Finite Element Analysis Of AA5083 In GTAW Process

Balakrishnan.A [1], Ajithkumar.K [2], Karthikeyan.K[3]
Department of Mechanical Engineering
[1, 2, 3]UG Scholer,Sri Shanmugha College of Engineering & Technology , Salem,Tamil Nadu,India

Abstract-*Welding is widely used manufacturing process for the development of structural components. During the welding process weld distortion and residual stress are the major issues. Particularly the scope of residual stress has to be taken to the consideration of HAZ, size of weld bead, base metal. Measuring of residual stress is complex and also time consuming. The HAZ can control by the help of temperature distribution. In this Tungsten Inert Gas welding on Aluminium Alloy 5083 (AA5083) has taken for the analysis. First experimental analysis is carried out and the temperature value is finding by using thermocouple. Further the model of the material is simulated by Finite element analysis in ABAQUS software for the different pass in welding and finds the temperature distribution and for avoiding the difficult in measuring the residual stress it is simulated by the Finite Element Method in ABAQUS. Hence the experimental value of temperature distribution at different pass is compared with the FEA value. From the results the experimental Value closes to the Finite element Simulation Results.*

Keywords- Welding, Finite Element Analysis, Coupled Temperature Displacement Analysis

I. INTRODUCTION

Welding is defined by the American Welding Society (AWS) as a localized coalescence of metals or non-metals produced by either heating of the materials to a suitable temperature with or without the application of pressure, or by the application of pressure alone, with or without the use of filler metal. There are various welding processes used in industry today, the main factors for their distinctions being the source of the energy used for welding, and the means of protection or cleaning of the welded material.

During the welding process high amount of heat produced. The main objective of this simulation is the determination of temperatures and stresses during and after the process. Temperature distribution defines the heat affected zone where material properties are affected. Due to the high temperatures introduced during welding and the subsequent cooling of the welded metal,

Welding can produce undesirable residual stresses and deformations. Residual Stresses are those stresses that would exist in a body if all external loads and restraints were removed. Welding residual stresses are generated in a structure as a consequence of local plastic deformations introduced by local temperature history consisting of a rapid heating and subsequent cooling phase. During the welding process, the weld area is heated up sharply compare to the surrounding area and fused locally. The material expands as a result of being heated. The thermal expansion of the material is restrained by the surrounding cooler area, which gives rise to thermal stresses. The thermal stresses are partly exceeds the yield limit, which is lowered at elevated temperatures. Welding residual stresses may be very complex and their distribution is very difficult to predict. Many techniques have been used for measuring residual stresses in metals including stress relaxation techniques,

Hole method, diffraction techniques, cracking techniques and techniques by use of stress sensitive properties. The finite element method is the conventional means of calculating residual stresses.

Finite element method is now a widely used tool in the product design and development. Its application in the manufacturing processes is a relatively new field. Industry is constantly looking towards improved productivity and quality as it is essential to optimize the design and process parameters. Welding is an important fabrication process whose study is paramount to ensure the integrity of the mechanical structures.

Aluminum alloy AA5083 important lightweight structural materials that have been widely used in Aviation, aerospace, transportation and other areas for their excellent specific strength and good Weldability. In recent years, the applications of aluminum alloy sheets and their welded structures have drawn more and more attention because of their light weight and ease of processing.

II. PROCEDURE

A. *Experimental Procedure*

The experiment was carried on a GTAW machine for welding of AA 5083 material using filler wire of ER5356. It is 3 phase, 50Hz frequency, 400A current, forced air cooling machine of size 760x313x500mm. The trolley is used to travel work piece which will move at perfect path with speed varying up to 65cm/sec. Gas flow rate in the welding can be adjusted and measure with the help of flow meter. Also, wire feed rate, welding voltage and current are adjusted in the GMAW machine and the speed of the welding can be adjusted and measured. The welded test work piece had the dimensions of 300x200x10 mm. The work piece has been edge prepared by taking the groove angle between welded pieces is 45^0.The distance between two plates maintained 3mm during welding The K type thermocouple is used for measuring the temperature distribution in the plate after the welding. The tip diameter of thermocouple is 1.5mm and wire length is 300mm. The thermocouples arrangement on the welding plate for measuring temperature distribution is as shown in **Figure1.** The tip of four thermocouple wire is kept at four different locations on a top surface in a vertical position and temperature is indicated by temperature indicator.

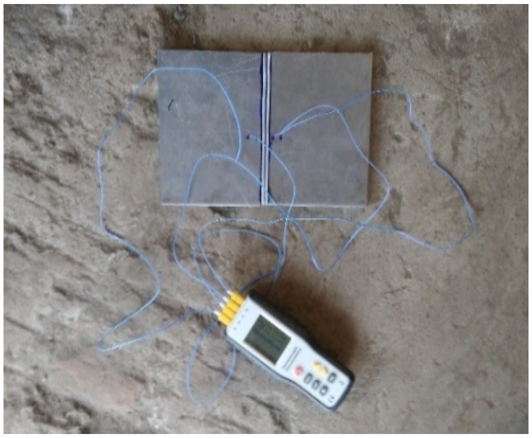
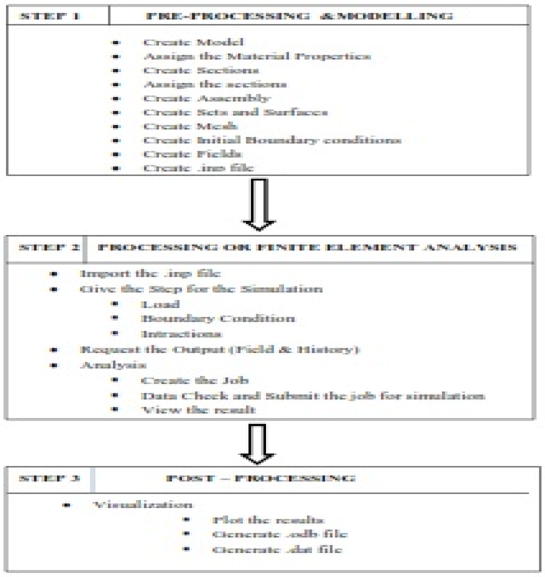


Figure 1: Experimental Setup

B. Simulation Procedure Using ABAQUS

The simulation procedure consist of the following distinct steps:

- Physical Model
- FEA Model
- Analysis
- Post processing



STEP 1 PRE-PROCESSING &MODELLING

- Create Model
- Assign the Material Properties
- Create Sections
- Assign the sections
- Create Assembly
- Create Sets and Surfaces
- Create Mesh
- Create Initial Boundary conditions
- Create Fields
- Create .inp file

STEP 2 PROCESSING OR FINITE ELEMENT ANALYSIS

- Import the .inp file
- Give the Step for the Simulation
  - Load
  - Boundary Condition
  - Intractions
- Request the Output (Field & History)
- Analysis
  - Create the Job
  - Data Check and Submit the job for simulation
  - View the result

STEP 3 POST – PROCESSING

- Visualization
  - Plot the results
  - Generate .odb file
  - Generate .dat file

Figure2:Simulation procedue

Table 1: Properties of AA5083

S.No	Parameters	Symbol	Values	Units
1	Young's Modulus	E	72	GPa
2	Poisson's Ratio	υ	0.33	-
3	Density	P	2650	Kg/m^3
4	Tensile Strength	σ_t	260	MPa
5	Specific Thermal	C	900	J/kg K
6	Thermal	K	121	W/Mk
7	Thermal	A	$25X10^{-6}$	/K
8	Melting Point	-	570	^{0}C
9	Latent Heat		$0.80x10^{-9}$	J/m^3
10	Liquids	-	638	^{0}C
11	Solids	-	591	^{0}C

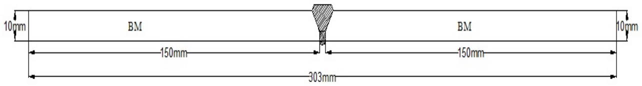


Figure3: Weld Bead Geomentry

The welded test work piece has the dimensions of 300x200x10 mm. The groove angle between welded pieces is 60^0. By using dimensions, model is prepared as shown in Figure3 A model was generated in Abaqus (A general purpose FEA software) using couple of temperature (3D solid element with temperature dof) and PLANE 55 (A 2D Solid Element with 4 nodes), as per the dimensions of the plate taken for the experimentation. A refined mesh is made based on the convergency criteria and the analysis is performed to estimate the temperature distribution. Firstly a transient thermal analysis was carried out by giving heat flux as the time varying input to estimate the temperature variation. The non-linear material properties are fed for the heat transfer solution. Then coupled field analysis is carried out to get the residual stresses and distortion by coupling thermal analysis to static analysis. The variation of the temperature with time, the residual stresses and distortions are Obtained.

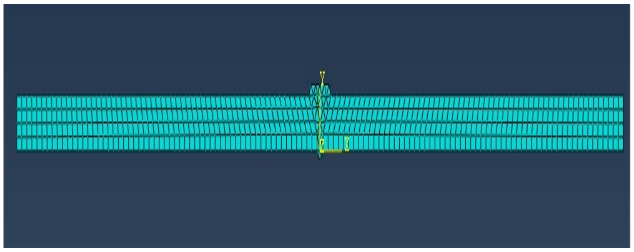


Figure4: Mesh Model

The elements in the Finite Element Mesh forming each weld pass are assigned a group name so that each pass can be deposited independently during the simulation. At the inception of the simulation, the elements of all the weld passes are made to become inactive, rendering them thermally dormant, while still keeping all the elements of the mesh attached together. This is achieved in Abaqus by using the command **"Model Change, Type = Element, Remove"** at the beginning of the first step in the input file of the thermal analysis. Each weld pass is deposited in the corresponding step in the thermal analysis by using the command "Model Change, Type = Element, Add", which reactivates the corresponding elements in the FE mesh.

The requirement of the FE model to exchange heat, at its surface, with the surrounding environment is facilitated using the option "sfilm" in the input file. A surface film or "sfilm" is generated by selecting the outside surface of revenant elements which are expected to exchange heat with the surroundings by convection and radiation or due to direct conduct with the solidus material as in the case of electric blanket. The surface film is specified and the film properties are assigned values to allow appropriate amount of heat to be exchanged at the surface.

The boundary conditions of a finite element model include thermal boundary conditions and mechanical constraint boundary conditions, in which the main consideration of the thermal boundary conditions is heat radiation, heat conduction and convection thermal cooling. The main consideration of the mechanical boundary conditions is to simulate the constraints of the fixtures on the work piece during the welding process and the constraints of removing the fixture after cooling to room temperature.

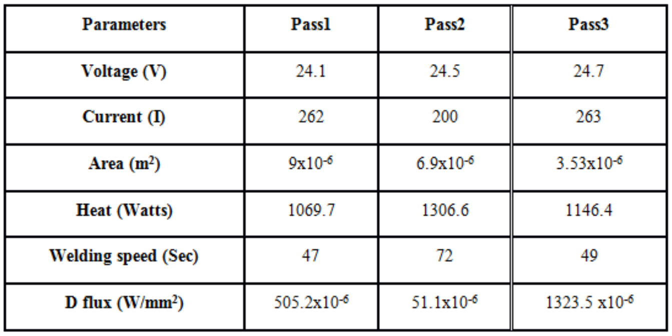
Heat Input Calculation for FEA

$$\mathbf{D_{flux} = [U.\ I.\ \eta] / A \quad (W/mm^2)}$$

Where,

U – Voltage in Volts
I – Current in Amps
Ƞ – Arc efficiency
A – Weld pass Area

Table 2: Observation Table



Parameters	Pass1	Pass2	Pass3
Voltage (V)	24.1	24.5	24.7
Current (I)	262	200	263
Area (m²)	$9x10^{-6}$	$6.9x10^{-6}$	$3.53x10^{-6}$
Heat (Watts)	1069.7	1306.6	1146.4
Welding speed (Sec)	47	72	49
D flux (W/mm²)	$505.2x10^{-6}$	$51.1x10^{-6}$	$1323.5\ x10^{-6}$

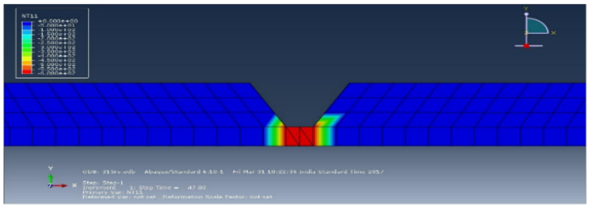


Figure5: Temperature Distribution at pass 1

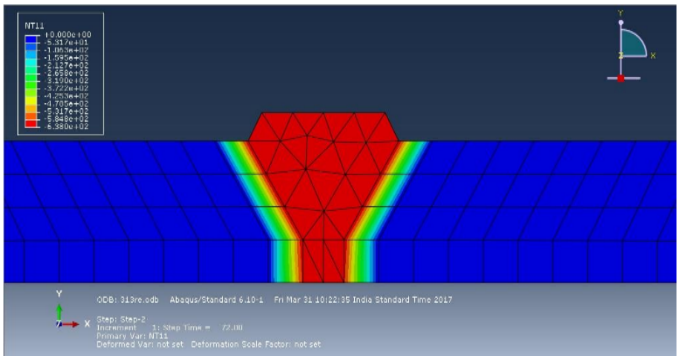


Figure6: Temperature Distribution at Pass 2

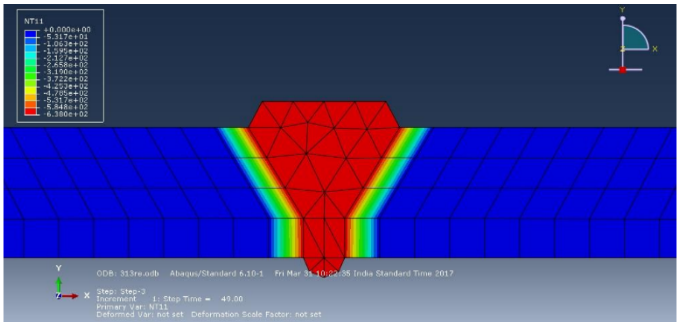


Figure7: Temperature Distribution at Pass 3

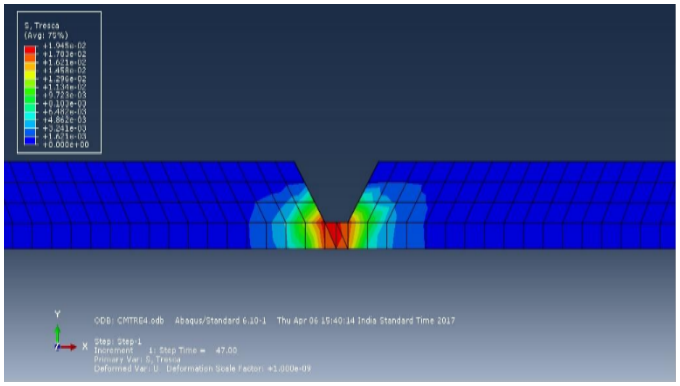


Figure8: Residual Stress Distribution at Pass 1

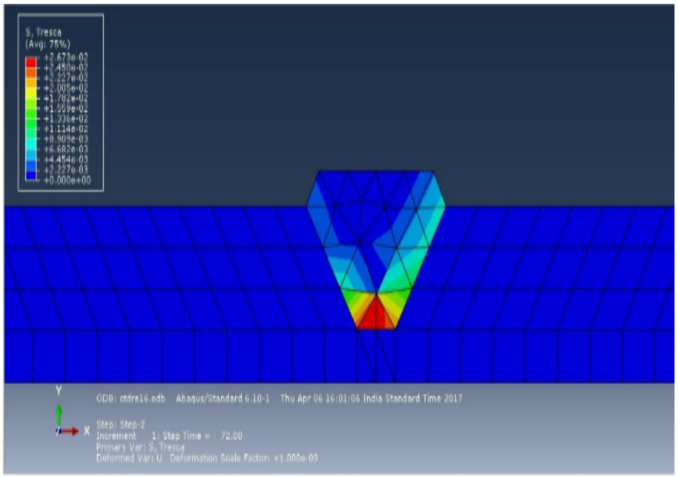
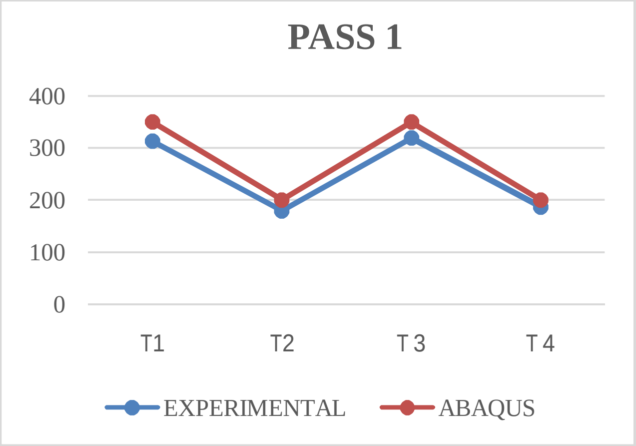
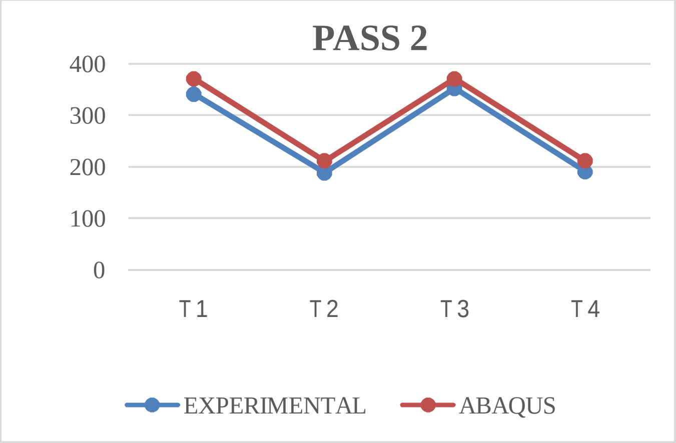


Figure8: Residual Stress Distribution at Pass 2

III. RESULTS AND DISCUSSION

TEMPERATURE DISTRIBUTION





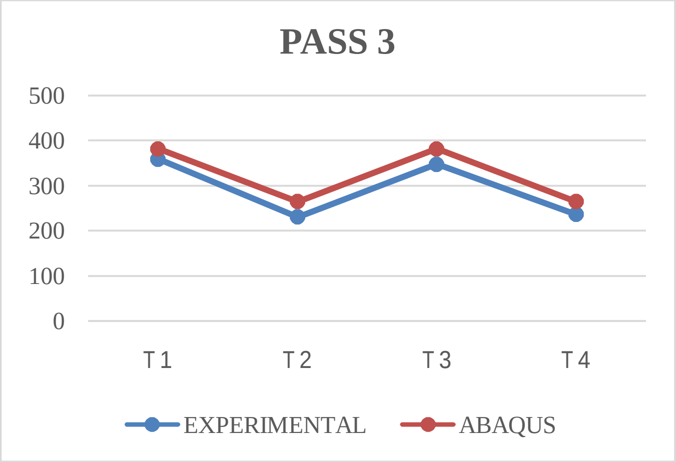


Figure 9: Comparison Between Experimental and FEA Temperature Values

The temperature of the plate was monitored during welding by attaching thermocouple wire to the surface of the plate and recording the temperatures with a chart plotter throughout the welding process. The thermocouples measured the surface temperature at four different locations with varying distance from the weld center line (WCL).

The finite element thermal results have been validated by comparing them to the temperature history measured by thermocouple at various locations after welding. The temperature history at the locations of the thermocouple predicted by the finite element thermal analysis can be made to vary significantly by altering the magnitude of the heat fluxes and the corresponding time spans over have to changed together so as to keep the fusion zones unchanged. This way, the finite element temperature history at the thermocouple locations can be changed without disturbing the fusion zone which are consistent with realist expectations. The final set of heat fluxes and time spans have been reached and their average values are reported in previous subsection.

Comparisons between the finite elements predicted temperatures and the measured temperatures by the thermocouples ae dissipated in the Figure. The **figure 9** shows measured temperature of the different pass in welding the corresponding finite element temperatures at 3 mm to the left side of the WCL. Although both points are located at an equal distance from the WCL but they are different lateral locations around the plate, they exhibit slightly different temperature history.

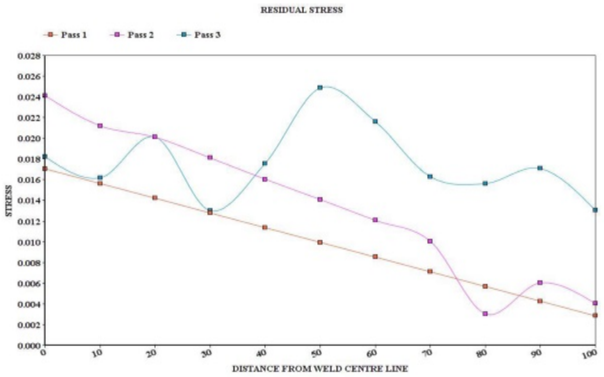


Figure 10: Residual stress graph

The residual stress was obtained by doing the Coupled Temperature displacement analysis. It was observed that the stress is at peak value in the weld bead. The stress is gradually decreased with increase of distance. The stress is low at the end of the plate. Upto HAZ the stress value at peak level. And then it reduced.

The experimental point (thermocouple position) for the thermal history were located at the weld bead on the top surface of horizontal plate. It is observed from figure that there is a close agreement between the simulation and experimental thermal profile. The FEM model predicts a little more temperature than the measured peak temperature. A small temperature gradient difference is due to effect of radiation in experiment. In this study, an experiment was conducted to verify the simulated results. As the simulation thermal profiles are nearly matching with experimental results it can be predict that stress profiles got by simulation must be correct.

VI. CONCLUSION

The Finite element analysis of AA5083 was done and it concluded that, there is a close Relationship between the simulation and experimental thermal profile. As the simulation thermal profiles were nearly matching with experimental results, it can be predict that stress profiles got by simulation must match with experimental profile. There are different experimental methods for measuring residual stresses developed in welded parts such as hole method, X- ray diffraction method etc. But experimental measurements are costly, require equipment and time consuming. However, finite element package is enough for getting results with negligible variation to that of experimental results. Simulation process can be carried out where welding applications deals with complex products.

REFERENCES

[1] Dirk Schroepfer, Arne Kromm, Thomas Kannengiesser (2016) „Engineering Approach to Assess Residual Stresses in Welded Components‟, Weld World.

[2] Dong.P, Hong.J.K, Bouchard.P.J (2004) „Analysis of Residual Stresses at Weld Repairs‟, International Journal of Pressure Vessels and Piping.

[3] Fanrong Kong, Junjie Ma, Radovan Kovacevic (2011) „Numerical And Experimental Study of Thermally Induced Residual Stress In The Hybrid Laser– GMA Welding Process‟, Journal of Material Processing and Technology

[4] G.Mi, C.Li, Z.Gao, D.Zhao (2014) „Finite Element Analysis of Welding Residual Stress of Aluminium Plates under Different Butt Joint Parameters‟, Engineering Review Vol 34.

[5] Rother.K, Fricke.W (2016) „Effective Notch Stress Approach for Welds Having Low Stress Concentration‟, International Journal of Pressure Vessels and Piping.

[6] Shirish R.Kala, Siva Prasad. N (2013) „Numerical Studies on Effect of Interpass time on Distortion and Residual Stress in Multipass Welding‟, Advanced Materials Research Vol 601.

[7] Yaghi A.H, D. W. J. Tanner, T. H. Hyde, A. A. Becker, (2012) „Finite element thermal analysis of the fusion welding of a P92 steel pipe‟, Mechanical Science Vol 03

[8] "Rate Processes in metallurgy" by Mohanty A K., PHI, 2009.