ANALYSIS OF STRESS AND TEMPERATURE DISTRIBUTION IN FRICTION STIR WELDING USING NUMERICAL SIMULATION

[1]K.VASU, [2]H.CHELLADURAI, [3]ASHUTOSH KUMAR, [4]DEEPAK KUMAR PRASAD

[1]Assistant Professor, Department of Mechanical Engineering, Indian Institute Information Technology, Design and Manufacturing, Jabalpur-482005, India.
[2]PhD research scholar, Department of Mechanical Engineering, Indian Institute Information Technology, Design and Manufacturing, Jabalpur-482005, India
E-mail: chella@iiitdmj.ac.in

Abstract - In this paper the stress and temperature distribution of friction stir welding have been analysed by using numerical simulation. Several attributes of FSW were analysed such as tool-work piece surface interaction, plastic deformation, and friction-induced heat generation. A nonlinear redirect coupled-field study has been conducted to study the work piece's thermal and mechanical properties of the FSW process. Stresses, Temperature distribution and isotherms are obtained by giving input parameters viz. normal force, shoulder diameter, rotational speed, and feed to the tool. The results obtained from the simulation in ABAQUS software were validated with experimental results published by other researchers. It was noticed that peak temperature value during the welding was lesser dependent on the tool shoulder diameter than other parameters.

Keywords - Friction Stir Welding, Numerical Simulation, Stress Distribution, Temperature Distribution

I. INTRODUCTION

In the past few years numerous methods of joining different materials have been developed and are being utilized in various fields of the manufacturing and production industry. Based on the type of state of welding, metal welding processes can be classified into (i) fusion welding process, and (ii) solid-state welding process. Fusion welding process requires high temperature for melting the base metals and joining them. The solid-state welding process is performed under the application of pressure alone or a mixture of heat and pressure. In solid-state welding process materials are welded together without melting the base material. The friction stir welding (FSW) also comes under solid-state welding process, which was discovered and patented by 'The Welding Institute' (TWI), UK in 1991. It is possible to join materials up to the thickness ranging from 0.5-65 mm through Friction Stir Welding (FSW), such as aluminium and other lightweight materials. Figure 1 shows the schematic diagram for the friction stir welding process. The FSW has four phases such as plunge phase in which the tool starts rotating and plunges into the base material whereas in the dwell phase, tool rotates in its position so that sufficient heat is generated to plastically deform the base material. The weld phase, the tool traverses along the length of the workpiece and last is retracting phase, the tool travel and rotation direction are opposite.

Tool rotating speed, welding speed, and tool pin plunge depth are some of the processing parameters that have an impact on the starting and completion of the weld joint during the FSW process. The mechanical strength of a friction stir welded joint can be optimized by the effective and efficient application of certain process parameters. These parameters are also responsible for temperature and force variation during FSW process. Hence, a comprehensive literature survey has been performed to know the effects of various process parameters on the mechanical properties of friction stir welded joints.

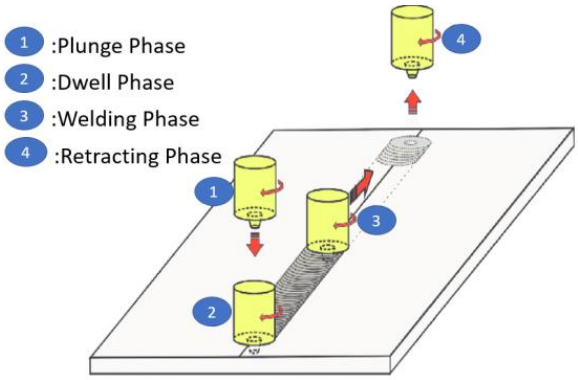


Figure. 1Schematic of Friction Stir Welding

Zhu et al. [1] presented a finite element analysis code in the WELDSIM software for welding simulation to investigate the relationship between transient temperature change and residual stress in a friction stir welded 304L stainless steel plate. A three-dimensional elastic-plastic thermo-mechanical simulation was used to identify the transient temperature field and subsequently to quantify the residual stresses in the welded plate.Under varied welding settings, Tang et al. [2] measured the temperature at several points parallel to the pin tool travel direction and perpendicular to the specimen thickness direction. The welding seems peak temperature was discovered to be less than 0.8m. The orientation of the specimen thickness has little effect

on the temperature. Finally, it was established that the shoulder of the pin tool was crucial to the welding procedure. Chao et al. [3] differentiated the FSW process heat transfer into two boundary value problems (BVP), one for the tool that is in a steady state and one for the workpiece that is in a transient state. Particularly, the findings demonstrated that most of the heat produced by friction, among that about 95%, was transmitted to the workpiece and only 5% flowed into the tool, and the percentage of the rate of plastic work dissipated as heat is approximately 80%. Prasanna et al. [4] created a model for simulation using finite element analysis with better capacity to estimate temperature propagation in stainless steel. The simulation model was analysed with existing experimental results obtained by Zhu et al. [1] on 304 L stainless steel. The eleven simulation findings were well aligned with the outcomes of the experiments. Between the theoretical values for 304 L steel acquired from ANSYS and the experimental values obtained by Zhu, error analysis was performed. Batako et al. [5] discussed a theoretical model and experimental analysis of friction stir welding of copper and aluminium. Friction-stir welding of the 5083-H116 aluminium alloy and pure copper was accomplished by shifting the pin to the aluminium side and managing the FSW parameters. A COMSOL Multiphysics finite element study was used to support this work. Additionally, it was observed that copper maintained a straight baseline while aluminium diffused more quickly at the interface according to energy dispersive spectroscopy. Kim et al. [6] used a commercial finite volume method (FVM) algorithm that is based on the Eulerian formulation to do thermo-mechanical simulation for AA5083-H18 sheets. Under the assumption of a steady state, the distribution of temperature and strain ratewere estimated, and predicted temperature distributions (profile and peak values) were contrasted with experimental data. Balasubramanian et al. [7] examined how the tool shoulder diameter and pin profile affected the production of FSP zones in AA6061 aluminium alloy. Five distinct tool pin profiles viz. straight cylindrical, tapered cylindrical, threaded cylindrical, triangular, and square with three different shoulder widths were used to fabricate the joints. In comparison to alternative tool pin profiles, they discovered that the square pin profiled tool with an 18 mm shoulder diameter generated welds that were mechanically sound and devoid of metallurgical defects. Song et al. [8] presented a three-dimensional heat transmission model on a moving coordinate for friction stir welding (FSW). The control equations were solved using the finite difference method, and the authors concluded that preheating the workpiece before FSW is beneficial.

Researchers from all around the world have conducted several studies in the area of friction stir welding (FSW). The most important constraint in deciding the quality of weld is the tool geometry. Conclusions drawn from the literature survey reveals the following areas in FSW, which needs to be emphasized and worked upon in the future. Limited study has been done on simulation of heat flux in FSW process and demonstration of temperature distribution during FSW process for a higher thickness plate. Heat affected zone is also less explored. Therefore, the motives of this research work are to develop the thermo-mechanical simulation model to show Temperature and stress variation during FSW process. Further, this model also used to find out difference in temperature variation for different input parameters to investigate the heat zone and heat flux for AA5059 plates during FSW process.

II. METHODOLOGY

The surface interaction of the work piece and tool, heat generation from friction, and plastic deformation are only a few of the features of FSW that are discussed. Since the thermal and mechanical behaviours are interdependent and connected a non - linear direct coupled-field analysis is performed throughout the FSW process. Since it can be challenging to locate a complete set of engineering data to replicate the FSW process, the simulation is prioritised over the numerical results in this issue.

Problem Description

Property	Density	Specific Heat	Strength	Young's Modulus
Values	2670 kg/m^3	875 J/kg.K	160 MPa	2×10^{11} Pa

Table 1 Work piece Material AA5059 Aluminium Properties

The model consists of a cylindrical tool and two rectangular steel plates along with a coupled-field solid element with structural and thermal degrees of freedom. The model is subjected to all relevant mechanical and thermal boundary conditions. The plunge, dwell, and traversal phases of the operation are represented by the three load steps that make up the simulation. The rubbing of the tool against the workpiece causes a temperature to develop at the contact surface. FSW typically happens when the temperature in the welded-line area hits 70% to 90% of the material's melting temperature.The temperature that was attained in this example in the weld line region is in the 15-range described by Zhu [1] and Prasanna [4], while the highest temperature that was attained is far below the melting temperature of the workpiece. According to calculations for frictional heat generation and plastic heat generation, most of the heat is produced by friction between the tool shoulder and the workpiece. At the plates' contact interface, a bonding temperature is given to simulate welding done by the tool. When this bonding

temperature is exceeded, the contact turns into a bond when it is still above that temperature.

Geometry configuration and computational formulation

Geometry configuration

On a scaled-down variant, a direct coupled-field analysis is carried out. Additionally, for a more accurate simulation, a rotating and moving tool is used as opposed to the reference model's moving heat source ignoring the tool pin. Since the heat produced at the pin only accounts for around 2% of the overall heat, it is insignificant and neglected. As seen in the Figure 2 and 3, the simulation uses a cylindrical form tool to weld to theAA5059 aluminium plates (the workpiece).

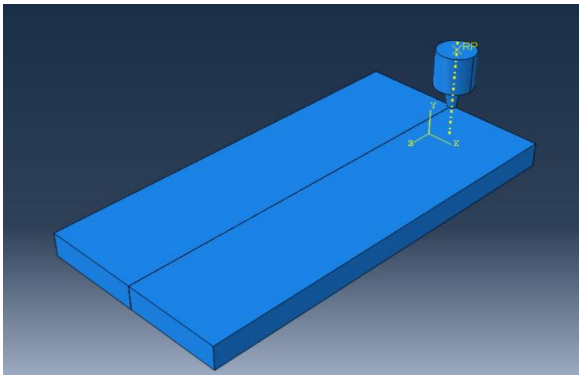


Figure.2 3D Model of Workpiece and Tool

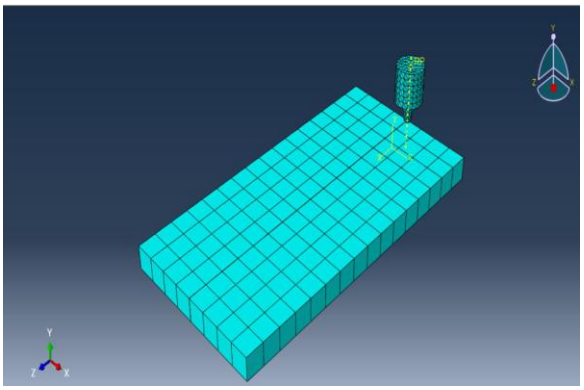


Figure.3 3D Meshed Model of Workpiece and Tool

The tool used in the FSW process is often made of a material that is tougher than the material of the work piece being welded Previously, FSW was utilised for soft work piece materials like aluminium. This example models a cylindrical HSS tool. To replicate the clamping ends, the plate slides parallel to the weld line are restrained in each direction. To imitate support at the bottom, the workpiece's bottom side is restrained in the perpendicular (z) direction. On every surface of the model, heat losses are considered. The weld centre line is the centre of symmetry for all boundary conditions. Three load steps are used to run the simulation, each representing one of the three FSW process phases (plunge, dwell, and traverse).

Modelling

Two rectangular shaped work pieces (like those used in the associating model) are used as the work piece. The plate size is 300 x 150 x 19 mm. The tool shoulder diameter is 15 mm. The thickness of the plate is unchanged from the reference model. The plate width is decreased since the welding process has less of an impact in the areas far from the weld line, and because this example primarily focuses on the heat generation and temperature rise in the area closest to the weld line. To prevent mesh orientation dependency, a rectangular and triangular mesh is employed instead of a tetrahedral mesh. In the weld line region, a finer mesh is used to get more precise results.

Material Properties

Material properties of the plate and tool is described in the Table 1 and 2.

Property	Density	Specific Heat	Hardness	Young's Modulus
Values	8150 kg/m^3	420 J/kg K	65HRc	207 GPa

Table.2 Material Properties of the HSS Tool

Analysis and method

Methods for thermo-mechanical analysis

The incremental theory of plasticity is used in the thermomechanical analysis. The von Mises yield criterion and the corresponding flow rule are presumptively obeyed by the materials' plastic deformation.The interaction between thermal stresses' rate components, σ_{ij}, and strains, ϵ_{ij}, is described by equation 1.

$$\dot{\varepsilon}_{ij} = \frac{1+\nu}{E}\dot{\sigma}_{ij} - \frac{\nu}{E}\dot{\sigma}_{kk}\delta_{ij} + \lambda s_{ij} + \left[\alpha + \frac{\partial \alpha}{\partial T}(T - T_o)\right]\dot{T} \quad (1)$$

Where,α is coefficient of thermal expansion, E is modulus of elasticity, ν is Poisson's ratio, $\dot{\varepsilon}_{ij} = \sigma_{ij} - 1/3\sigma_{ij}\delta ij$ are thecomponents ofdeviatoric stresses, λ is plastic flow factor, $\lambda = 0$ for elastic deformation or $\sigma_e < \sigma_s$, $\lambda > 0$ for plastic deformation or$\sigma_e \geq \sigma_s$.

Boundary Conditions and Loading

There are two types of boundary condition applied on this FSW system.

- Boundary Conditions for thermal analysis
- Boundary Conditions for mechanical analysis

Boundary Conditions for Thermal Analysis

The FSW process generates frictional and plastic heat, which quickly spreads to remote areas of the plates. Convection and radiation are responsible for heat loss to the environment on the top and side surfaces of the workpiece. Additionally, there are

conduction losses from the workpiece's bottom surface to the backing plate. Except for the bottom surface, the convection coefficient for the surfaces of the workpiece ranges from 10 to 30 W/m^2 °C.

Boundary Conditions for Mechanical Analysis

Each plate is clamped to hold the workpiece. The plates' clamped sections are restrained in all directions. All bottom nodes of the workpiece are restrained in the perpendicular direction to simulate support at the bottom of the plates (z direction).

Loading

There are three main stages to the FSW process:

- The Plunge – In this phase the tool is pushed slowly into the workpiece
- Dwell – In this phase heat is produced at the initial tool position because of friction between the rotating tool and the workpiece, which continues until the workpiece reaches the required welding temperature.
- Traverse (or travelling) – In this phase the rotating tool moves along the weld line to produce the weld.

III. SIMULATION IN COMSOL

Model definition

For 19mm thickness aa5059 plates with tapered tool pin:

Around the weld, the model geometry is symmetrical. Therefore, modelling just one aluminium plate is sufficient. The plate dimensions are 300×75×19 mm, surrounded by two infinite domains in the x-direction. Figure 4 shows the resultant geometry of the model. Input parameters for the welding is described in the Table 3.

Rotational speed (N)	600 rpm
Normal force (F_n)	10 kN
Shoulder radius ($r_{shoulder}$)	15mm
Avg. Pin radius (r_{pin})	7mm

Table. 3 Input parameters for 19mm thickness AA5059 plates with tapered pin tool.

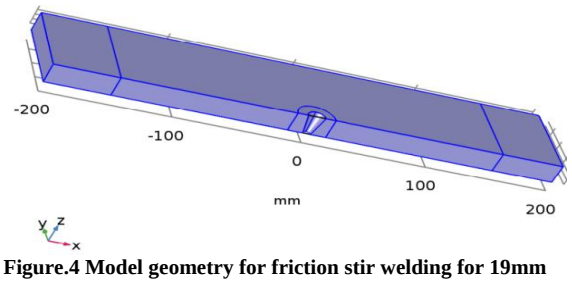


Figure.4 Model geometry for friction stir welding for 19mm plate with tapered pin

Simulation of Heat Flux

The simulation of heat flux distribution during friction stir welding has been done using COMSOL multi-physics software. The heat transfer in the plate is described by the following equation. The equation also contains a convective term in addition to the conductive term because of setting the coordinate system in the welding tool. The equation 2 describes heat conduction.

$$\rho C_p u \cdot \nabla T + \nabla \cdot (-k\nabla T) = Q \qquad (2)$$

Where, k isthermal conductivity k, ρis density of the material, C_p is specific heat capacity of material and u is velocity of the tool.

The model sets the velocity to 15×10^{-3}m/min the negative x direction. Both the heat at the tool pin/workpiece contacts and the heat at the tool shoulder/workpiece are considered. Heat input from the tool pin is given by equation (3)

$$q_{pin}(T) = \frac{\mu}{\sqrt{3(1+\mu^2)}} r_p \omega \bar{Y}(T) \qquad (3)$$

Where μ is friction coefficient, r_p is radius of pin, ω is angular velocity of pin, and $\bar{Y}(T)$ is average shear stress of the material.

IV. RESULTS AND DISCUSSIONS

Stresses for AA5059 Aluminium in ABACUS:

During the welding process in AA5059 Aluminium, change in values of various quantities come into effect. The Figure 5 shows the values of Von Misses stresses in weld phase.The maximum value of Von-Mises and maximum Principal stresses is 202.4 MPa and 203.9 MPa respectively which is more than the yield strength of AA5059 Aluminium (150MPa). From the results it can be interpreted that in the plates plastic deformation is taking place, due to which material flows during the welding process. As the tool rotates large amount of stress is generated to deform material plastically. As the tool traverse along the weld path, the material gets completely yielded due to the heat generated by the tool rotation and the stresses get reduced and material flows as the tool moves forward.

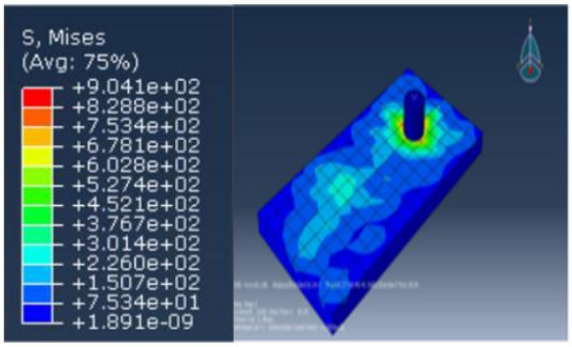


Figure.5 Von Mises during the welding phase

Temperature Results for AA5059 Plate in COMSOL

For 600 rpm tool rotation speed with average pin radius of 7mm, the temperature distribution in shown is Figure 6. The temperature is maximum at the centre where tool rotates, and from heat is conducted to the rest of the base plate and gets convected from upper and lower surfaces of the base plates.

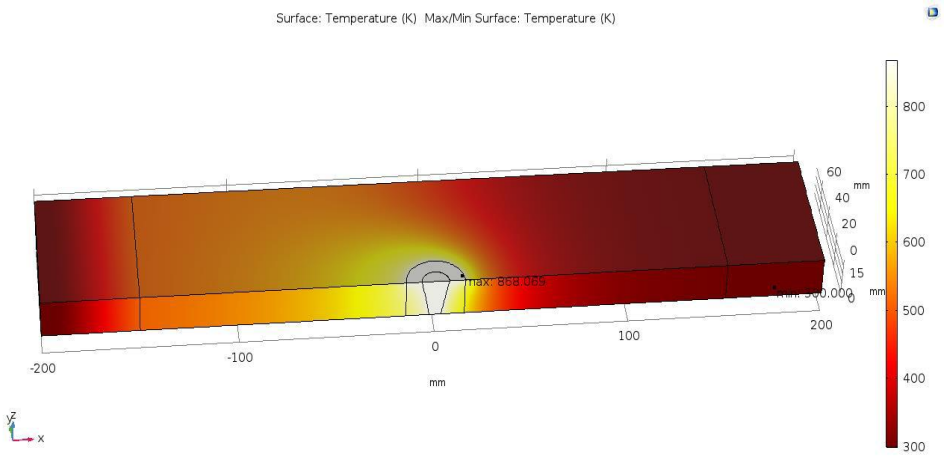


Figure.6 Temperature Contour for 19mm thick plate

Heat Zone in AA5059 Plates Of 19mm Thickness In COMSOL

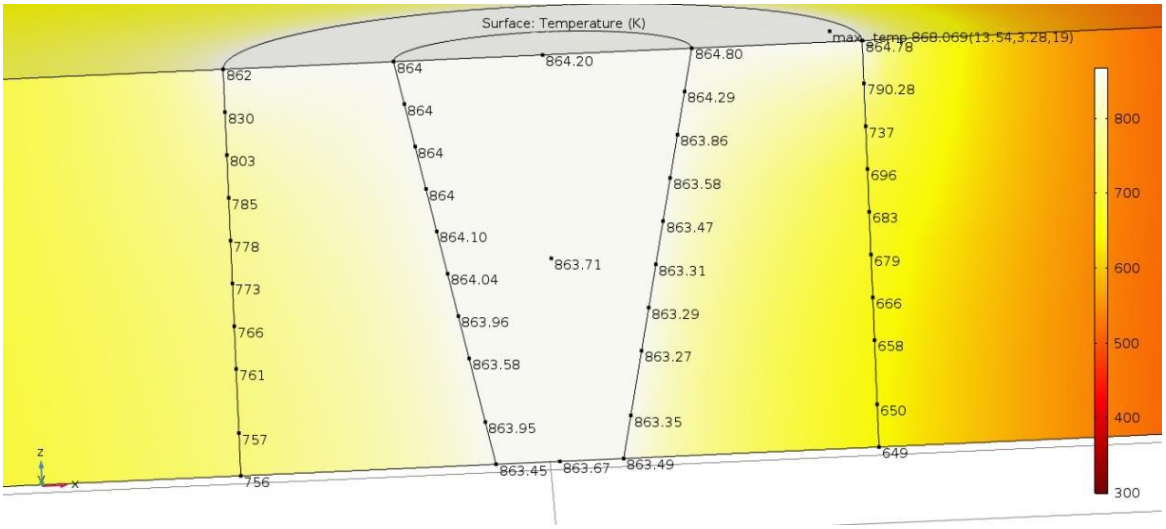


Figure.7 Temperature distribution under the shoulder and pin area

The distribution of temperature for AA5059 plates with 19mm thickness is shown in the Figure 7. The various heat zones obtained in the COMSOL software tells that maximum temperature is at the tool shoulder and workpiece top surface. From the temperature distribution one can find which tool material to use in FSW process.

V. SUMMERY AND CONCLUSION

In this simulation, Stresses, Temperature distribution and isotherms were obtained by giving input parameters viz. Normal force, shoulder diameter, rotational speed, and feed to the tool. The results obtained from the simulation in ABAQUS software were compared with the research paper followed during the simulation and it is observed that the results obtained from simulation verify the experimental values of research paper. Based on the above investigations, the following conclusions can be drawn:

1. Most of the heat is produced mostly through friction, along with some heat generated through plastic deformation.
2. At constant speed of tool (600 rpm), the tool feed for 19mm thick plate is 15mm/min.
3. Peak temperature value is less dependent on the tool shoulder diameter, as it changes very slightly with the increasing tool shoulder diameter in comparison with other input parameters.
4. Experimental results of the output responses considered are compared with the simulated results and it is concluded that the experimental

and simulated results are in close approximation with each other.

REFERENCES

[1] X.K. Zhu, Y.J. Chao, "Numerical simulation of transient temperature and residual stresses in friction stir welding of 304L stainless steel", Journal of Materials Processing Technology 146 (2004) 263–272

[2] W. Tang, X. Guo, J.C. McClure, L.E. Murr, A. Nunes, Heat input and temperature distribution in friction stir welding, J. Mater. Process. Manuf. Sci. 7 (1998) 163–172.

[3] Yuh J. Chao, X. Qi, W. Tang, "Hear transfer in Friction stir welding-Experimental and Numerical studies", J. Manuf. Sci. Eng 125(1), 138-145 (Mar 04, 2003)

[4] Prasanna, P., B. S. Rao, and G. K. Rao. "Finite Element Modeling for Maximum Temperature in Friction Stir Welding and its Validation." Journal of Advanced Manufacturing Technology. 51 (2010): 925-933.

[5] Andre D. L. Batako,Ahmed O. Al-Roubaiy&Saja M. Nabat, "Experimental and theoretical analysis of friction stir welding of Al–Cu joints", Int J Adv ManufTechnol (2014) 71:1631–1642

[6] Dongun Kim, Harsha Badarinarayan, Ji Hoon Kim, Chongmin Kim, Kazutaka Okamoto, R.H. Wagoner, Kwansoo Chung, "Numerical simulation of friction stir butt welding process for AA5083-H18 sheets", European Journal of Mechanics A/Solids 29 (2010) 204-215

[7] KaveryElangovan and V. Balasubramanian, "Influences of tool pin profile and tool shoulder diameter on the formation of friction stir processing zone in AA6061 aluminium alloy", Materials and Design 29(2):362-373 • December 2008.

[8] H. K. Mohanty, M. M. Mahapatra, P. Kumar, P. Biswas and N. R. Mandal, "Effect of Tool Shoulder and Pin Probe Profiles on Friction Stirred Aluminium Welds – a Comparative Study", J. Marine Sci. Appl. (2012) 11: 200-207.

★★★