Computational Studies on the Heat Transfer behaviour during the Friction Stir Welding of Aluminium Alloy

Vishwanath M M*[1], Dr. N Lakshamanaswamy[2], Meenakshi Devi Parre[3]

[1,2]Department of Mechanical Engineering, UVCE, Bangalore University, India.

[3]Department of Mechanical Engineering, IIT-RK Valley Campus, RGUKT, Andra Pradesh, India

[1]vishwanath.malnad@gmail.com, [2]nlsvce@yahoo.co.in, [3]pmeenakshidevi@rguktrkv.ac.in

***Abstract*—** FSW is the most significant metal joining process in a decade and is a "green" welding technology due to its energy efficient process, environment friendliness, and versatility over the materials. Friction Stir Method consumes less energy compared to the conventional welding methods, and also requires no cover gas or flux, thereby making the process environmentally friendly. FSW is emerging as a very effective solid-state joining technique. It combines frictional heating and stirring motion to soften and mix the interface between the two metal sheets, in order to produce fully consolidated welds. One of its main qualities lies in the possibility of joining materials previously difficult to weld, and to offer excellent mechanical properties. The study of heat transfer behaviour is very important to know the temperature distribution during the welding process as the temperature distribution within and around the stirred zone directly influences the microstructure of the welds, such as grain size, grain boundary character, coarsening and dissolution of precipitates, and resultant mechanical properties of the welds, it is important to obtain information about temperature distribution during FSW. In the present study, the friction stir welding model chosen for this task is the heat transfer model of Chao, Qi and Tang. Hence, our first task is to replicate their model using the same process conditions as given in their paper, using Finite Volume Method software package FLUENT. The developed grid was then imported into the FLUENT solver for executing the solution. In order to validate the replicated model, the output of the model was then correlated with the published experimental data from the papers. The temperature distribution through the work piece was observed by means of the temperature contour plots and time-temperature graphs. The results of the simulation are in good agreement with the experimental results, thus verifying the validity of the model developed.

***Keywords*—Friction Stir Welding, Aluminum alloy, Heat Source model, Transient analysis.**

I. INTRODUCTION

FSW is emerging as an effective method of solid state metal joining/welding technique and established very rapidly after its invention. It is generally used to weld high strength aluminium alloys especially applicable in aerospace industry and other alloys which are finding difficult by regular fusion welding methods. These aluminium alloys from 2XXX to 7XXX series are largely used to develop aerospace structures like wings, body, fuselage, fins and also find its applications in military transport vehicles, aircraft bodies, automotive parts, railway rolling, stock markets, ship building, etc. But, these aluminium alloys with higher strength are difficult for welding by regular fusion welding techniques due to hot cracking formation during welding. Therefore, joining in aerospace bodies in large amount is done by riveting resulting in increase of production cost and the process was very complex. The invention of FSW gives an opportunity for the manufacturing industry to alter earlier riveting approach for the production of lightweight structures with a simple and cost effective method.

In FSW the heat is generated as a result of friction between the tool shoulder and the work piece near the joint causes the welding action to takes place in the solid state without undergoing molten state. The amount of heat conducted to the work pieces to be joined play an important role in determining the quality of the welded joint and the stress developed. The amount of heat transferred to the tool determines the strength and life of the tool. The schematic representation of friction stir welding process is as shown in Fig 1.

A cylindrical non consumable tool designed with shoulder and the pin arrangement at a suitable rotating speed is inserted at the butting edges of two different work pieces to be joined and traversed in the direction of the joining line. This rotating tool creates the heat required for joining between the tool shoulder and the work piece due to friction and also causes the material movement to produce the joint.

Mechanical stability in the Friction Stir welding is one of the limitations for the tool to operate satisfactorily at a given range of operating temperature. The tool while operating should function as heating the work piece to required forging temperatures and also provide mechanical action for forging. Therefore tool should be capable of operating at high thermal loads without deformation and excessive wear and tear. The all classes of aluminium alloys are more suitable for the friction stir welding processes starting from 1xxx, 2xxx, 3xxx, 4xxx, 5xxx, 6xxx, 7xxx alloy series and also for newer alloys like Al-Li alloys with the similar and dissimilar combinations.

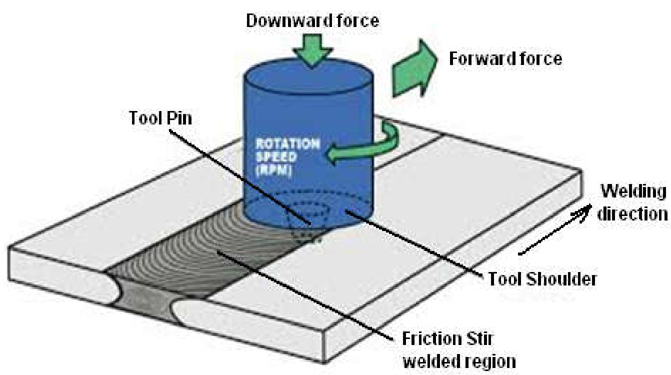

Fig 1. Friction Stir Welding Process

A model for temperature distribution with the experimental validations in the work piece was studied by Gloud and Feng [3]. In their studies they considered the heat is generated at the tool shoulder and analysed how heat is transferred to the work piece. The heat flux was applied at the top surface of the work piece with the model dimension as that of the tool shoulder. The heat flux is a function of rotating speed of the tool and the force applied on the work piece. For the better quality of the weld the heat input is made in such a way that the maximum temperature on the work piece should be 80-90% of the melting temperature of the work piece material as suggested by the Tang et. al. [2] and Cole grove et. al. [4] so that the defects are avoided. Chao and xi [5,6] made a finite element analysis with the experimental validations for the temperature distributions in the friction stir welding by developing a moving heat source model. The study also examines the residual stresses and distortion in welds. This study focuses on the heat transfer in both work piece as well as the tool as the heat is generated at the interface of the tool shoulder and workpiece. The heat is conducted in both sides. Very recently Song and Kovacevic[7] investigated on a heat source moving in a 3D co-ordinate system using finite difference method in order to minimize the difficulty of modelling the moving tool during FSW. Nandan et al [8] modelled a 3-Dimensional viscoplastic material flow. The temperature distribution curve is well validated with the experimentally obtained temperature data. Quasim M Doe's et.al. [9] Made a mathematical analysis and investigated that the thermal models with transient heat conduction is more accurete and similar with the experimental data as compared to the thermal-fluid model. Rajamanikan et.al. [10] Studied on Thermo-Mechanical model by Finite Element Method. They analysed that the thermal modelling are very useful for finding the temperature distribution within the work piece and nearer the tool shoulder in F S W processes. Hani et.al. [11] Developed a numerical model to study the heat transfer in friction stir welding of copper alloys. In this study different shapes of tool pins are used to sudy the heat transfer behaviour and the results are more reasonable with the temperature datas. Hamed Pashazadeh et.al. [12] Studied on the temperature distribution in the friction stir welding of copper plates. He developed a numerical model and concluded that the temperature profiles are not symmetrical around the line of joint and the temperature is always maximum behind the tool pin.

II. FORMULATION OF THE BOUNDARY CONDITIONS

The heat transfer in the welding process is the transient phenomenon; the heat is conducted due to the friction with some convective losses. The problem in the study is formulated as the standard boundary value problem and in the simulation it is solved by the inverse modelling approach where the data obtained by the formulation and simulation is compared with the experimentally measured data. The study makes a steady state approach for the tool and the transient analysis for the work piece. The assumption made by the Chao is briefly summarized as follows:

There is an application of downward force to the work piece for creating friction with the uniform pressure between the tool and the work piece.

There are no other forms of heat other than the heat generated by friction

The heat input at the centre is maximum and decreases linearly with the distance on the surface

The Fig 2 shows the different boundary conditions that exist during the friction stir welding. The surfaces of the work piece is exposed to atmospheric air hence convective boundary condition is applied at the rotating tool shoulder and the work piece contact face and Neumann boundary condition is applied between tool pin and the work piece. The faces that are exposed to the atmospheric air at the backing plate are also applied with convective boundary condition.

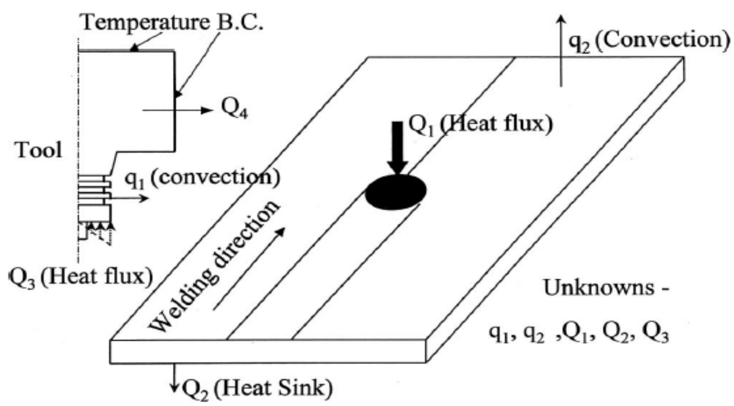

Fig 2. Shows the heat losses at different boundary in the tool and work piece.

The heat flow formulations are made for the tool as well as work piece. The heat generated by the friction is transferred to the work piece with some losses. The boundary value problem for the work piece is schematically shown in Fig. 2.

$$Q_1 = Q_2 + q_2 + Q \text{ ------------------ } (1)$$

Q_1 in the equation (1) is the heat developed due to friction at the interface of tool and work piece will be balancing energy equation with heat flux Q_2 is conducted to the backing plate from the work piece bottom surface, q_2 is the convective heat loss to the surroundings from the work piece, and Q is the heat absorbed by the work piece and the radiation losses are neglected.

The heat conducted to the rotating tool through the contact face and then to the machine head can be related as.

$$Q_3 = Q_4 + q_1 \text{ ------------ } (2)$$

Where, Q_3 is the heat transferred per unit area to tool contact surface created by friction with work piece, q_1 is the convective heat loss at the tool surface to the surroundings and Q_4 is the heat conducted to the machine head.

III. SIMULATION USING ANSYS-FLUENT

ANSYS is the established world leading provider of simulation software. In the today's world ANSYS is creating a robust product development platform considering the complex interactions of hardware, software, thermo mechanical, and thermo fluids based simulations in the manufacturing industries. ANSYS develops and supports the simulation engineer to predict the welding related simulations to the real world Environment. ANSYS provide clarity and insight for a researcher for the most complex design challenges through reliable, accurate and fast simulation procedures. Hence, in the present study Finite volume software ANSYS-FLUENT is used to replicate the same model developed by Chao, Qi and Tang [6] and meshing is made. The developed mesh is then called for ANSYS-FLUENT simulation. This simulation results were validated by comparing Experimental and FEM validations made in the Chao's results with that of the present simulation results. The main objective is to observe the results of temperature distribution (Time-Temperature) curves with the experimental results. As the ANSYS FLUENT is a leader in the simulation of heat transfer and fluid flow models, it permits the simulation engineer to take the results of temperature distribution. Hence the models are developed by Finite Volume Method on a collocated grid and are discretised into Finite control volume or cells sets. It also provides greater flexibility of meshing the developed tool and the work piece models. Since the FLUENT is written in C-Programming language, provides a flexibility to use. Once the meshing is developed it is called in FLUENT, then defining Thermo-Mechanical properties, applying boundary conditions and Analysis of the results etc can be carried out in solver. In the present work a User Defined Function (U D F) is developed using C-Programming Language to apply the transient conditions on the top surface of the work piece and is linked to FLUENT.

IV. RESULTS AND DISCUSSIONS

In the present investigation, numerical simulation of heat distribution behaviour during FSW of aluminium alloy were performed using the ANSYS tool to validate the simulation process the experimental data's and the FEM results obtained by the Chao, Qi and Tang were used and compared. Generally UDF's are developed to customize the ANSYS-FLUENT code to fit necessary modelling requirements. FLUENT user interface does not allow the transient boundary conditions to apply; a UDF is developed to perform the simulation by applying heat flux to work piece at the top face. Heat flux is incorporated as a customized condition that changes with respect to spatial co-ordinates and with varying time.

This User Defined Function is compiled by built in complier-interpreter at the runtime. Once User Defined Function is compiled and interpreted, this function is present in the graphical interface for further usage. This user function is adapted at each time step in the runtime. At the surface with in the loop, the weld position and the distance between the centres of each face is calculated. Temperature-dependent thermal material properties for the Aluminium alloy Al2195-T used in the modelling as listed in Table 1. The thermal conductivity and heat capacity at various temperatures are shown

TABLE I

THERMAL PROPERTIES OF THE AL2195-T USED FOR NUMERICAL SIMULATION

Temperature ^oC	**Thermal Conductivity** W/m ^oC	**Specific heat capacity** J/Kg ^oC
0	87	835
50	100	910
100	108	945
150	120	1000
200	130	1050
250	140	1085
300	145	1100

The simulations carried by using ANSYS –FLUENT are made to compare with that of heat transfer carried by the experimentation and finite element studies carried by the Chao et al. The following fig 3 shows the simulations of the chao's model and further the simulation results shown in the fig 4, fig 5 and fig 6 shows the results obtained from the present study for the temperature distribution and the heat transfer studies made by using the ANSYS simulation with the user defined functions at the different time step of 30, 60, 90, 120 and 210, 240 respectively. The results obtained for the thermocouple placed at a distance of 305mm exactly at the mid distance of the sample plates taken and the calibration is made at 12mm from the weld centre and 4mm from the top of the plate.

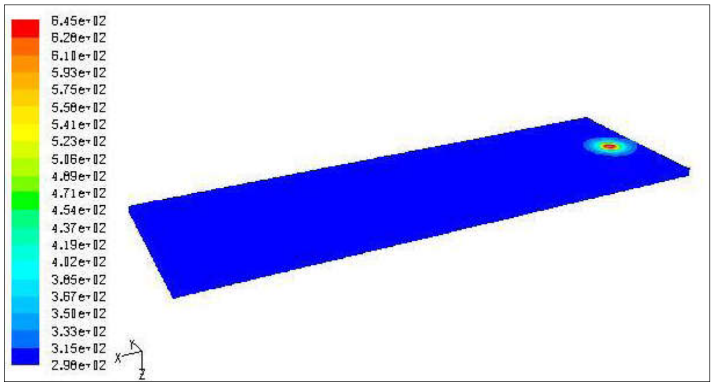

Fig 3. Chao's Temperature contours

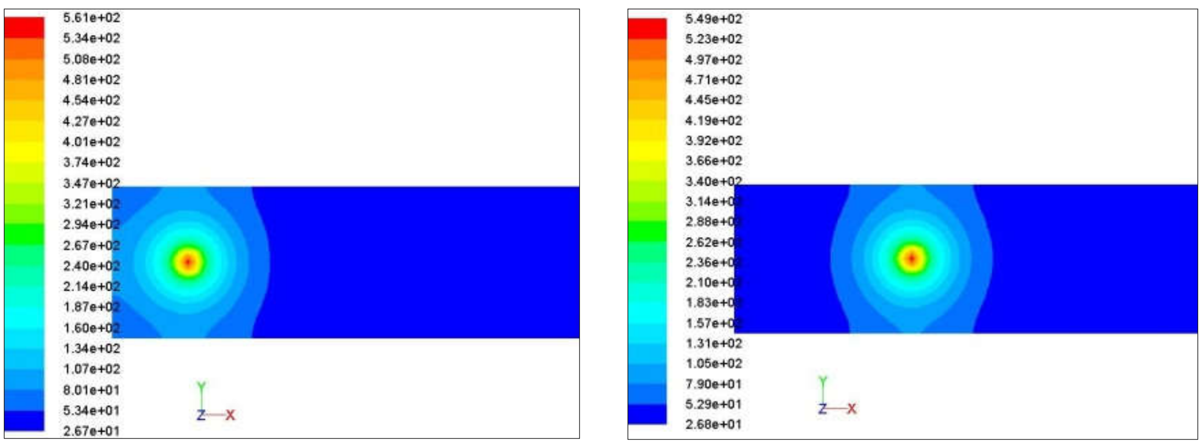

Fig 4. Present simulated Temperature contours at 30. 60 seconds.

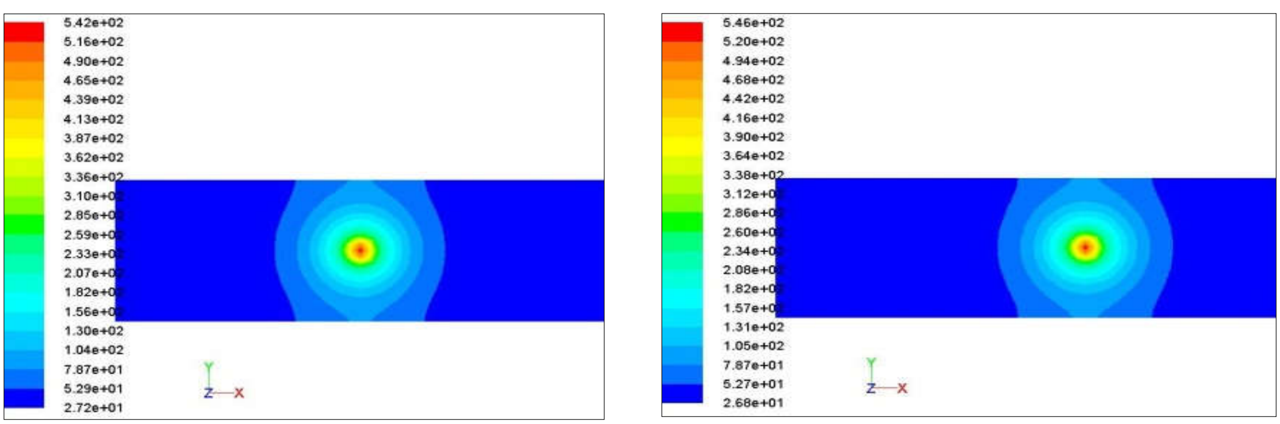

Fig 5. Present simulated Temperature contours at 90. 120 seconds.

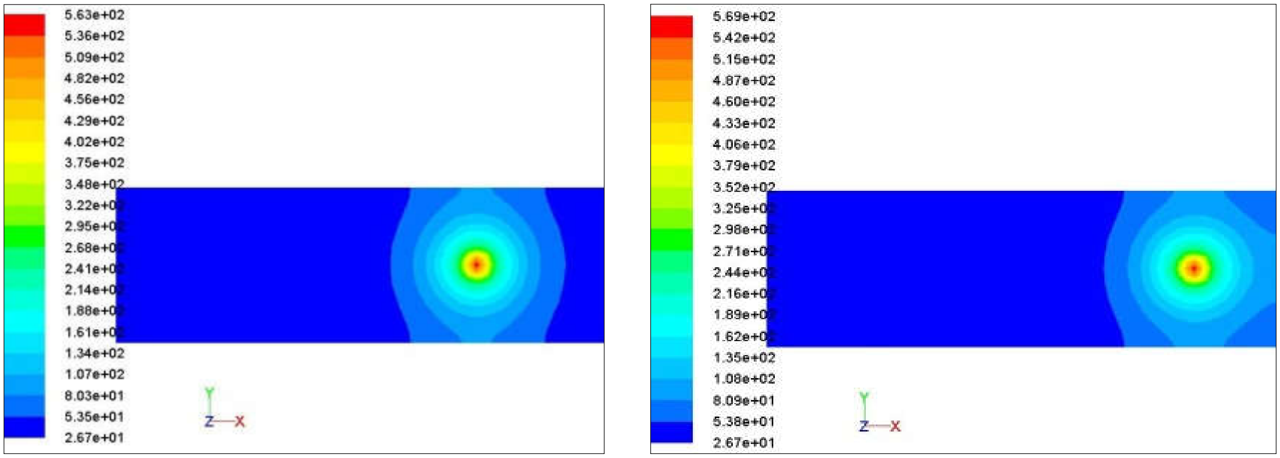

Fig 6. Present simulated Temperature contours at 210 and 240 seconds.

The results obtained from the simulations is plotted for the Temperature – time profile in order to validate the present study whether the study is carried in the proper directions the following Fig 7 shows the Time – Temperature curve for the experimentally validated Chao's result and the Time – Temperature curve obtained from the present work.

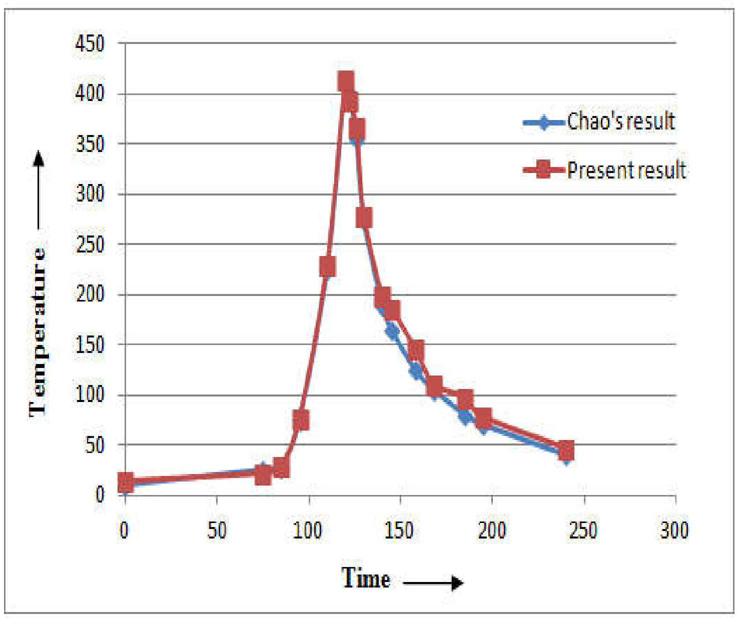

Fig 7. Time-Temperature comparison of Chao's work with present work.

It is observed from the above *fig 7*. The temperature distribution curve obtained from the simulation using ANSYS-FLUENT indicated in the red curve is compared against the results obtained by the experimental and FEM work made by Chao indicated in the blue curve, it is observed that the peak temperature obtained by the numerical simulation is 415°C is very nearer to the Chao's model which is obtained at a distance 305mm in the welding line, where the thermocouple is placed at the a distance of 5mm from the weld line and 4mm from the top surface. The temperature distribution curve obtained in the present numerical investigation is similar to the temperature distribution presented by Chao, and hence the numerical simulation carried in the present study by using Finite Volume Method software package ANSYS FLUENT is validated experimental results and the method can be further used for different numerical simulations of FSW.

V. CONCLUSION

The simulation is made to study the heat transfer behaviour Aluminium alloy using Finite Volume Method software package ANSYS FLUENT, the following conclusions were drawn.

1. The tool and the work pieces made are identical with the experimental details given by the model trend of Chao, Y.J., Qi, X. and Tang to validate the process.
2. The model is meshed to know the proper temperature profile is matching with the node the thermocouple are placed.
3. The time and temperature graph it is observed that the results obtained from the Chao, Y.J., Qi, X. and Tang experimentation are identical with the simulated temperature results hence validating the simulation process with the experimental.
4. The time temperature curve helps in giving the heat distribution patterns from the tool to the work piece to form a strong welded joint and also helps in studying the welding of different types of metals in a similar pattern by directly validating with the experimental time temperature curve which helps us to study the welding behaviour for different metals.

REFERENCES

[1] McClure, J. C., Tang, T., Murr, L. E., Guo, X., and Feng, Z., 1998, A Thermal Model of Friction Stir Welding. Trends in Welding Research, J. M. Vitek,et al., eds., Proceedings of the 5th International Conference, Pine Mountain, GA, June 1–5, pp. 590–595.

[2] Tang, W., Guo, X., McClure, J. C., Murr, L. E., Nunes, A., 1988, Heat Input and Temperature Distribution in Friction Stir Welding, Journal of Materials Processing and Manufacturing Science, 7-2, pp. 163–172.

[3] Gould, J., and Feng, Z., 1998, Heat Flow Model for Friction Stir Welding of Aluminum Alloys, Journal of Materials Processing & Manufacturing Science, 7-2, pp. 185–194.

[4] Colegrove, P., Painter, M., Graham, D., and Miller, T., 2000, 3 Dimensional Flow and Thermal Modeling of the Friction Stir Welding Process, Proceedings of the Second International Symposium on Friction Stir Welding, June 26–28, Gothenburg, Sweden.

[5] Chao, Y. J., and Qi, X., 1999, Heat Transfer and Thermo-Mechanical Analysis of Friction Stir Joining of AA6061-T6 Plates, Proceedings of the First International Symposium on Friction Stir Welding, June 14–16, Rockwell Science Center, Thousand Oaks, California.

[6] Yuh J. Chao, X. Qi and W. Tang, 2003, Heat Transfer in Friction Stir Welding—Experimental and Numerical Studies, Journal of Manufacturing Science And Engineering., Vol. 125, pp. 138–145

[7] M.Song and R.Kovacevic, 2003, Thermal modeling of friction stir welding in a moving co-ordinate system and its validation, International Journal of machine tools and manufacture 43, 605-615.

[8] R.Nandan, G.G.Roy and T.Debroy 2006, Numerical simulation of 3D heat transfer and plastic flow during friction stir welding, Metallurgical and materials transactions. Volume 37a, 1247-1259.

[9] Qasim M.Doos, J.J.Muhsin and D.R.Sarmad. 2008, Analysis of friction stir welds. Part I: Transient thermal simulation using moving heat source, First regional conference of Eng. Sci. NUCEJ Spatial issue vol.11, No.3, 429-437.

[10] N.Rajamanickam, V.Balusamy,P.R. Thyla and G.Hari Vignesh, 2009, Numerical simulation of thermal history and residual stresses in friction stir welding of Al 2014-T6, Journal of scientific and industrial research, Vol.68, 192-198.

[11] Hani Aziz Ameen, Ahmed Hadi Abood and Nabeel Shallal Thamer, 2013, Theoretical and experimental investigation of friction stir welding for copper alloy, AL-Qadisiya Journal For Engineering Sciences, Vol. 6.No 3, 332-351.

[12] Hamed Pashazadeh, Jamal Teimournezhad & Abolfazl Masoumi, 2014, Numerical investigation on the mechanical, thermal, metallurgical and material flow characteristics in friction stir welding of copper sheets with experimental verification, Materials & Design 55, 619–632