficiency and effectiveness of the work is better, this is caused by the price of material which is relatively expensive and limited working time is only done in the welding workshop.
3. With the results of the welding parameters obtained, especially in the temperature distribution, the mechanical properties that will be generated from the welding process can be predicted. Temperature conditions contribute to the micro structure changes, both recrystallization and grain growth. Grain growth to an undesirable level will reduce mechanical properties according to the Hall-Petch formula about the relationship of grain diameter to yield strength.

References

[1] K. P. Prajadhiana *et al.*, “Comparative distortion analysis of welded T-Joint between 2D-shell and 3D-solid element using FEA with experimental verification,” *J. Mech. Eng.*, vol. 5, no. Specialissue2, pp. 99–115, 2018.

[2] D. Sebayang *et al.*, “Numerical simulation of distortion and phase transformation in laser welding process using MSC Marc/Mentat,” *IOP Conf. Ser. Mater. Sci. Eng.*, vol. 453, no. 1, 2018.

[3] R. A. M. Napitupulu, S. L. M. H. Simanjuntak, C. Manurung, C. Hutabarat, S. Sitompul, and J. N. Aoh, “Friction stir welding of aluminium alloy 6061-t651,” *IOP Conf. Ser. Mater. Sci. Eng.*, vol. 508, p. 012064, 2019.

[4] MSC Software, “Volume A: Theory and user information,” *MSC.Software Corp.*, 2016.