Jurnal
Teknologi

Full Paper

# SIMULATION ANALYSIS OF PEAK TEMPERATURE IN WELD ZONES DURING FRICTION STIR PROCESS

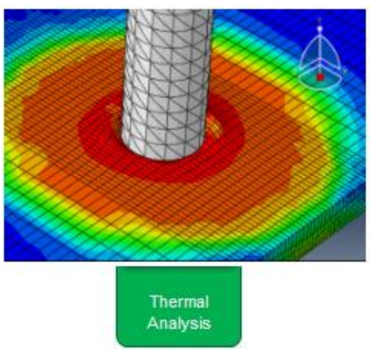
M. Shamil Jaffarullah, Nur'Amirah Busu, Cheng Yee Low*, J. B. Saedon, Armansyah, Mohd Saiful Bahari Shaari, Ahmed Jaffar

Faculty of Mechanical Engineering, Universiti Teknologi MARA, Malaysia

**Article history**
Received
*31 January 2015*
Received in revised *form*
*30 April 2015*
Accepted
*31 May 2015*

*Corresponding author
chengyee.low@salam.uitm.edu.my

**Graphical abstract**




## Abstract

A three-dimensional (3D) finite element model was created to simulate the friction stir welding process of 6063-T5 aluminum alloy. The analysis studies the fundamental knowledge of FSW process with respect to temperature difference in material to be joined and to simulate the temperature distribution in the workpiece as a result of a Friction Stir Welding. The simulation uses HyperMesh and HyperView solver from Altair Hyperworks. The simulation provides better understanding for the peak temperature distribution in the friction stir process. Two cases have (i) constant traverse speed, but varying been simulated rotational speed, and (ii) constant rotational speed, but varying traverse speeds. Simulation results show the peak temperatures increased when the traverse and rotational speeds were increased.

*Keywords*: Friction stir welding, AA 6063 T-5, peak temperature, simulation, altair hyperworks



## 1.0 INTRODUCTION

Friction Stir Welding (FSW) was invented by Wayne Thomas in 1991 at The Welding Institute (TWI), Cambridge, United Kingdom, and patented in 1995 [1]. The FSW process consists of the following three phases, (a) plunging, (b) traversing and (c) retracting. Heat is generated, due to friction and plastic deformation at the tool and workpiece interface, during the process [2]. In FSW, a cylindrical, rotating, non-consumable tool, normally consisting of a pin and a shoulder, is slowly plunged into the joint line between two workpieces of sheet or plate material. Heat, due to friction, is generated between the wear-resistant welding tool and the work pieces; causing the latter to soften without reaching the melting point and allowing the tool to traverse along the weld line [3]. Finally, the tool is retracted leaving a solid phase bond between the two plates. Friction Stir Welding is considered to be (i) a green technology, due to its efficiency; whereby it does not require finishing processes after welding, such as grinding or heat treatment, and (ii) an environmentally friendly process that requires no filler material or shielding gases. FSW also reduces manufacturing costs, increases productivity, and provides improved mechanical and corrosion properties [4]. The objective of the present study is to clarify the peak temperature field in the vicinity of the FSW-tool in finite element using Altair Hyperwork. Even though the actual temperature in the nugget (where severe plastic deformation is induced) cannot be measured directly, the results obtained represent a cornerstone for the verification of models in future experimental studies.

## 2.0 MODEL DESCRIPTION

Two 6063-T5 Al alloy plates, each with a dimension of 150 x 50 x 6 mm, are butt welded in FSW. Figure 1(A) shows the picture of the setup used for tool, workpieces information and process condition while Figure 1(B) shows mesh generation in Altair Hyperwork.

The tool (made of H13 tool steel) consists of shoulder and pin, with diameter of 20 and 7 mm, respectively.

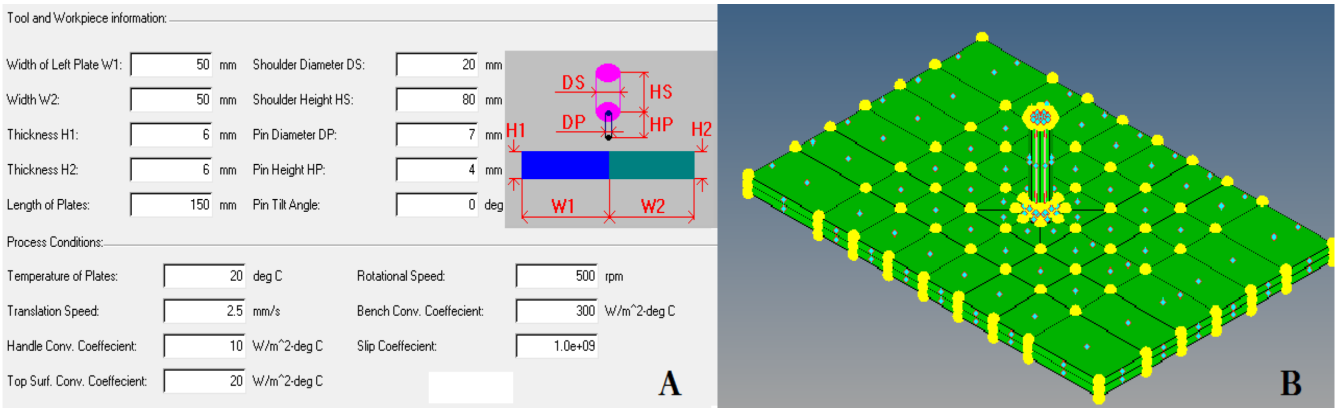



**Figure 1 (A)** Setup utilized for tool, workpieces data and methodology condition in Altair Hyperwork, **(B)** Generate mesh and nodes

The assumptions made when characterizing the loads and boundary conditions for the simulation are (i) the coefficient of friction is expected to differ somewhere around 0.4 and 0.5 in view of the surface temperature at the contact of tool and workpiece; (ii) the tool pin is assumed to be cylindrical, and only its thermal effect is considered in the model; (iii) the radiation heat loss is ignored as it is extensively less contrasted with the conduction and convection losses [4]. The finite element thermo-mechanical model uses the temperature varying material properties (thermal conductivity, specific heat and density) for both the tool and workpiece. There is assumed to be no material melting, since the maximum temperature is maintained below the melting temperature of the aluminium alloy (i.e., 616°C).

FSW simulations were performed with various tool rotational and traverse speeds. The amount of the heat transferred to the tool dictates the life of the tool and the capability of the tool to produce a good processed zone [5]. The maximum temperature made by the FSW process ranges from 70% to 90% of the melting temperature of the work-piece material [6, 7], with the goal that welding defects and distortion commonly occurred with fusion welding, are minimized or avoided. Subsequently, understanding the distribution of heat and obtaining the temperature profiles, is strikingly huge in understanding the general process of friction stir welding.

## 3.0 HEAT TRANSFER MODEL

The principle heat source in FSW is generally considered to be the friction between the rotating tool and the welded plates, and the 'cold work' in the plastic deformation of the material in the vicinity of the tool The heat generation from the plastic deformation of the material is considered to some extent, in the model with the use of variable friction coefficient, but not explicitly accounted for as a heat source. The heat generated at the surface of the tool is transferred into the tool following the Fourier's law of heat conduction. The heat transfer equation, for the tool in a static coordinate system, is [8]:

$$\rho c \frac{\partial (T)}{\partial t} = \frac{\partial}{\partial x}\left(k_x \frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(k_y \frac{\partial T}{\partial y}\right) + \frac{\partial}{\partial z}\left(k_z \frac{\partial T}{\partial z}\right) \qquad \text{(Eq. 1)}$$

Where ***T*** is the temperature, ***c*** is heat capacity ***P***, is the density and $\boldsymbol{k_x}$, $\boldsymbol{k_y}$, $\boldsymbol{k_z}$ are heat conductivities that vary with temperature in the calculations.

## 4.0 FINITE ELEMENT SIMULATION

In this study, the finite element simulation procedure for the FSW process is developed using various options in the general-purpose Altair Hyperwork code. The three-dimensional geometry described above is divided into four-node quadrilateral elements. Figure 2 shows the fine meshing involved in the modeling process.

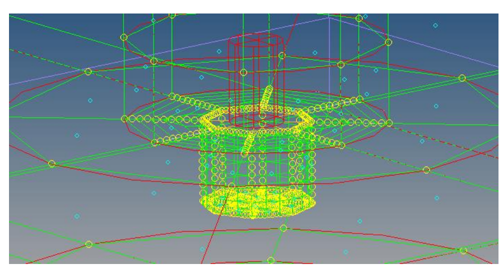


**Figure 2** Fine meshing at the pin

The smallest elements are placed nearest to the tool pin boundary and have the dimension of 0.20 mm × 0.30 mm. The material particles move with a constant speed of v relative to the pin in the direction opposite to the translation movement of the pin.

## 5.0 MATERIAL

Aluminium alloy 6063 is one of the most extensively used, of the 6000 series, aluminium alloys. It is a versatile, heat treatable, extruded alloy, with medium to high strength capabilities.

Aluminium alloy 6063 is typically used for heavy duty structures in rail coaches, aircraft, and aerospace applications and components including transport, boiler making, rivets and motorboats [2]. Table 1 represents the composition of aluminium alloy AA6063 – T5 that is used in this study. Meanwhile, Figure 3 represents the physical properties and thermal properties of this material.

**Table 1** Composition of Aluminium Alloy AA6063 – T5 `

| Component | Wt. % | Component | Wt. % | Component | Wt. % |
|---|---|---|---|---|---|
| Al | Max 97.5 | Mg | 0.45 - 0.9 | Si | 0.2 - 0.6 |
| Cr | Max 0.1 | Mn | Max 0.1 | Ti | Max 0.1 |
| Cu | Max 0.1 | Other, each | Max 0.05 | Zn | Max 0.1 |
| Fe | Max 0.35 | Other, total | Max 0.15 | | |

## 6.0 SIMULATION RESULT AND DISCUSSION

Simulations are performed with different process parameters. The traverse speeds considered are 1.5, 2.0, 2.5 and 3.0 mm/s while the tool rotational speeds used are 500, 800 and 1100 rpm. Table 2 gives the input welding parameters used in the finite element calculation with different process parameter. Figure 3 shows the flow process to achieve in order to get the peak temperature distribution in simulation. The plasticization of material under the tool increases with increase in rotation speed and with decrease in tool traverse speed resulting in the reduction of vertical force. The finite element simulation coupled the moving tool with the workpiece and also considered the thermal effect of the initial tool pin penetration before the start of the weld.

During the penetration phase, the rotating tool pin penetrated into the workpiece, until the tool's shoulder comes in contact with the workpiece. Figure 4 presents cross-sectional views of the calculated temperature contours in the workpiece especially temperature near the pin tool. The cross-sectional views graphically illustrate the peak temperature history of the workpiece and the tool during the welding process. When the tool approaches to the material, the temperature goes up quickly. From the Figure 4, the graph shows the temperature first increased sharply. As tool is passing though the welding distance, the temperature begins to fluctuate and finally the temperature showed a steady decreasing trend; because of the shear friction factor had a negative correlation with the temperature, and the variation of the friction heat affected the temperature.

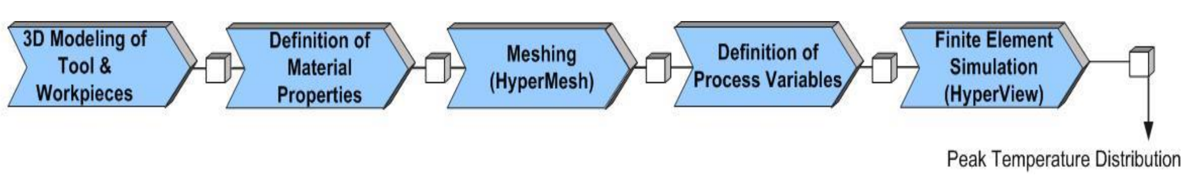



**Figure 3** Process flow of the simulation

**Table 2** Temperature at the tool workpiece interface under different process parameters

| Rotation Speed (rpm) | Traverse Speed (mm/s) | Peak Temperature (°C) |
|---|---|---|
| 500 | 1.5 | 467 |
| | 2.0 | 453 |
| | 2.5 | 438 |
| | 3.0 | 418 |
| 800 | 1.5 | 534 |
| | 2.0 | 519 |
| | 2.5 | 507 |
| | 3.0 | 499 |
| 1100 | 1.5 | 611 |
| | 2.0 | 595 |
| | 2.5 | 583 |
| | 3.0 | 573 |

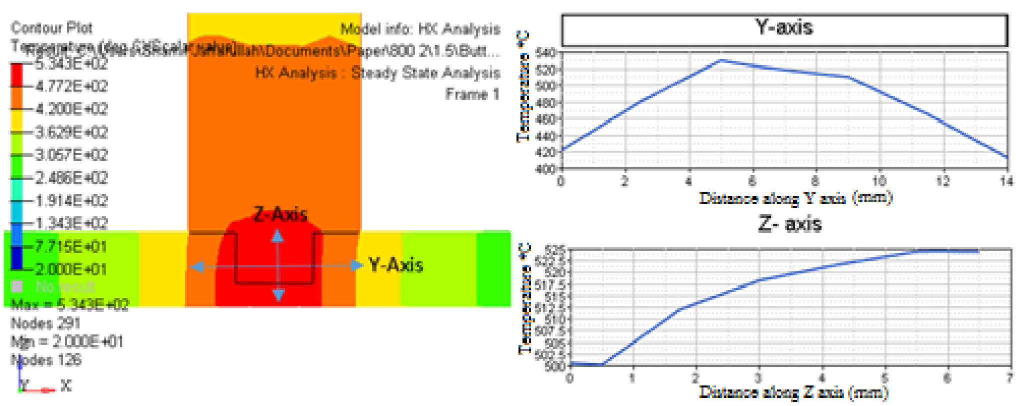



**Figure 4** Cross-sectional views of the temperature contours and graph of temperature plot near the pin tool

Figure 4 shows the modelling temperature contour distribution when the tool moving along the welding line at mid position of the work-piece. Figure 4 shows that the maximum temperature reaches approximately 534°C in the region near the pin tool that is below the melting temperature of AA 6063 (616°C) and is within accepted temperature range in FSW process.

## 7.0 CONCLUSION

In the present study, a three-dimensional simulation model has been developed to determine the temperature distribution during FSW of AA 6063-T5. The finite element model was replicated using Altair Hyperwork, with improved capabilities to predict maximum temperature. A heat source with a heat distribution simulating the heat generated from the friction between the tool shoulder and the workpiece is used in the heat transfer analysis. Peak value and distribution of welding temperature are key factors to affect the properties of welding joint. Consequently, the temperature field is always the research emphasis in the field of welding.

The effect of process parameters on peak temperature showed an increasing in peak temperature as rotational and traverse speeds increased. Further studies, such as observation through experimental works or comparing and validating the results obtained from different simulation tools, are required to validate the results.

## Acknowledgement

The research team wishes to thank to the Ministry of Education Malaysia under the grant numbers 600-RMI/ERGS 5/3 (16/2013) and 600-RMI/ERGS 5/3 (82/2012) for the financial support and the Research Management Institute (RMI) of UiTM for the management assistant.

## References

[1] Armansyah, I. P., Almanar, M., Saiful Bahari Shaari, M., Shamil Jaffarullah, Nur'Amirah Busu, M. and Amlie A. Kasim. 2014. Temperature Distribution in Friction Stir Welding Using Finite Element Method. *International Journal of Mechanical. Aerospace. Industrial and Mechatronics Engineering*. 8(10): 1625-1630.

[2] Dawes, C. J. and Thomas, W. M. 1996. Friction Stir Welding for Aluminum Alloys. *Weld. J*. 75(3]: 41-45.

[3] Mishra, R. S. and Ma, Z. Y. 2005. Friction Stir Welding and Processing. Reports: A Review Journal. Science direct. *Materials Science and Engineering*. R50: 1-78.

[4] Manthan Malde. 2009. Themo Mechanical Modeling and Optimization of Friction Stir Welding. M.S Thesis. Osmania University. Hyderabad. India.

[5] Khandkar, M. Z. H., Khan, J. A. and Reynolds, A. P. 2003. Predictions of Temperature Distribution and Thermal History During Friction Stir Welding: Input Torque Based Model. *Science and Technology of Welding and Joining.* 8(3): 165-174.

[6] Prasanna, P., Subba Rao, B. and Krishna Mohana Rao, G. 2010. Finite Element Modeling for Maximum Temperature in Friction Stir Welding and Its Validation. *International Journal of Advanced Manufacturing Technology*. 51: 925-933.

[7] Colegrove, P., Painter, M., Graham, D. and Miller, T. 2000. Three-dimensional Flow and Thermal Modeling of the Friction Stir Welding Process. *Proceedings of the Second International Symposium on Friction Stir Welding*. Gothenburg. Sweden.

[8] Tang, W., Guo, X., McClure, J. C., Murr, L. E. and Nunes, A. 1988. Heat Input and Distribution in Friction Stir Welding. *Journal of Materials Processing and Manufacturing Science*. 163-172.