International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395 -0056
Volume: 02 Issue: 02 | May-2015 www.irjet.net p-ISSN: 2395-0072

EXPERIMENTAL AND FINITE ELEMENT ANALYSIS OF THERMALLY INDUCED RESIDUAL STRESSES FOR STAINLESS STEEL 303GRADE USING GMAW PROCESS

[1]Abhishek B P, [2]Anil kumar G ,[3]Madhusudhan.T

[1] M.Tech, student, Mechanical Engineering, SJB Institute of Technology, Bangaluru

[2]Associate Professor, Mechanical Engineering, SJB Institute of Technology, Bangaluru

[3] Professor, Mechanical Engineering, SJB Institute of Technology, Bangaluru

***

Abstract - *Gas metal arc welding (GMAW) control the metal from the wire rod by develop the arc as well as by control the input process parameter. High heating at a 1 locations during welding and further rapid cooling generate residual stress and distortions in the welding and base metal. In the last few decades, various* ***research's effort have been directed towards the control*** *of welding process parameter aiming at reducing residual stress and distortion they are strongly affected by many parameters like structural, materials and welding parameters. Such welding failures can be minimize by control the weld heat input. The distributions of the temperatures in weld joint of AISI303 grade high strength steel is investigated by* ***Finite Element Method (FEM) using ANSYS software's*** *and experimental has been performed to verify the developed thermo-mechanical finite element model using the GMAW process. Basic aim of our paper is to analyse temperatures distributions and residual stresses in dissimilar metal welded plate to avoid future failures in materials because experimental process is costlier. The behaviour of welded zone is affected by variation in temperature distributions, microstructures and mechanical properties of the materials. The residual stress gradient near the fusion zones is higher than in any other locations in the surrounding areas. Because of this stress gradient, cold crack at the fusion zones in high strength steel occur. The main objectives of this simulation is the determination of temperatures and stresses during and after the process. Temperature distribution define the heat affected zone (HAZ) where materials properties are affected. Stress calculations is necessary because high residual stresses may be caused fracture, fatigue which causes unpredictable failure in regions near the weld bead regions.*

Key Words: GMAW, FEM, Transient Thermo-mechanical simulation, residual stress,heat affected zones,fatigu failure

INTRODUCTION

ARC welded joint is one of the most important joining methods in industries. Accordingly SS and SS products are the most commonly used productes in the welding techniques.. The main aim of thermo mechanical analysis is to realize the significance of simulation of arc welding using finite element method. In machine industry and automotive industry the thin steel metals are used. The main objective of this simulation is the determination of temperatures and stresses during and after the process. Temperature distribution define the heat affected zone where material properties are affected.. The steady state temperature profile of the welded plates were solved by finite difference method. Variation of thermal and residual stresses are investigated inside a thin mild steel plate during welding processes. According to them experimental analysis is costly so they prefer FEM analysis. R. Kovacevic et al. [4] carried out numerical and experimental study of thermally induced residual stress in the hybrid laser- GMA welding process. They use both simulation and experiment process to obtain stress distribution and temperature distribution in the weld. Numerical simulation shows that higher residual stresses is distributed in the weld bead and surrounding heat affected zone. Effect of welding speed on the isotherm and residual stress of the welded joint are also studied

2.LITERATURE REVIEW

S. Muruganet all [1], studied the Temperature distribution and residual stresses due to multipass welding in type 303 stainless steel and low carbon steel weld pads. In a multipass welding operations, the residual stresse are they developed. This changes causes stresses with every welding pass. Among various welding operations they carried out MMAW i.e., Manual Metal Arc Welding. This gives tensile residual stresses increases susceptibility of weld to fatigue damage, stress corrosion cracking as well as fracture.

M. Jeyakumar et al [2] did the evaluation of residual stress in butt-welded steel plates. The causes residual stresses and distortions are dominated by deformation of the metals in the HAZ of weld joints as well as by external and internal restraints causes in the material. The residual stress effects may be either beneficial or detrimental which depends upon the magnitude and distribution of stresses..

K Punitharani, et al [3] discussed the FEM for the residual stresses and distortion in the hard faced gate valves in the processes. The process of depositing a filler material on the surface of carbon and low alloy steel base metal is called the hard facing. In this 1131 work residual stresses are predicted in hard face gate valve using FEA and with the help of X-ray diffraction technique stresses measured are being validated. Here the load steps fairly are very high, therefore programming language called ANSYS parametric design language is used and the coding was employed to perform both thermal and structural analysis.

PROCEDURE

A. Experimental process description

The experiment was carried on a semi-automatic GMAW machine for welding of AISI 202 material using filler wire of AISI 308L. It is 3 phase, 50Hz frequency, 300A 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 designed of 250x100x10 mm. The groove angle between welded piece is 600. 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 fig. (1). The tip point of 3 thermocouples is kept at three different locations on a top surface in a vertical position and temperature is indicated by temperatures indicators as shown in fig.1

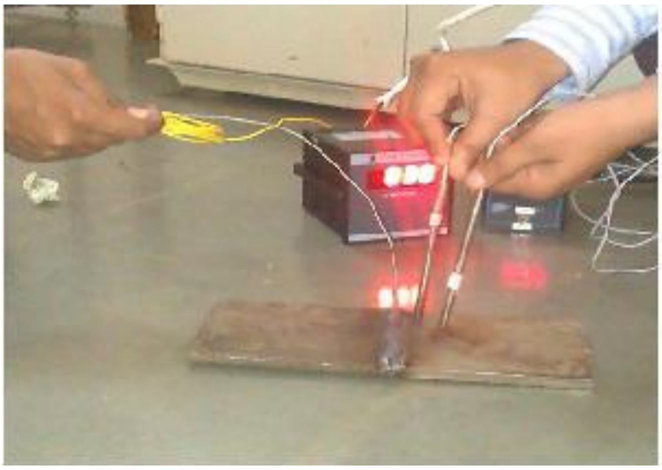


Figure 1.Experimental set up

B. Simulation procedure using ANSYS

It consisting of 3 individual steps:

- build the model,
- apply loads and obtain the solution

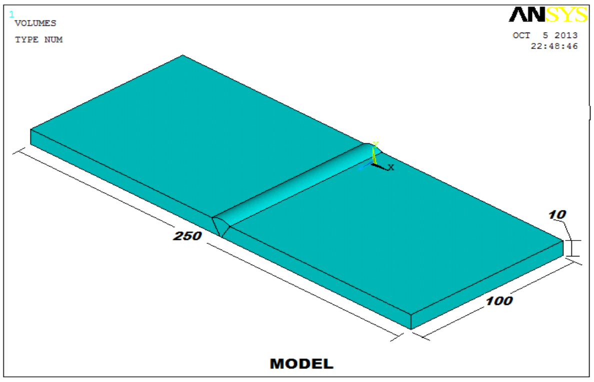


Figure 2.Flowchart of simulation procedure

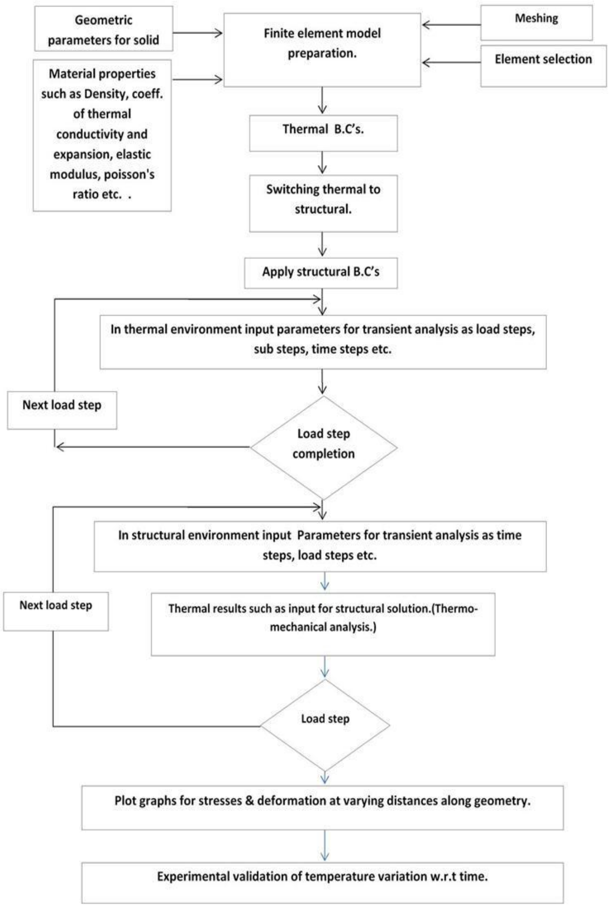


Table 1:Thermal material properties

Temperatures (^{0}C)	Thermal conductivity (W/m-^{0}C)		Thermal expansion coefficient (x10^{-6}/^{0}C)	
	AISI 202	AISI 308L	AISI 202	AISI 308L
100	16.2	16.1	17.5	17.2
200	17.5	17.3	17.8	17.6
300	19	18.7	18.4	17.8
400	20.8	20.1	18.8	18.1
500	21.6	21.6	19.2	18.4
600	21.9	23.2	19.6	18.8
700	22.2	24.6	19.9	19.1
800	22.6	26.3	20.2	19.6
900	23.2	28.2	20.4	20
1000	23.6	29.5	20.6	20.3
1100	23.9	31.8	20.8	20.6

The welded test work piece has the dimensions of 250x100x10 mm. The groove angle between welded pieces is 600. By using dimensions, model is prepared as shown in fig. 3(a).

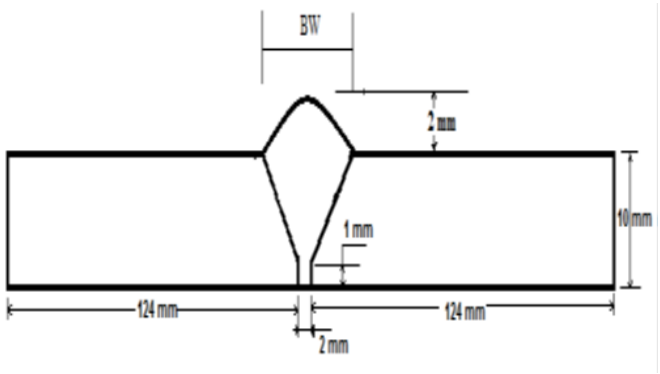


Figure 3.Model developed

The model with the FE mesh has been shown in Fig.4. The eight- node brick elements with linear shape functions are used in meshing the model. The SOLID70 and SOLID185 elements have been used for meshing. SOLID70 has a three-dimensional thermal conduction capability. The element has eight nodes with a single degree of freedom, temperature, at each node. The element is applicable to a three-dimensional, steady-state or transient thermal analysis. The element also can compensate for mass transport heat flow from a constantvelocity field. If the model containing the conducting solid element is also to be analyzed structurally, the element should be replaced by an equivalent structural element (such as SOLID185). Total number of elements after meshing are 5992.

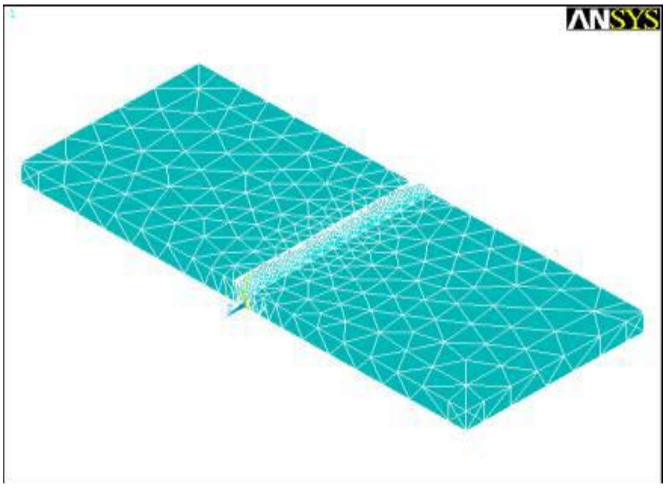
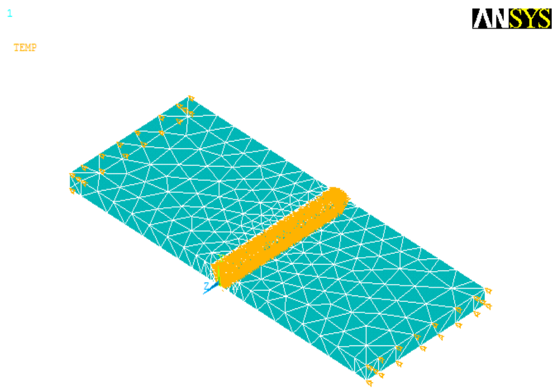


Figure 4.Meshed model

temperature of 13850C (melting point temperature of filler material AISI 308L) is given at the welded joint that is between the plates along the centre line. Bulk temperature of 280C is given at the end of the plates. The simulation has been carried out for thermal analysis in a time period of 1000 seconds. The number of sub steps are 5. The time at the end of load step is 1000 second. The time step size is 200 seconds. The model with thermal boundary conditions has been shown in fig.(5)



Clamping has made at the ending surface to make avoiding distortions while welding of plates. As effect of clamping at both end surfaces, vertical displacement of plates due to welding force is constrained. The model with the structural boundary conditions has been shown in fig.(6)

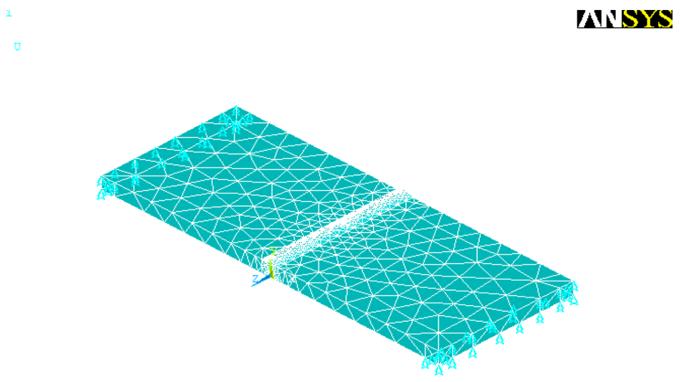
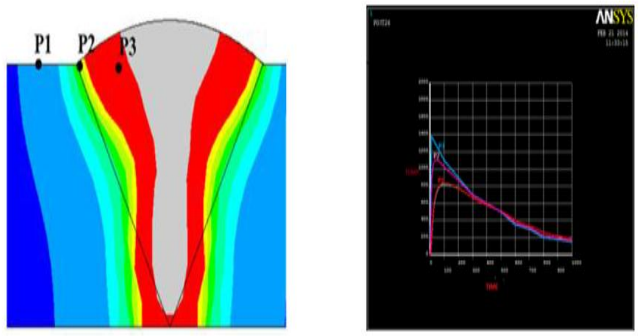


Figure 6.Structural boundary conditions

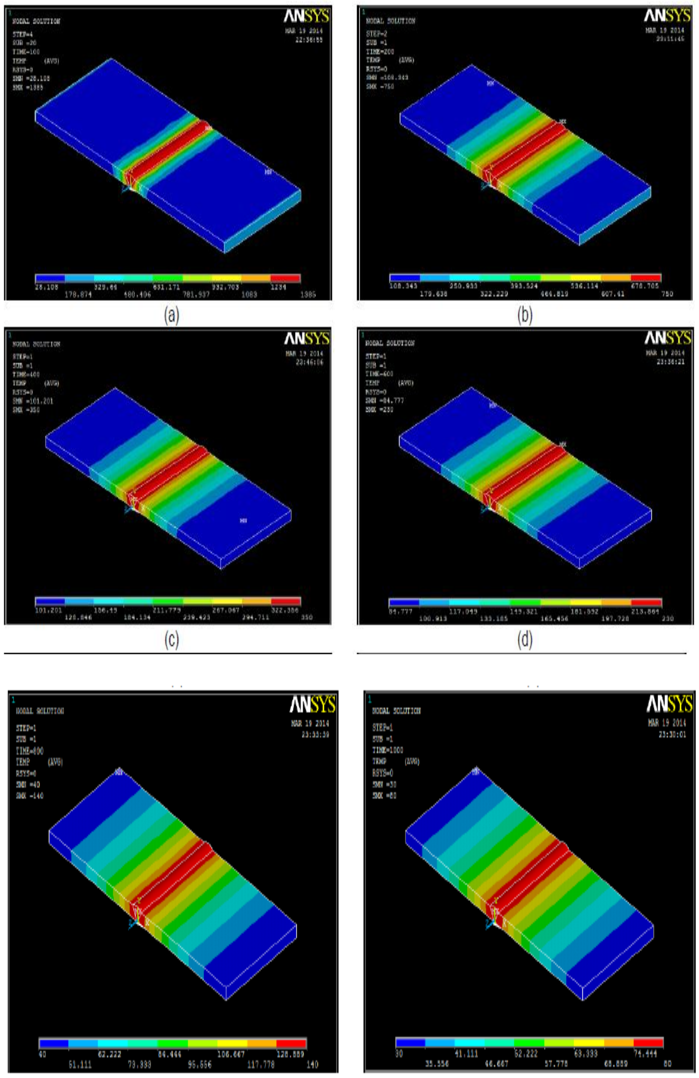
RESULTS AND DISCUSSION

As shown in Fig. 7(a) cross sectional view of weld bead with 3 points at 3 main different locations (at weld zone, heat affected zone and base metal) at the top surfaces of the weld element and the distance from the centre line. That temperature distribution curve is plotted at these three locations with respect to time as shown in the fig. 7(b). It is observed that high temperature exist at the weld zone where the melting occurs by arc heating. When the welding process is done, welded specimen is cooled up to the room temperature (280C) in air. Temperature value decreases when further moving away from the centre line of the weldment. Fig.7 (b) shows that they are maximum temperature reaches to 14400C at weld zone (at point P3). Similarly at point (P1 and P2) on parent plate maximum temperature values are decreases. Figure 10(a-d). shows the stress distribution along weld bead at three different paths at the top surface of weld bead at z=20, 50 and 80 mm. Figure 10(a) shows that maximum compressive stresses are formed near the weld bead. The intensity of stresses are more at the middle of weldment and nearly same at 20mm and 80mm. The same amount of tensile stresses are induced near end surfaces due to clamping. At z=50mm, compressive residual stresses of about 300 MPa near the weld bead are developed. Fig. 10(b) shows that compressive normal stresses are developed at end surfaces due to clamping. Similarly compressive stresses are more atz=50mm, compressive residual stresses of about 300 MPa near the weld bead are developed. Fig. 10(b) shows that compressive normal stresses are developed at end surfaces due to clamping. Similarly compressive stresses are more at weld bead and tensile stresses are developed on heat affected zone. Residual stresses of range 50-100 MPa are developed at the weld bead. Fig.10(c) shows that more compressive stresses are induced near the weld bead. Compressive stresses upto 470 MPa are developed at the weld zone. Similarly tensile stresses developed in the parent material zone are about 150 MPa. The little amount of compressive stresses are induced near the end surface of the weldment. The tensile and compressive stresses are maximum they at middle of the weldment. Fig. 10(d) shows that all the stresses are tensile in nature and it is maximum at weld bead and minimum at the end of the weldment. Von mises stresses developed at the weld bead are upto 380 MPa. Figure 11(a-d) shows the stress distribution across the weld bead at the middle section (z = 50mm) along thickness. Fig.11(a) shows that residual compressive stresses are maximum near weld bead zone and tensile stresses are observed in base material near the end surfaces. The end portion of the weldment is free from stresses. The residual compressive stress of nearly 300 MPa are developed in the weld zone. Fig. 11(b) shows that compressive stresses are developed at weld bead. Residual compressive stresses of 50-150 MPa are developed at the weld zone. Also little compressive stresses are developed near end surfaces.Stresses induced at all locations are almost same. Fig. 11(c) shows that longitudinal stresses at middle, top and bottom surface are same. More compressive stresses are developed near the weld bead upto 500 MPa. Also tensile stresses induced in parent material. Figure 11(d) shows that all the stresses are tensile in nature and it is maximum at weldment and minimum at the end of the weldment. The tensile stresses are goes on decreasing as move away from weld bead zone. Von mises stress developed in entire weldment is nearly same. From fig 10(a-d) and 11(a-d), it is observed that at weld zone compressive stresses are developed which are more than yield strength of material. Hence failure may occur at the weld zone if maximum loading is done at this zone



(a) (b)

Figure 7. Temperature distribution at various locations (a) cross sectional view of weld bead, (b) temperature distribution curve

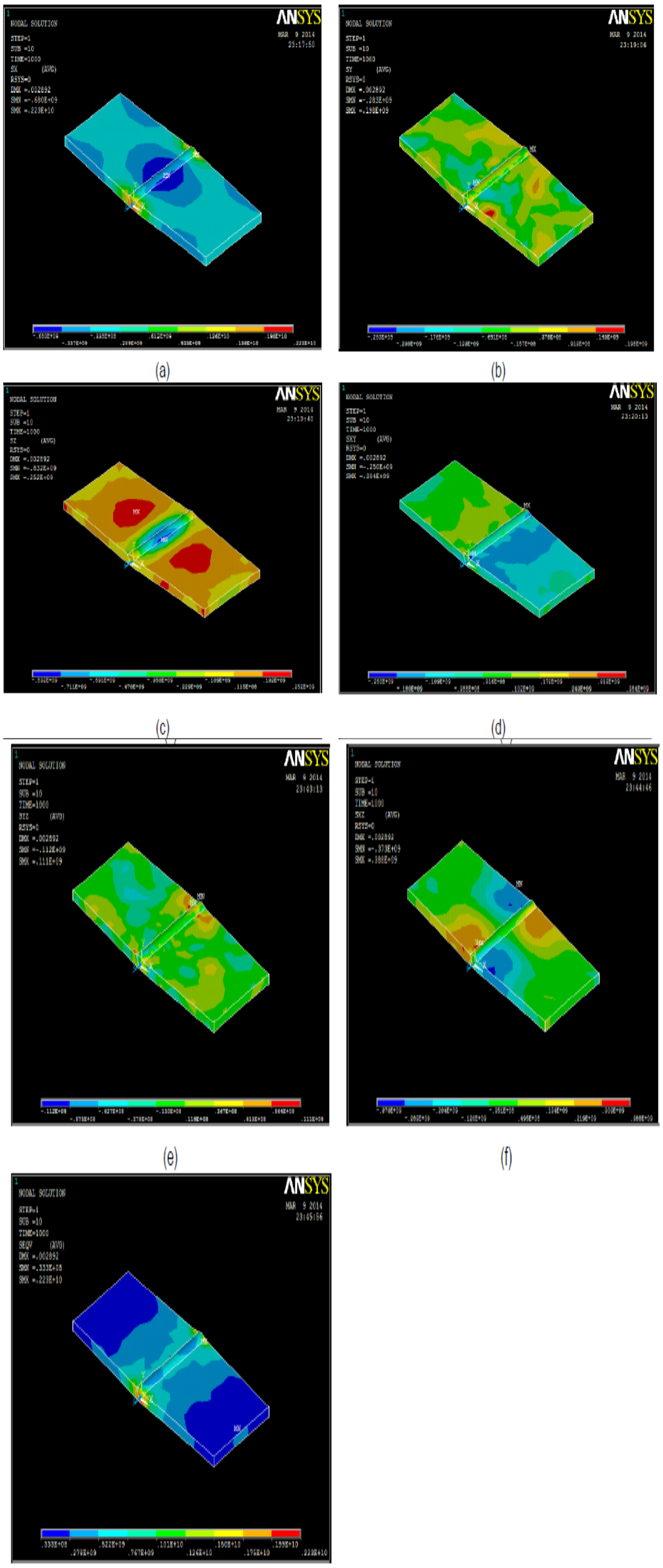


(a) (b)

(c) (d)

(e) (f)

Figure 8. Temperature distributions along plate due to conduction at different times a) at t=20 sec, b) at t=200 sec, c) at t= 400 sec, d) at t= 600sec, e)at t=800 sec, f)at t=1000 sec.



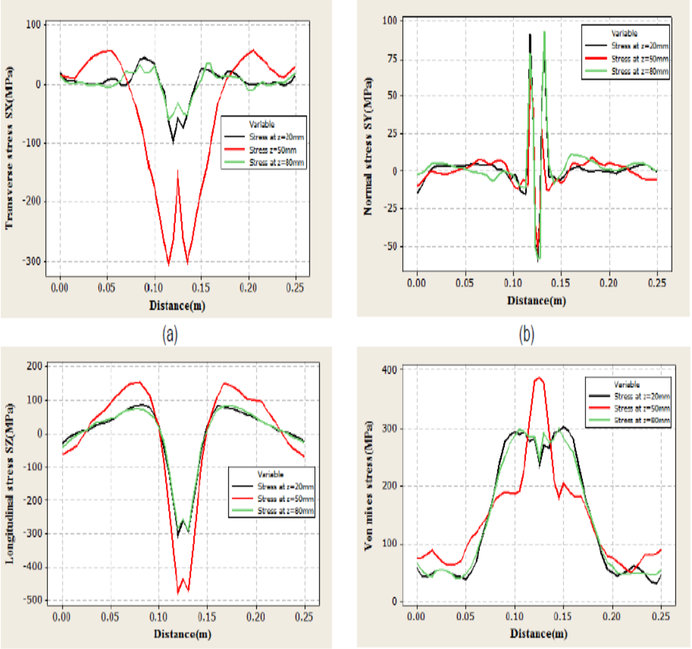
(a) (b)

(c) (d)

(e) (f)

(g)

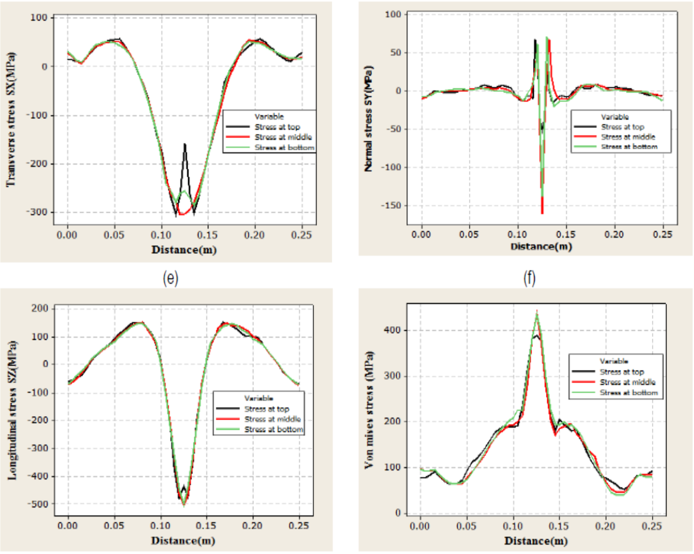
Figure 9. Residual stress distribution: (a) transverse stress SX, (b) normal stress SY, (c) longitudinal stress SZ, (d) x–y plane shear stress SXY, (e) y–z plane shear stress SYZ, and (f) z–x plane shear stress SZX, (g) Von mises stress SEQV.



(a) (b)

(c) (d)

Figure 10.Stress distribution along weld bead at three different paths at the top surface of weld bead obtained by GMA welding: (a) transverse stress SX, (b) normal stress SY, (c) longitudinal stress SZ, and (d) Von Mises stress SEQV



(e) (f)

(g) (h)

Figure 11. Stress distribution across the weld bead at the middle section (z = 50mm) along thickness: (a) transverse stress SX, (b) normal stress SY, (c) longitudinal stress SZ, and (d) Von mises stress SEQV.

COMPARISON OF SIMULATION AND EXPERIMENT RESULTS

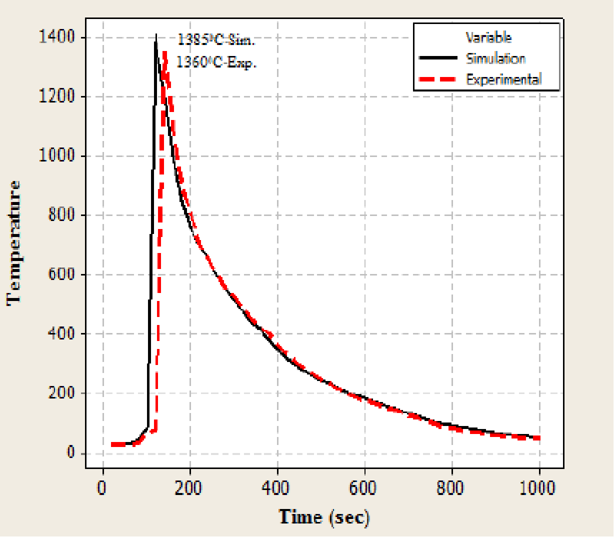


Figure 12. Comparison of simulation and experimental thermal profiles.

Fig. 12 shows the analysis & experimental temperature distribution profiles in graph. The experimental point position (thermocouple position) for the thermal history were they 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 analysis 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

CONCLUSION

1) There is a close agreement between the simulation and experimental thermal profile. As the simulation thermal profiles are nearly matching with experimental results, it can be predict that stress profiles got by simulation must match with experimental profile.

2) 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

REFERENCES

[1] M Sundar, G Nandi, "Assessment of residual stress and distortion in welding by finite element method", International conference on mechanical engineering 2005, Dec 2005,Vol. 1, pp. 1-6.

[2] N.Akkus, E. Toptas, "Thermo-mechanical analysis of arc welded joint by finite element method", AWST, Oct. 2011,Vol. 20, pp. 24-25.

[3] P. Biswas, "Numerical and experimental study on prediction of thermal history and residual deformation of double-sided fillet welding", J. Engineering Manufacturing, 2009, Vol. 224, pp. 125-134.

[4] Fanrong Kong, Junjie Ma, Radovan Kovacevic, "Numerical and experimental study of thermally induced residual stress in the hybrid laser–GMA welding process", Journal of material processing technology, USA, Jan 2011,Vol. 123, pp.1102-1111.

[5] Li Yajiang, Wang Juan, "Finite element analysis of residual stress in the welded zone of a high strength steel", Indian academy of sciences, April 2004, vol. 27 no. 2, pp. 127-132.

[6] S. Naser, "Transient Variations Of Thermal Stresses And The Resulting Residual Stresses Within A Thin Plate During Welding Processes", Journal of thermal Stresses, Taylor And Francis Group, 2004, Vol. 27, pp. 671-689.

[7] H. Lee, "Numerical And Experimental Investigation Into Effect Of Temperature Field On Sensitization Of AISI 304 in Butt Welds Fabricated by Gas Tungsten Arc Welding", Materials Transactions, 2011, Vol. 52, No. 7, pp. 1506-1514.

[8] Dr. Hani Aziz Ameen, "Influence of the butt joint design of TIG welding on the thermal stresses",eng.& Tech. Journal, 2011,Vol. 400,pp.400-420.

[9] H. Yu, "Thermal Analysis Of Welding On Aluminium Plates", Journal Of Marine Science And Technology,2003, Vol. 11, No. 4, pp. 213-220.

[10] A. Anca, "3D-Thermal-Mechanical Simulation Of Welding Processes", MecanicaComputacional, Vol. 23, pp.2301-2318.

[11] Rahib Kamal Kassab, "Experimental and finite element analysis of a T-Joint welding", Journal of mechanical engineering and automation, 2012, Vol. 400, pp. 411-421.

[12] M. Abid, "3D thermal finite element analysis of single pass girth welded low carbon steel pipe-flange joints", Turkish J. Eng. Env. Sci., 2009, Vol.33, pp.- 281-293.

[13] Dragistramenkovic, "Finite element of residual stress in butt welding two similar plates", scientific technical review, Serbia, 2009, Vol. 23, pp.342-360.

[14] M.Kaldhar, K.Venkata, "Optimization of process parameters in turning of AISI 202 austenitic stainless steel",ARPN journal of engineering and applied science, sept2010, Vol. (5) No. (9), pp.440-450.

[15] Carpinteri, "Fully Three-Dimensional Thermo-Mechanical Analysis Of Steel Welding Processes", ELSEVIER Journal Of Materials Processing Technology, 1995, Vol. 53, pp. 85-92.

[16] X. Zhu, Y. Chao, "Effects of temperature-dependent material properties on welding simulation" Department of Mechanical Engineering, University of South Carolina, Columbia, SC 29208, USA, 11 January 2002, pp. 967–976

BIOGRAPHIES

Abhishek b p is a M.Tech student in Department of mechanical engineering from SJBIT,Bangalore His interests are in the field of design and analysis.

Anil Kumar G is a faculty serving as Associate Professor Department of Mechanical Engineering from SJBIT, Bangalore. His interests area are Composites, design&analysis, Mechanics of machines

Prof. Madhusudhan T is a faculty Serving as head of the department of SJBIT, has a vast experience in Engineering academics. His special Interest areas are automobiles System, dynamics of machines.