SIMULATION OF TEMPERATURE AND RESIDUAL STRESS FIELD IN FRICTION STIR WELDED AISI 304 STAINLESS STEEL JOINTS

[1]S Sree Sabari, [2]M Chenchu Giri and [3]V Balasubramanian

[1]Associate Professor, [2] UG Scholar, [3]Professor
[1,2]Dept of Mech Engg., Sree Vidyanikethan Engineering College, Tirupati, AP
[3]Centre for Materials Joining Research (CEMAJOR), Dept of Manufacturing Engg., Annamalai University, Annamalai Nagar, TN
[1]E-mail: sreesabaridec2006@yahoo.co.in

Abstract: *Three-dimensional nonlinear thermal and thermo-mechanical numerical simulations are conducted for the friction stir welding of AISI 304 stainless steel. The finite element analysis code SYSWELD was used to simulate the results using inverse approach. Defect free welds were made experimentally using a rotational speed, welding speed and shoulder diameter of 800 rpm, 90 mm/min and 20 mm respectively. Residual stress measurement was carried out with X-ray stress analyzer employing* C_rK_α *radiation. The transient temperature fields were obtained by finite element simulation and the residual stresses in the welded plate are calculated using a three-dimensional elastic–plastic thermo-mechanical simulation. The results of the simulation are in good agreement with that of experimental results.*

Key words: *Stainless Steel, Friction Stir Welding, Finite Element Analysis, Temperature Distribution, Residual Stress*

1. INTRODUCTION

Austenitic stainless steels are widely used in many industries utilizing high temperature components such as heat exchangers and chemical reactors. In addition, it is also applied in aero engine, boilers, etc., because of their good mechanical properties at elevated temperatures and excellent corrosion resistance. While these alloys have useful properties in the wrought condition, welding is known to deteriote its properties by following three ways. Three main problems encountered in the welding of austenitic stainless steel stand out. They are sensitive structure developing after the formation of chromium carbide on the surface, the formation of hot fracture, and the formation of sigma phase risks encountered at high working temperatures [1]. Friction stir welding (FSW), which is a relatively new solid state joining process and has been the focus of constant attention in joining low and high temperature melting point materials that holds promise as an effective method for improving mechanical properties and suppressing sensitization phenomena in HAZ. To date, most of the published information is on the microstructure and mechanical properties of friction stir welded austenitic stainless steel joints[2]. The metallurgical changes during friction stir welding can be correlated with the peak temperature attained by different zones of welded joints.

However, the physical process involved in FSW is complex, which involves non-linear multi-physical phenomena. It is hard for any experimental technique to predict the complete mapping of residual stress and temperature field evolved during FSW [3]. In order to make it ease, FEM was used to predict the temperature field during FSW. Zhu et al [4] studied the temperature distribution and residual stress field of friction stir welded 304L austenitic stainless steel joints using WELDSIM finite element code and reported that the maximum temperature obtained from the simulation was in the range of 900 – 1000°C. Prasanna et al analyzed the temperature distribution during FSW of 304L austenitic stainless steel using ANSYS package and they reported that the maximum temperature was around 1054°C [5]. J. H. Hattel et al simulate the three dimensional model of FSW by considering the mechanical effect of the tool to predict the temperature and stress evolution by only accounting the heat generation by shoulder [6]. The fixturing effect and the force history involved in FSW were also analyzed in order to control the residual stress and to know the exact mechanism of FSW. Yihua et al investigated the heat generation at the tool/workpiece interface and the heat conduction to both tool and workpiece [7]. The heat source model represent the heat generation due to shear or friction force generated by the tool shoulder, but neglecting the heat generated by the probe surface where the pressure at the interface is assumed to be constant. Sanjeev et al carried out a numerical analysis by continuously adjusting the coefficient of friction, so that the temperature should not reach the melting temperature [8]. Various contact conditions at the interface between tool and workpiece surface were carried out for understanding the material flow and heat generation behavior [9]. However, studies to predict the temperature distribution during FSW of 304 SS and its effect on residual stress field in the welded joints is not yet fully reported.

In this paper, the temperature distribution and residual stress field during friction stir welding of 304 stainless steel is simulated using the finite element code SYSWELD. The temperature at specific locations was measured experimentally by the use of K-type thermocouples. The real time welding is interfaced with the computer by the use of LABVIEW data acquisition system. The temperature predicted numerically is adjusted with respect to the experimental values by the use of inverse method. Residual stresses in the welded plates were measured by the X-ray diffraction (XRD) technique. Temperature field obtained from numerical simulation was used to calculate the residual stress pattern and it is compared with the experimentally measured residual stresses.

2. EXPERIMENTAL PROCEDURE

In the friction stir welding, the workpiece is placed on the worktable and it is rigidly clamped. The FSW tool composed of flat shoulder and taperedpin. The tool shoulder is of 15 mm diameter and the pin diameter is one third of the shoulder diameter. Initially the tool rotational speed of 1000 rpm is plunged slowly into the workpiece until the shoulder surface and workpiece surface get contact to generate heat. After the dwell period, the tool traverse along the weld line was given at 1.67 mm/sec to achieve a complete coalescence. In this work, the weld piece geometry comprises of 75 mm × 150 mm × 3 mm plates (Fig. 1). K type thermocouples made of chromel and alumel, with a sensitivity of 41 μV/°C were used for temperature prediction. The probe capable of measuring up to 1600 °C range was used. Two thermocouples are embedded at the mid length of the plates with one located at the weld center and the other at 10 mm away from the weld center. The data acquisition system interprets the real case data and recorded in the computer by the use of Lab view Software. X-ray diffraction method was employed to measure the residual stress. The procedure used for measuring and calculating the residual stress was explained in references [10, 11].

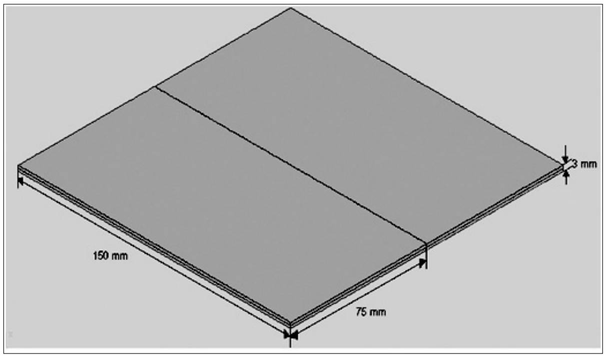


Fig 1. Geometry Configuration

3. FINITE ELEMENT ANALYSIS

This paper regards three-dimensional nonlinear thermal and thermo-mechanical analyses using the finite element welding simulation code SYSWELD. The eight node topology considering three degrees of freedom was used. The uniform meshed model of 18479 elements used for finite element analysis is depicted in Fig 2. The

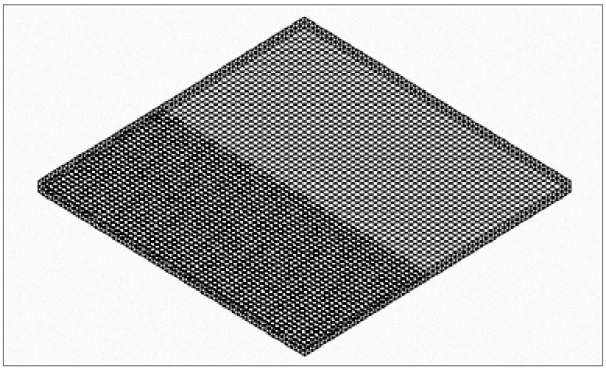


Fig. 2. Meshed Model of the Specimen

thermal analysis was carried out to predict the temperature profile and the thermal data obtained was used as the input for successive thermo-mechanical analysis to predict the stress values [12].

3.1 Thermal Boundary Condition

The governing differential equation for three dimensional heat conduction equations for a solid in Cartesian coordinate system is given by [13]

$$\frac{\delta^2 T}{\delta x^2} + \frac{\delta^2 T}{\delta y^2} + \frac{\delta^2 T}{\delta z^2} + \frac{q_g}{k} = \frac{1}{\alpha}\left(\frac{\delta T}{\delta t}\right) \quad (1)$$

Where, $\alpha = \frac{k}{\rho C_p}$ thermal diffusivity of the material, q_g is the heat generation per unit volume in W/m^3, ρ is density of the material in kg/m^3; C_p is specific heat in J/kgK and k_x, k_y, k_z are thermal conductivity W/mK.

At the solid boundary, by Newton's law of heating and cooling, convective heat is given

$$\text{as} -k\frac{dT}{dx} = h_c(T_s - T_\infty) \quad (2)$$

Where h_c includes the convective and radioactive heat transfer and calculated by empirical relationship, at all free surfaces.

$$h_c = 2.41 \times 10^{-3} \varepsilon T^{1.61} \quad (3)$$

Where ε is emissivity, T is temperature (K), T_s is surface temperature (K), and T_∞ is ambient temperature (K).

A temperature based material properties are considered for the finite element analysis [14]. Thermal boundary conditions are symmetrical across the weld centerline. Heat transfer from the workpiece to the clamp is negligible, thus heat conducted to clamp is not accounted in this investigation.

3.2 Mechanical Boundary Conditions

The specified mechanical boundary conditions are those just sufficient to prevent rigid body motion of the model. Zero displacement conditions were used for constraining the butt joint which resembling the complete fixed fixturing. The mid-plane of the butt joint is assumed to be a plane of symmetry in the analysis, which is parallel to the y-z plane

3.3 Thermal Process Modeling of FSW

The heat generated is concentrated locally and propagates rapidly into subsequent regions of the plates by heat conduction according to (Eq. 1) as well as convection and radiation through the boundary. To consider convection and radiation on all workpiece surfaces except for the bottom, the heat loss q_s is calculated by equation [4].

$$q_s = \beta T - T_0 + \varepsilon\sigma(T^4 - T_0{}^4) \quad (4)$$

Where T is absolute temperature of the workpiece, T_0 is the ambient temperature, β is the convection coefficient, ε is the emissivity of the plate surfaces, and σ= 5.67 x 10^{-12} Wcm^{2}°C is the Stefan-Boltzmann constant.

It is assumed that the heat flux, *q*(*r*), is linearly distributed in the radial direction of the pin tool shoulder, and has the following form

$$q(r) = 12\,Qr/\pi\left(d_o{}^3 - d_i{}^3\right) \text{for } d_i/2 \le d_o/2 \quad (5)$$

Where d_0 is the outside diameter of the pin tool shoulder, d_i is the pin diameter, and *Q* is the total heat input energy. Chao et al. developed a model representing the heat flux distribution as a function of heat generated between the shoulder and material surface by neglecting the heat generated from the tool probe.

$$q(r) = 3Qr/2\pi r_0{}^3 \text{for } r \le r_0 \quad (6)$$

Three-dimensional heat transfer model for the workpiece assumes the tool shoulder exist a constant heat flux. By trial and error method, heat input will be adjusted until all the temperatures values calculated numerically meet up with the values calculated by experiment using thermocouples. This analyzing procedure is called as inverse analysis method, which is dealt elsewhere.

3.4 Thermal Source Modeling of FSW

The moving heat source is represented as moving the heat generation in the nodes in each computational time step. Dong et al employed heat source model called double ellipsoid heat source model which use heat flux as the thermal load with ramping function [15]. This heat source model used the double ellipsoid method of heat deposition such that the size and shape of the energy source can be easily altered in order to replicate the FSW process (Fig. 3). Q_f is the energy input to the front quadrant of the ellipsoid in Watts per cubical millimetre, Q_r is the energy input to the front quadrant of the ellipsoid inW/m^3, a_f, a_r, b and c are the Gaussian parameters.

4. RESULTS AND DISCUSSION

During friction stir welding, the tool traverses along the joint line at a constant welding speed to create joint as shown in Figs. 4a-4d. Using the inverse analysis method welding simulations were performed for the FSW of 304 stainless steel joint and the final numerical values were accounted for comparisons. Fig. 5 shows the contour plot of temperature distribution at the centre of the weld length showing peak temperature of 1246˚C. Tang et al identified that the maximum temperature developed during FSW process ranges from 80% to 90% of the melting temperature of the base metal used. The melting point of 304 stainless steel is 1450˚C [16]. The peak temperature calculated is 86% of the melting point of alloy chosen for the study. The variations of temperature from FEA and from experimental measurement against time history at the middle length of workpiece are shown in Fig.6. At the weld centre, the temperature measurement by

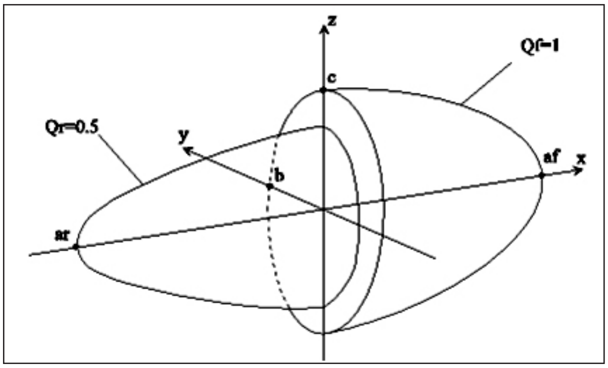
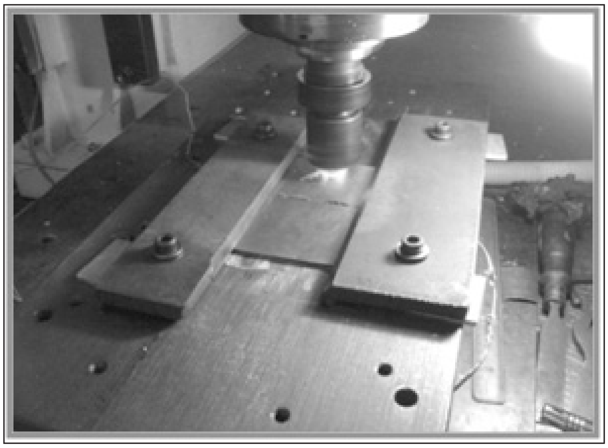
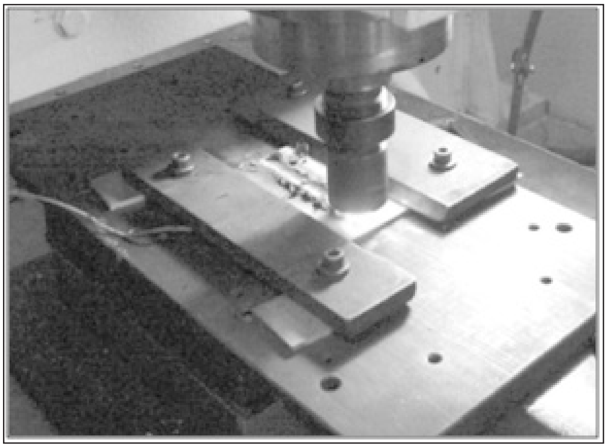


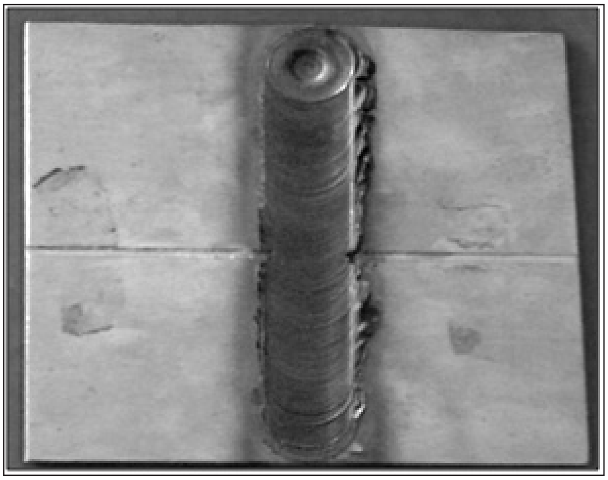
Fig 3. 3D Double Ellipsoid Heat Source Model



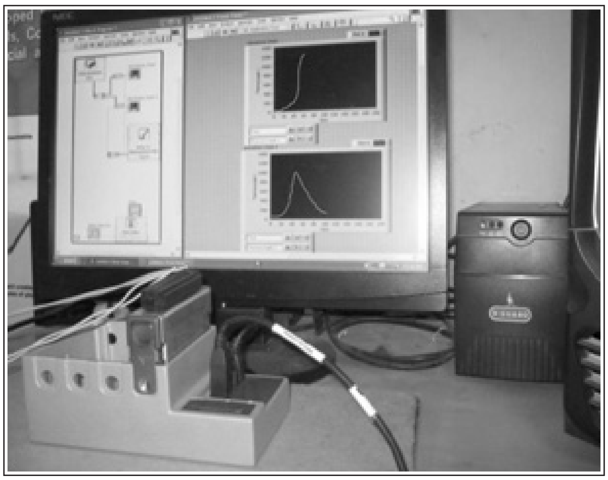
a. Photograph taken during plunge stage



b. Photograph taken during traverse



c. Fabricated 304 FSW joint



d. Data acquisition system

Fig 4. Fabrication of FSW Joints

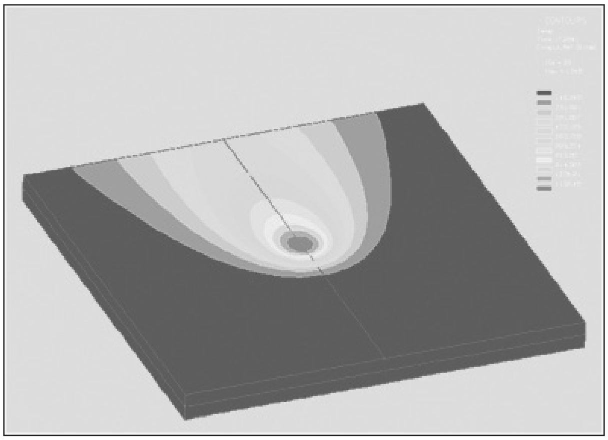
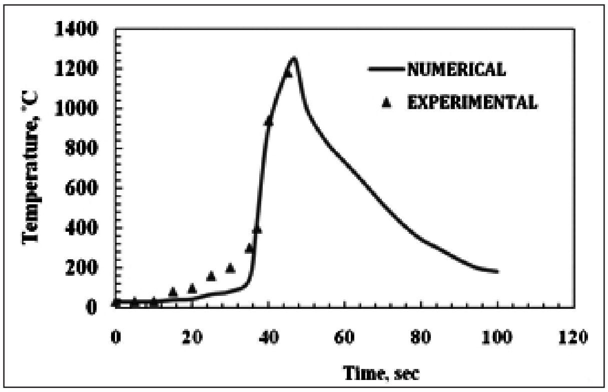
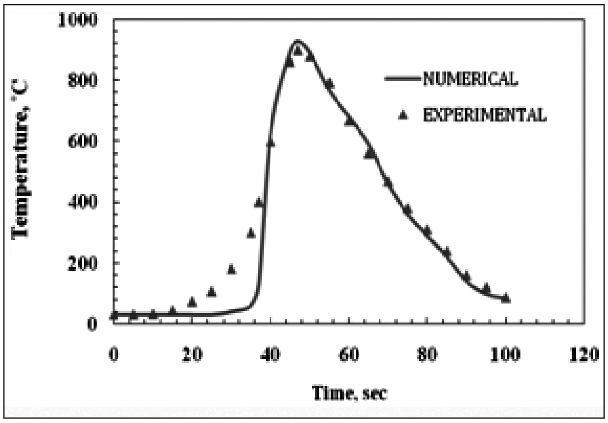


Fig 5. Temperature Contour Plots



a) At the weld center



b) 10 mm from the weld center

Fig 6. Temperature Measurements using Thermocouples

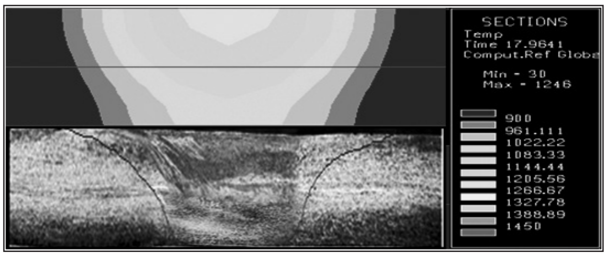


Fig 7. Comparision of Weld Bead Profile

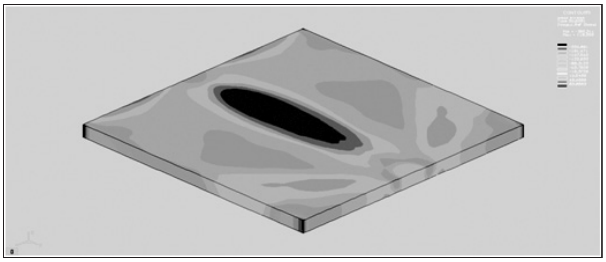


Fig 8. Residual Stress Contour Plots

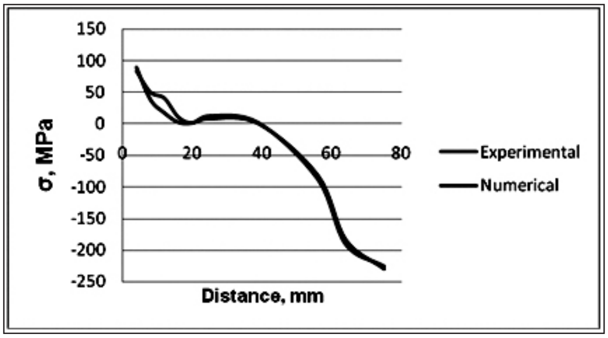


Fig 9. Numerical and Experimental Comparison of Residual Stress

thermocouples cannot be achieved over the complete cycle of welding owing to workpiece-tool interaction. So the rise of temperature until the tool reaches the thermocouple was predicted. The predicted temperature shows good agreement with the predicted value and the cooling curve can be obtained from the numerical curve is shown in the Fig. 6a. The temperature measured 10mm away from the weld line are near equal to the predicted temperature values on both advancing and retreating side. Fig. 6b shows a peak temperature value of 944˚C numerically and 926˚C experimentally. The macrostructure of weld zone and heat affected zone were correlated with the numerical results which shows good agreement depicted in the Fig. 7. By observations the peak temperature does not exceed the melting temperature and the heat affected zone temperature 944˚C shows good agreement with the experimental values.

It should be noted that the stress in the plates is mostly from the thermal effect created due to friction rather than from the mechanical action of the tool. Consequently symmetrical stress results in the plates on both the advancing side and the retreating side during the FSW. Residual stress distribution presented in Fig. 8 indicates that high stresses are developing close to the weld line, with the maximum values of 89 MPa. Fig. 9 depicts the numerical and experimental comparison of residual stress which has a good agreement. Due to the larger temperature gradient, thermal expansion coefficient and temperature distribution through the material affect the distribution of residual stress. Larger residual stress was observed parallel to the weld direction and near the weld zone. The tensile stress form in the region around the tool region was due to the thermal expansion effect. The region away from the weld line was affected by the compressive stress. However, it should be noted that much lower values were observed when compared to the residual stress values reported in friction stir welded austenitic stainless steel joints.

The restraints imposed by the FSW clamps prevent the contraction of the stir zone in the longitudinal and traverse direction due to cooling. This in turn will introduce traverse/longitudinal residual stress developed during FSW can have a significant effect on the service performance of the welded material with respect to fatigue properties and fatigue crack growth process [17].

5. CONCLUSIONS

- The temperature field and residual stress field in friction stir welding of 304 stainless steel plates were analyzed in detail. Three dimensional nonlinear thermal and thermo-mechanical simulations were performed for the FSW process using the finite element analysis code - SYSWELD.
- Using inverse approach, thermal modeling is carried out based on the experimental data of transient temperature history at two locations during the FSW. Results showed that due to unknown heat energy input from the process, this inverse analysis method was unique and effective for the calculation of temperature field in the FSW.
- The maximum temperatures determined from the simulation are 944°C at the heat affected zone and 1246°C at the weld line, which is significantly less than the melting temperature of 304 stainless steel at 1550°C.
- By using the thermal database file thermo-mechanical model is generated. From the three dimensional elastic-plastic mechanical simulation, the maximum residual stress along mid transverse section (at the stir zone) is 89 MPa and it is also observed that the numerical analyzed profile are consistent with the experimental data.

REFERENCES

1. Hsieh, Chih-Chun; Wu, Weite: Overview of intermetallic Sigma (σ) phase precipitation in stainless steels, 'ISRN Metallurgy', vol. 2012 , 2012, 361-381.
2. Siddiquee, AN; Sunil, P; Zaman, NK: Friction stir welding of austenitic stainless steel: a study on microstructure and effect of parameters on tensile strength, 'Materials Today: Proceedings', vol. 2, no. 4-5, 2015, 1388-1397.
3. He, Xiaocong; Gu, Fengshou; Ball, Andrew: A review of numerical analysis of friction stir welding, 'Progress in Materials Science', vol. 65, August 2014, 1-66.
4. Zhu, XK; Chao, YJ: Numerical simulation of transient temperature and residual stresses in friction stir welding of 304L stainless steel, 'Journal of Material Processing Technolology', vol. 146, no. 2, 2004, 263-272.
5. Prasanna, P; Subba Rao, B; Krishna Mohana Rao, G: Finite element modeling for maximum temperature in friction stir welding and its validation, 'International Journal of Advanced Manufacturing Technology', vol. 51, no. 9-12, 2010, 925-933.
6. Hattel, JH; Schmidt, HNB; Tutum, C: Thermomechanical modeling of friction stir welding, 'Proceeding of the 8th International Conference on Trends in Welding Research, 2009, 1-10.
7. Xiao, Yihua; Zhan, Haifei; Gu, Yuantong; Qinghua, Li: Modeling heat transfer during friction stir welding using a meshless particle method, 'International Journal of Heat and Mass Transfer', vol. 104, 2017, 288-300.
8. Sanjeev, NK; Vinayak, M; Suresh, H: Effect of coefficient of friction in finite element modeling of friction stir welding and its importance in manufacturing process modeling applications, 'International Journal of Applied Sciences and Engineering Research', vol. 3, no. 4, 2014, 755-762.
9. Schmidt, H; Hattel, J; Wert, J: An analytical model for the heat generation in friction stir welding, 'Modelling and Simulation in Material Science and Engineering', vol. 12, no. 1, 2003, 143-157.
10. Buffa, G; Ducato, A; Fratini, L: Numerical procedure for residual stresses prediction in

friction stir welding, 'Finite Elements in Analysis and Design', vol. 47, no.4, 2011, 470-476.

11. Prime, Michael B; Gnäupel - Herold, Thomas; Baumann, John A; Lederich, Richard J; Bowden, David M; Sebring, Robert J: Residual stress measurements in a thick, dissimilar aluminum alloy friction stir weld, 'Acta Materialia', vol. 54, no. 15, 2006, 4013-4021
12. ESI-Group, SYSWELD reference manual, 2007.
13. Soundararajan, Vijay; Zekovic, Srdja; Kovacevic, Radovan: Thermomechanical model with adaptive boundary conditions for friction stir welding of Al 6061, ' International Journal of Machine Tools and Manufacture', vol. 45, no. 14, 2005, 1577-1587.
14. Zhu, XK; Chao, YJ: Effects of temperature-dependent material properties on welding simulation,'Computers and Structures', vol. 80, no. 11, 2002, 967-976.
15. Bae, Dong Ho; Kim, Chul Han; Cho, Seon Young; Hong, Jung Kyun; Tsai, Chon, Liang: Numerical analysis of welding residual stress using heat source models for the multi-pass weldment, 'KSME International Journal', vol. 16, no. 9, 2002, 1054-1064.
16. Khandkar, MZH; Khan, JA; Reynolds, AP: Prediction of temperature distribution and thermal history during friction stir welding: input torque based model, 'Science and Technology of Welding and Joining', vol. 8, no. 3, 2003, 165-174.
17. Lakshminarayanan, AK; Balasubramanian, V: Assessment of fatigue life and crack growth resistance of friction stir welded AISI 409M ferritic steel joints, 'Material Science Engineering: A', vol. 539, 2012, 143-153 ◘

Dr. S Sree Sabari is currently working as Associate Professor in the Department of Mechanical Engineering, Sree Vidyanikethan Engineering College, Tirupati. He has published 10 research papers in reputed International Journals. He has presented more than 40 research articles in the national and international Conferences. He has made 4 invited talks on the topic "Finite element analysis of Weld structure" in the various Engineering colleges. He is a recipient of "D & H Secheron Award" for the best paper presentation in the International Welding Congress 2013 held at Delhi. His areas of research interest includes Welding Technology, Material Science and Numerical modeling. (E-mail: sreesabaridec2006@yahoo.co.in)

M Chenchu Giri is currently pursuing his undergraduate degree in the Department of Mechanical Engineering, Sree Vidyanikethan Engineering College, Tirupati. His areas of interests are material science and finite element analysis. (E-mail: m.chenchugiri@gmail.com)

Dr. V Balasubramanian is working currently as Professor, Department of Manufacturing Engineering, Annamalai University, Annamalainagar, India. He graduated from Government College of Engineering, Salem, University of Madras in 1989 and obtained his post graduation from College of Engineering Guindy, Anna University, Chennai in 1992. He obtained his Ph. D from Indian Institute of Technology Madras (IITM), Chennai in 2000. He has published more than 300 papers in SCOPUS indexed Journals and supervised 18 Ph.D scholars. His areas of interest are: Materials Joining, Surface Engineering, Nanomaterials, and Fracture Mechanics. He has completed various Sponsored R & D projects from many agencies such as AICTE, UGC, DST, DRDO, ARMREB, DAE, NRB & ARDB, Ministry of Environment & Forest. (E-mail: visvabalu yahoo.com)