Simulation of friction stir welding using thermo-mechanical coupled finite element method

J. Abhilash, Apeksha, B. Acharjee*
Department of Production Engineering, BIT Mesra, Ranchi – 835215, India.
bappa.rana@gmail.com

Abstract. Friction stir welding (FSW) is a solid-state welding process for joining of two metals where the heat is generated from the mechanical friction between the work materials and a rotating tool. Also, additional heat is generated by plastic deformation of the workpiece material. In this paper, the process involved in this technique of welding without using any filler metal is simulated using ANSYS® parametric design language. A thermo mechanical coupled model is developed to analyze the thermal and mechanical responses during the welding process. The method of direct coupling is applied by using SOLID 226 thermo-mechanical coupled element. Unlike the sequential 1-way coupling method where the result from thermal analysis is used as input data for structural model, direct coupling solves both the thermal and mechanical boundary conditions simultaneously for the coupled model. Coupled analyses are more computationally intensive. Usually, physics of coupling is ignored or simplified but a coupled analysis provides more realistic results.

1. Introduction
Friction stir welding (FSW) is a solid-state joining process developed and patented by The welding Institute (TWI) in 1991, in which the welding is carried out without melting the work materials. Friction stir welding is used to weld metal plates by employing a specially designed non-consumable rotating tool [1]. During friction stir welding, heat is not supplied to the weld zone from any external source, but generated with the material by friction between work piece and high speed rotating tool. Tough the major source of the heat is friction between tool and work piece, a part of heat is also generated due to the plastic deformation of work material during mechanical working by the tool. The amount of heat generation due to friction and plastic deformation and at work-tool interface during the FSW depends mainly on welding parameters and geometry of the tool [2,3]. The heat generated during the process is transferred to both workpiece as well to the tool. The rate of heat transfer is governed by the rate of heat generation and heat conduction within material and convection losses to the surroundings. Amount of heat conduction to the plate plays a major role in determining the temperature distribution across the work piece. Thus, proper and detailed thermal modelling of FSW is required to be performed to predict the temperature distribution during FSW relating to tool geometry, tool rotation and welding speed. If expressed in a simple way, friction stir welding is carried out mainly in three steps according to the relative motion between the tool and the workpiece in a given time period. In the first step, the rotating tool is vertically penetrated into the joint line (plunge period). This period is followed by the dwell period in which the tool keeps on rotating at same place without moving forward along the line. Heat is produced due to frictional work and material plastic
deformation work at rotating tool and the stationary work piece interface. This heat is dissipated into the neighboring material, leading to an increase of temperature and consequent material softening. After these two initial steps the welding operation can be initiated by moving either the tool or the work piece relative to each other along the joint line [4]. During FSW, the tool rotates and also translates along the weld line, which involves synchronized use of pressure and relative motion, which heats up the metals above a threshold temperature where metals starts to flow plastically [5]. The applied pressure perpendicular to the plane of motion serves to extrude the heated material including any dirt and oxide films from the interface, bringing the components to be joined into intimate contact [6,7]. FSW comprises of several highly coupled, non-linear and transient physical phenomena, including large plastic deformation, material flow, mechanical stirring, surface interaction between the tool and the work piece, dynamic structural evolution and heat generation resulting from friction and plastic deformation. The behavior of the FSW joints is not only influenced by the geometry of the tools and the joints but also by different process parameters [8]. The experiments are often time consuming and costly. Thus, numerical investigations are becoming very popular [9-13]. In present work an attempt is made to develop a thermo-mechanical model of friction stir welding process to predict the thermal and mechanical responses during the welding process.

2. Model description

2.1. Geometric configuration
Two thin plates, each 76.2 mm in length, 31.75 mm in width and 3.18 mm in thickness made of 304L stainless steel were welded together using the FSW process. The tool is made of PCBN (Polycrystalline cubic boron nitride) cutting material and has a diameter of 15.24 mm with a height approximately equal to its diameter. The plates (workpieces) are clamped on the edges using L-shaped clamps and are supported with a large base plate on the bottom. The bottom support was simulated by constraining the component of displacement in z direction on the bottom side. As the plates are similar in mesh and dimensions, all boundary conditions are symmetric across the weld centerline. Model geometry is reduced using symmetry to reduce the simulation time.

2.2. Mesh generation
Any complex continuum problem can be solved by The Finite Element Method (FEM). The method works by subdividing the problem into a series of simple interrelated problems (mesh generation). FEM is most commonly utilized in numerical analysis for getting approximate solutions to wide range of engineering issues. A full Newton-Raphson method is used for better convergence. As this is a problem involving contact analysis, this program couples the unsymmetrical contact stiffness matrix with the sliding and the normal stiffnesses. This can be done using the command: NROPT, UNSYM in ANSYS® [14].

A coupled field element, SOLID 226 with structural-thermal option is used to create the geometric model of workpiece and tool. A hexahedral mesh with dropped mid-side nodes is preferred over a tetrahedral mesh because a tetrahedral mesh can lead to dependency of the solution on mesh-orientation. To avoid any oscillations in the thermal solution, mid-side nodes were dropped (or quadratic interpolation functions) as it can lead to nonphysical temperature distribution. Figure 1 shows the 3-D meshed model of tool and plates.

The hexahedral mesh has a total of 22257 nodes, 6765 elements and an extra pilot node to control the movement of tool. A finer mesh near the weld-line region gives more accurate results.

2.3. Governing equation
Following Fourier’s equation is the governing equation for the temperature calculations of the model:

\[ \rho c \frac{dT}{dt} = \text{div} k \text{grad}T + Q_{\text{int}} \]  

(1)
where $Q_{int}$ is the internal heat resource which here represents the power generated due to the frictional stress developed between the tool and the workpiece, and also due to the plastic deformation at weld zone. $k$ is the coefficient of thermal conductivity, $\rho$ is the material density, $T$ is the temperature and $c$ is the heat capacity.

2.4. Contact analysis
The simulation involves three main contact analyses. The contacts that are needed to be modelled along with the parameters are as follows:

2.4.1. Contact between the two plates: Two contact pair elements CONTA175 and TARGE 174 are used to define a standard surface to surface contact between the plates. A high thermal contact conductance (TCC) value of $2E06$ W/m$^2$°C is used to simulate a perfect thermal contact between the plates to ensure continuous bonding. The close metal to metal contacts changes to metallic bonds as and when the temperature at the contact surface exceeds the bonding temperature (i.e., 1000°C). The following commands can be used to make these changes:

- et, ITYPE1, TARGE170
- et, ITYPE2, CONTA174
- keyopt, ITYPE2,1,1
- keyopt, ITYPE2,4,3
- rmodif, ITYPE1,14,2e06
- rmodif, ITYPE1,35,1000

2.4.2. Contact between tool and workpiece: This contact was also modelled using CONTA174 and TARGE 170 contact pair element. The top surface of the workpiece is assigned as contact surface by CONTA174 element and the TARGE170 element is used for the tool (Figure 2).

Two real constants are specified to model friction-induced heat generation viz FHTG and FWGT. The FHTG real constant is set to 1 which means all the frictional dissipated energy gets converted into heat. The second one FWGT that accounts for the distribution of heat between contact and target surfaces is defined next and is set to 0.95. Such high value of FWGT implies that most of the frictional heat generated flows into the workpiece and only five percent flows into the tool. Since most of the heat generated is transferred to the workpiece so a low TCC value (10 W/m$^2$°C) is specified. Some
additional heat is also generated by plastic deformation of the work piece material. A variable coefficient of friction is defined varying from 0.4 to 0.2.

2.4.3. Contact between pilot node and tool surface: A pilot node is created either using contact manager of using ESURF to control the movement of the tool. The top surface of the tool is defined using CONTA174 and the pilot node is defined as TARGE170. The coordinates for Pilot node in this case was 0, 7.62 mm, 15mm (Figure 3).

The following contact settings are used for the CONTA174 elements:
- KEYOPT (2) = 2 sets contact algorithm to MPC algorithm.
- KEYOPT (4) = 2 specifies a rigid surface constraint for the contact nodes.
- KEYOPT (12) = 5 assigns the behavior of contact surface as bonded (always)

2.5 Boundary Conditions
The thermal and mechanical boundary conditions applied on the FSW model is explained in this section.
2.5.1. Thermal Boundary Conditions: The heat generated due to friction and plastic deformation at weld zone propagates speedily into regions away from the weld line of the plates. The top and side surfaces of the workpiece lose heat to the surrounding through convection and radiation. A huge amount of heat is lost via the bottom surface of the workpiece to the backing plate. From the research it was found that the value of the convection film coefficient, $h$ is $30 \text{ W/m}^2\text{°C}$ for workpiece and tool surfaces whereas it is $300 \text{ W/m}^2\text{°C}$ for the bottom surface of the plates. These coefficient values were decided by trial and error in the previous literatures. The large value accounts for the higher amount of heat loss due to transfer via conduction to the large plate beneath the workpiece. An initial temperature of 25°C is applied on the model and also the reference temperature is set to 25°C.

2.5.2. Mechanical Boundary Conditions: Workpiece is clamped by constraining 20% of each of the plates. All the nodes lying in the clamped regions of the plates are constrained in all directions (Figure 4). Similarly, to provide support at the bottom of the plates, all nodes at the bottom of the workpiece are constrained in the vertical direction (z direction). The first two stages of FSW, namely plunge and dwell is carried upto 6.5 seconds. The last step traverse is simulated upto 29 seconds time.

![Figure 4. Clamping of the workpiece from sides and bottom](image)

3. Results and Discussion
The numerical simulation is carried out in ANSYS® Mechanical APDL (MAPDL), commercial FEM software by analyzing a three-dimensional FEA model of the welding process.

Figure 5 presents the temperature contours after 6.5 seconds (after plunging and dwelling period). The maximum temperature attains at weld zone (for this case) is 1031°C, which is more than the bonding temperature of the selected material, above which the metal starts to flow under application of pressure. The zone above this temperature forms a pool of plasticized metal. Friction stir welding changes the metal’s state from solid into a "plastic-like" state, and then under pressure mechanically stirs the materials together to form a welded joint. Frictional heat is generated at the tool-work piece interface by the tool under the action of tool’s vertical load, thereby reducing the material flow stress. Unlike conventional welding processes, there is no actual melting is involved in this process. Also, the weld produced is of the same fine-grained condition as the parent metal.

In friction stir welding the temperature at weld zone can be controlled by three process variables namely tool rotation speed, travel speed and tool pressure. The model is parametrically built and, therefore, any values of these three process variables can be taken as inputs to find out the suitable parametric values for a full penetration friction stir weld joint.
Figure 5. Temperature contours after 6.5 seconds (after plunging and dwelling period)

Figure 6. Temperature contours after 29 seconds (tool traversing)

Figure 6 presents the temperature contours after 29 seconds. Maximum temperature attained during the welding is 1092°C. It is seen from the Figures 5-6, that the temperature raises steeply during dwell period. Once the bonding temperature is reached, the tool is traversed to complete the weld. Temperature at a point on weld line starts to decrease once the tool left that position and moves forward.
Figure 7 presents the displacement of the plates along z direction under the pressure of the tool. Due to this deflection high stresses are developed on the workpiece beneath the tool, as shown in this Figure 8.

![Figure 7. Displacement of the plates along z direction](image1)

![Figure 8. Von Mises stress developed on the plates developed after plunging](image2)

4. Conclusion

A thermo mechanical coupled model for analysis of friction stir welding is presented in this paper. The model is parametrically built which can be applied for prediction of temperature and stress/strain profile during friction stir welding for a combination of process parameters at different levels. The parameters considered in this model are tool diameter, plunge depth, tool rotation speed, and welding speed which affects the peak temperature. The devolved model can be used for process optimization.
Further refinement of the model by incorporating the physical aspects, those are ignored as simplification assumptions, will be carried out as future scope of this work.

References
[1] Biro A L, Chenelle B F and Lados D A 2012 Processing, microstructure, and residual stress effects on strength and fatigue crack growth properties in friction stir welding: A review Metall. Mater. Tran. B 43 1622.
[2] Kumar H M A, Ramana V V and Shanmuganathan S P 2018 Experimental investigation of mechanical properties and morphological studies on friction stir welded aluminum 2024 alloy, Mater. Today: Proc. 5 700.
[3] Mosallaee M and Dehghan M 2014 Improvement of structural and mechanical properties of Al-1100 alloy via friction stir processing, J. Mater. Eng. Perf. 23 3786.
[4] Mehta K P and Badheka V J 2017 Hybrid approaches of assisted heating and cooling for friction stir welding of copper to aluminum joints J. Mater. Process. Technol. 239, 336.
[5] Lotfi A H and Nourouzi S 2014 Effect of welding parameters on microstructure, thermal, and mechanical properties of friction-stir welded joints of AA7075-T6 aluminum alloy Metall. Mater. Tran. A 45 2792.
[6] Sorensen C D and Nelson T W 2007 Friction stir welding of ferrous and nickel alloys Friction stir welding and processing ed. M W Mahoney and R S Mishra (Materials Park: ASM International).
[7] Bilgin M B and Meran C. 2012 The effect of tool rotational and traverse speed on friction stir weldability of AISI 430 ferritic stainless steels Mater. Des. 33 376.
[8] Liu F C and Ma Z Y 2008 Influence of tool dimension and welding parameters on microstructure and mechanical properties of friction-stir-welded 6061-T651 aluminum alloy Metall. Mater. Tran. A 39 2378.
[9] Meshram A D 2014 Finite element modeling of friction stir welding-thermal analysis Int. J. Eng. Res. Technol. 3 159.
[10] Wang X J, Lu Y X, Zhang Z K, Liang J L and Guo T K 2013 The temperature field test and numerical simulation of steel’s friction stir welding process Adv. Mater. Res. 706-708 370.
[11] Chao Y J, Qi X and Tang W 2003 Heat transfer in friction stir welding - experimental and numerical studies J. Manuf. Sci. Eng.-Trans. ASME 125 138.
[12] Ferro P and Bonollo F 2010 A semianalytical thermal model for fiction stir welding Metall. Mater. Tran. A 41 440.
[13] Vignesh R V, Padmanaban R, Arivarasu M, Thirumalini S, Gokulachandran J and Ram M S S S 2018 Numerical modelling of thermal phenomenon in friction stir welding of aluminum plates IOP Conf. Ser.: Mater. Sci. Eng. 310 012045
[14] ANSYS Help Navigator.