Janusz Pikula, Krzysztof Kwieciński, Grzegorz Porembski, Adam Pietras

FEM Simulation of the FSW Process of Heat Exchanger Components

Abstract: The paper presents results of the Fem simulation of Fsw process. The object of the research was a heat exchanger used for cooling of electrical components of propulsion systems. The tests enabled the obtainment of the field of temperature, stresses and displacements during the process and residual stresses and displacements of welded elements after cooling. Knowledge of the thermal conditions of the process, the stress and strain fields were used while designing clamps and pads of a welding stand for the welding of the heat exchanger components.

Keywords: Friction Stir Welding, Fsw, Finite Element Method, Fem Simulation

DOI: *10.17729/ebis.2016.3/3*

Introduction

Aluminium alloys are widely used in the aviation, automotive, transport, military, ship-building and other industries as the basic low density structural materials. Such a situation is the result of, among other things, increasingly strict requirements regarding strength – mass ratio and high corrosion resistance. In this case aluminium alloys are regarded as the best choice, particularly in high-volume production. This, in turn, triggers demand for high-quality joints. Presently, the most popular and efficient method for joining metals is welding. Due to their high thermal and electric conductivity, aluminium alloys pose many problems both in conventional fusion welding and in pressure welding. Welding process metallurgy makes some high-strength aluminium alloys simply unweldable due to the risk of hot crack formation and disadvantageous changes in the HAZ. A solution to those inconveniences is the Fsw (Friction Stir Welding) method developed at The Welding Institute in Cambridge in 1990, recognised as one of the greatest material joining-related achievements in the last two decades [1, 2].

The Fsw method consists in friction welding combined with weld material stirring leading to the solid-state joint of materials. The method was developed for joining high-strength and advanced aluminium alloys. Presently, Fsw is mainly used for joining long elements made of aluminium alloys as well as for making high-quality joints of copper, lead, magnesium, titanium, zinc, mild steel, selected stainless steels and nickel alloys. The method also enables making dissimilar welds joining various aluminium alloys, aluminium-copper, aluminium-steel and aluminium-magnesium [1–3].

mgr inż. Janusz Pikuła (MSc. Eng), mgr inż. Krzysztof Kwieciński (MSc. Eng) – Instytut Spawalnictwa, Testing of Materials Weldability and Welded Constructions Department; mgr inż Grzegorz Porembski (MSc. Eng); dr inż. Adam Pietras ((PhD (DSc) Eng.) – Instytut Spawalnictwa

Finite Element Method (FEM) is presently one of the most widely used methods of solving various engineering problems. FEM simulation of welding process is applied primarily for determining fields of temperature created while welding, stresses and strains of welded components as well as using this method it is possible analyse objects having complicated shapes and complex properties [4].

Instytut Spawalnictwa in Gliwice collaborating with the CEMAD foundry and ZBUS Ltd.developed and industrially implemented a modern technology for making complex bimetallic elements utilising the advanced FSW method. Within the tests conducted, it was possible to establish the principles of selecting welding conditions, process thermal conditions and the field of stress and strains. In the first stage of the project, the check valve ball was the object of the research [5]. While testing the FSW process of the ball halves, which were pressed against each other using special clamps, was simulated. The welding process consisted in the rotation of elements to be welded; the tool rotated only around its own axis. The results obtained thanks to the conducted FEM calculations were utilised while developing the designs of clamps for the welding station. The second stage was concerned with developing the FSW technology for joining components of a heat exchanger of electric components of power transmission systems. FEM simulation presented in this paper was also aimed at determining stresses and strains of the component being formed while welding and the results were used during the designing of clamps and pads of the welding station.

FSW Process Principle

The principle of the FSW process is presented in Figure 1. The preparatory stage involves pressing the edges of the elements to be joined (against each other) and fixing them rigidly in the tooling. A welding tool is located at the beginning of a welding path. The welding process starts with the rotating tool penetration in the interface of the edges of elements being joined and moving the tool along the joining line. The rotation of the tool penetrating the material causes friction and material plastic deformation thus generating heat necessary for heating and greater softening of the material enabling its flow around the tool probe (Fig. 2) and mechanical stirring [3]. Along with an increasing temperature the coefficient of friction between the tool and elements being joined decreases (Fig. 3) [7] and so does the amount of heat generated. The combination of these three quantities makes a temperature growth dependent on the value of temperature at a given moment. The process finishes with removing (lifting) the tool where the welding process has finished.

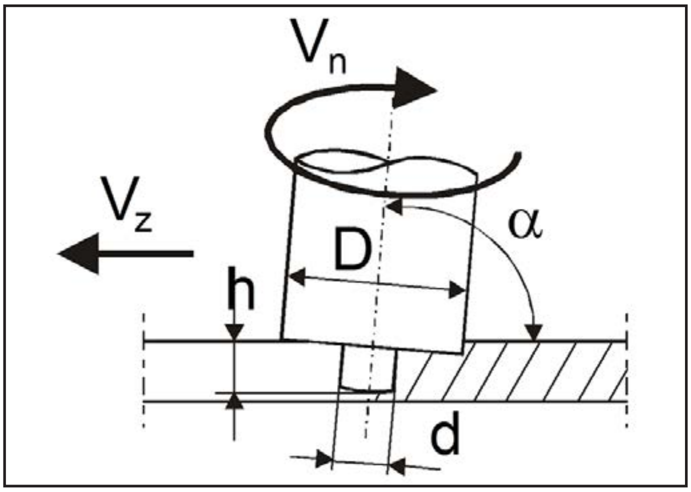


Fig. 1. Scheme of tool movement during FSW (v_n – rotation rate, v_z – tool linear velocity) [6]

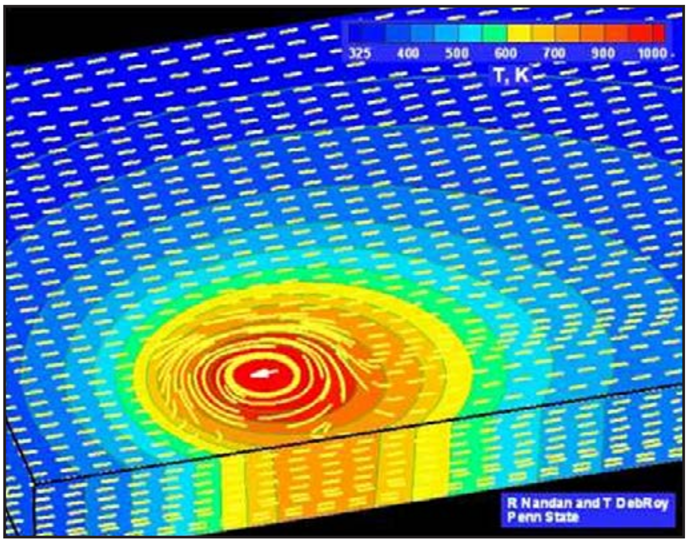


Fig. 2. Results of simulation of FSW welding process – temperature field and directions of the material flow [8]

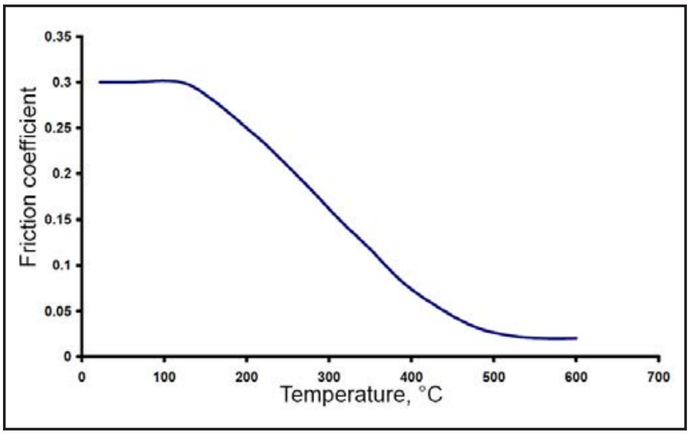


Fig. 3. Dependence of the friction coefficient on the 6061-T6 aluminium alloy temperature [7]

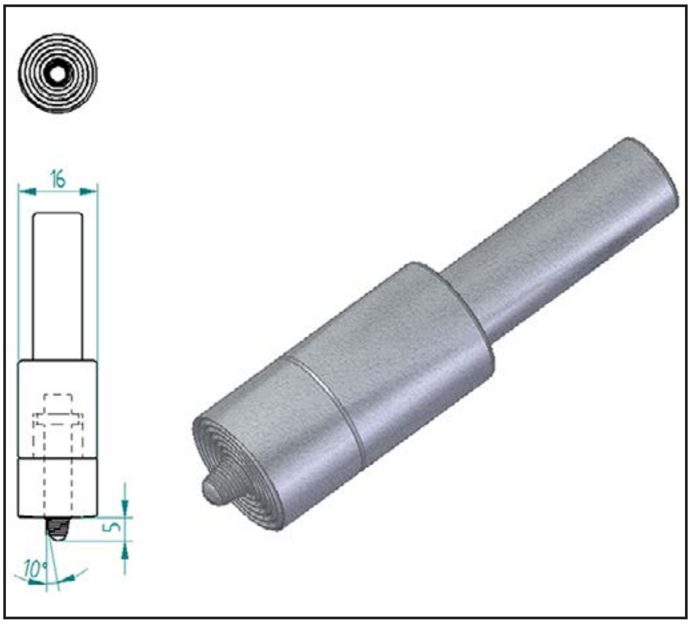
The literature contains a large number of publications describing the process of simulation of FSW joints using Finite Element Method [9-11], however these are the results of the testing of flat components having a straight welding line, and the numerical models were prepared for simulating processes in the specific welding conditions. The model presented in this article was prepared for the specific task – simulation of the welding process for joining elements of a heat exchanger (Fig. 4). The results of simulation (stress fields and strains) were used while designing clamps and pads on a welding station.

Fig. 4. Model Heat exchanger for tooling testing

Numerical Model

This article presents a model and simulation results related to the friction stir welding of a heat exchanger of electrical components of a propulsion system made of aluminium alloy. The welding tool used in the simulation was that consisting of a conical probe 5 mm in length and 3.5-5 mm in diameter as well as a shoulder having a diameter of 16 mm (Fig. 5). The calculations performed enabled the determination of the field of temperature as well as that of stress and strains during the welding process.

In addition, the tests resulted in determining the values of stress occurring after welding and strain after welding and cooling to the ambient temperature. The numerical model and simulation were developed using Ansys Mechanical Apdl 14.5 software.



Rys. 5. Shape and dimension of the tool used in the real FSW process

In the real process the heat exchanger was pressed using various clamps and pads in different configurations. An example of the configuration is shown in Figure 6. The welding process consisted in the rotation of the tool around its own axis and moving it along the welding path, whereas the component being welded remains fixed.

Fig. 6. Station for FSW of the components of the heat exchanger

The process of the developing FEM model of the heat exchanger components was conducted analogically to that of FEM in the case of the check valve ball [5]. The first stage involved the development of a geometrical model representing the real geometry of elements. Afterwards, the model underwent discretization, i.e. its area was divided into finite elements (Fig. 7). The size of the finite elements significantly affects the correctness of calculations. The smaller the elements used, the more accurate the geometry representation. This is of particular importance in cases of curves of the welding path (Fig. 8) and a circle (the shape of the heat source). The length of the finite elements was limited to 5.8 mm.

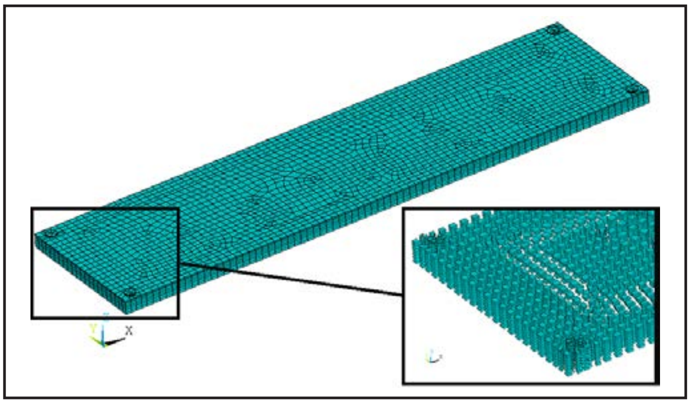


Fig. 7. Finite elements mesh

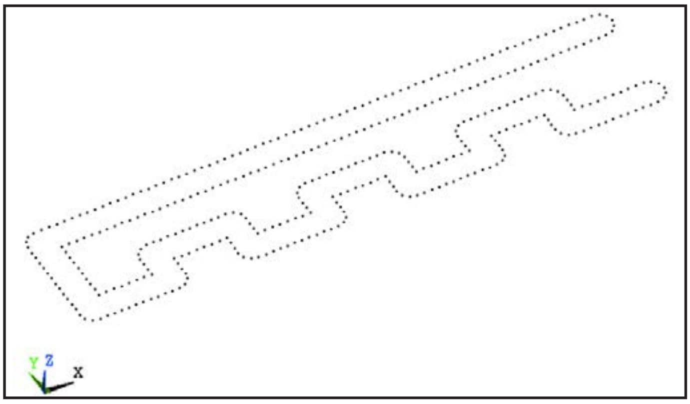


Fig. 8. FSW process path in FEM model

The analysis involved the use of SOLID226 three-dimensional thermo-mechanical solid elements. Such elements are used in thermo-structural analyses in the elastic-plastic range, and their application made it possible to obtain correct results in the FSW process of joining check valve ball elements [5].

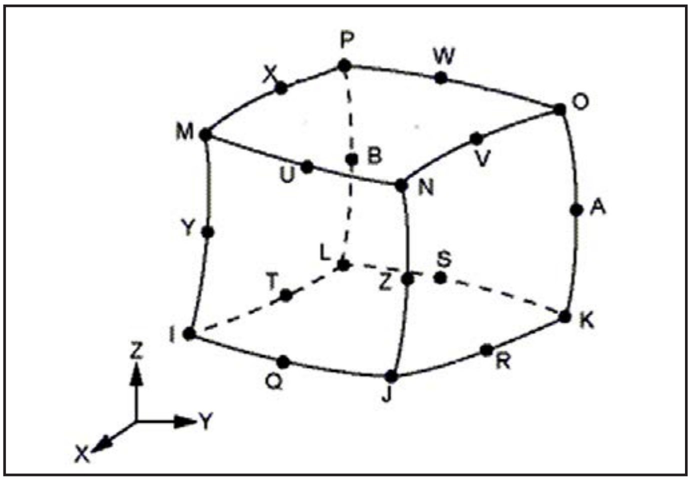


Fig. 9. SOLID226 three-dimensional thermo-mechanical solid element

Table 1 presents the welding process parameters used in the analysis. The value of the force in the direction of welding was determined experimentally during a real process involving the use of a Lowstir head intended for measuring friction moments, pressure force and force in the direction of FSW.

The increase in temperature, responsible for material softening, results, among other things, from the forces of friction between the tool and elements being welded. The amount of momentarily generated heat is affected by the rate of rotation and the turning moment. In the simulation, the heat generated due to the friction of the tool and elements being welded was represented as the heat flux applied to the external surface of heat exchanger surface at the point of contact with the tool. The model was simplified by omitting the heat emitted due to the plastic strain (material flow). It was assumed that the power transmitted by the tool was entirely transformed into the thermal power determined from the following dependence (1):

$$P = 2\,\pi\,f\,M \tag{1}$$

where
- P – power,
- f – rate of rotation, rpm,
- M – turning moment, Nm

The value of heat flux calculated amounted to 2229.4 W. The process of welding along the path was simulated as a heat source having a geometry of a circle moving at a rate of 280 mm/min. The heat source diameter on the surface amounted to 16 mm, which corresponded to the shoulder diameter of the real tool. The simulation took into consideration the effect of the force in the direction of welding exerted by the tool on the elements being welded. The path was determined along the welding line.

The simulation involved the use of material data for 6061-T6 grade aluminium alloy in the function of temperature such as heat conductivity, heat capacity, density, the Young module, yield point and thermal expansion (Table 2). Such an approach enabled the obtainment of more accurate results than those obtained during analyses, taking into consideration constant values of material data.

The simulation involved welding at the ambient temperature taking into consideration the reactions of the clamps on the heat exchanger elements and pads (Fig. 10), which are present in the real process. The numerical model assumed that there was no heat exchanger element Z-axis displacement at the pads effect point (nodes were void of displacement freedom degree). Additionally, nodes which belong to the field of clamps reaction projected to the lower surface, were void of the possibility to move in the X and Y axes. Clamps pressed the welded elements with force of 2000 N, which caused displacement of nodes of upper surface of clamps towards axis Z (Fig. 11).

Table 1. Parameters of simulated welding process

Welding rate	280 mm/min
Rate of rotation	710 rev/min
Turning moment	30 Nm
Fz (force in the direction of welding)	0.72 kN

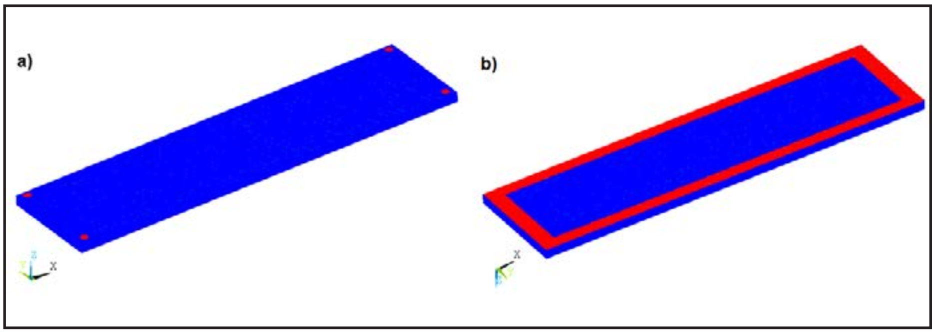


Fig. 10. Reactions of clamps on elements being welded (marked red) a) top view - clamps, b) bottom view - pads

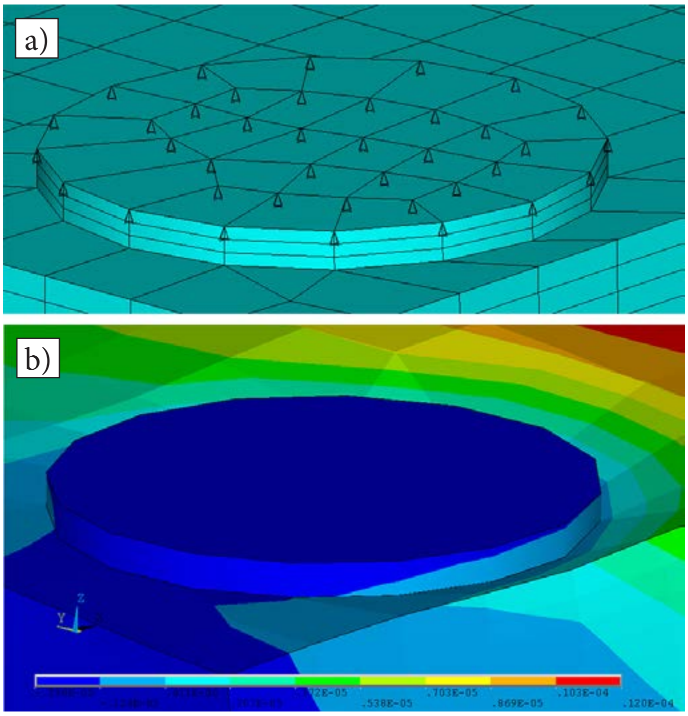


Fig. 11. Nodes of clamps displaced towards axis Z, a) elements, b) value of displacement, mm

Table 2. Material data of 6061-T6 aluminium alloy [7]

Temperature	°C	37.8	93.3	148.9	204.4	260.0	315.6	371.1	426.7
Heat conductivity	W/m°C	162.0	177.0	184.0	192.0	201.0	207.0	217.0	223.0
Specific heat	J/kg°C	945.0	978.0	1004.0	1028.0	1052.0	1078.0	1104.0	1133.0
Density	kg/m^3	2685.0	2685.0	2667.0	2657.0	2657.0	2630.0	2630.0	2602.0
Young module	GPa	68.5	66.2	63.1	59.2	54.0	47.5	40.3	31.7
Yield point	MPa	274.4	264.6	248.2	218.6	159.7	66.2	34.5	17.9
Thermal expansion	10^{-6}/°C	23.5	24.6	25.7	26.6	27.6	28.5	29.6	30.7

The model assumes that welding process is conducted at room temperature. In the point of projection of the contact with pads and clamps, heat conduction occurs, while in other areas occurs the convection. On the groove surface (internal) the heat exchange with ambient environment was not assumed due to the fact that volumetric heat capacity is so small and can be neglected.

Simulation results

The temperature of the elements welded during the simulated FSW process was about 500°C (Fig. 12 a). The course of temperature in the function of time at the point located on the welding process path indicates a rapid growth of temperature under the tool (Fig. 13 b). The greatest stress during the FSW process conducted on the heat exchanger elements was present at the clamps and pads contact points and along the welding path where parts were welded (Fig. 13 a). The value of residual stress and strain were the highest along the welding path and at the welding process completion – i.e. tool removal – point (Fig. 13 b).

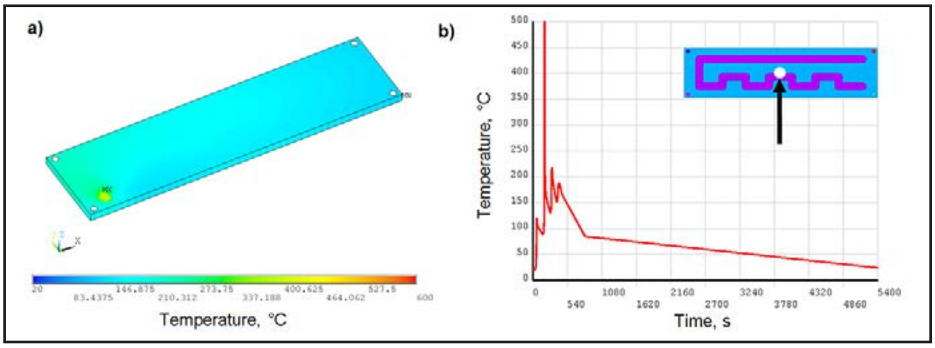


Fig. 12. Temperature analysis a) temperature field after 457 seconds following the welding process start, b) temperature changes in the function of time at point indicated by the arrow

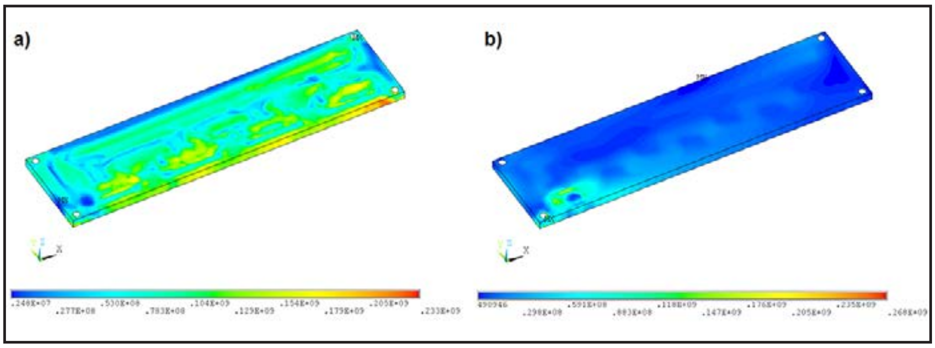


Fig. 13. Values of a) reduced stress (according to the Huber - Mises – Hencky hypothesis) 457 seconds following the welding process start, b) residual stress (5400 seconds following the welding process completion)

Summary

The developed FSW FEM numerical model enabled simulating and obtaining the field of temperature, stress and displacement. The FSW mathematical model presented contained certain simplifications. Nonetheless, it was possible to observe the convergence of the calculated temperature field with experimental temperature measurement results of the real friction stir welding of the heat exchanger elements made using a thermovison camera. The correctness of calculated stress during the process and of the residual stress as well as displacement after element cooling to the ambient temperature were not verified.

Knowledge of process thermal conditions, stress fields and displacement was utilised while developing the designs of clamps and pads for the station used for welding heat exchanger elements of electric components of propulsion systems.

Concluding remarks

1. In order to presents the tool effect in the FSW process it is possible to use a moving heat source having a circle geometry of a diameter equal to a shoulder diameter. The results obtained were in conformity with the temperature measurement using thermovision camera.

2. A welding thermal cycle affects residual stress and post-weld element strain.

3. The greatest stresses during the FSW process are present before the welding tool and in the area of the reaction of clamps and pads to elements being welded.

4. The values of residual stress were the highest along the welding path and in the welding completion area – the site at which the tool was removed.

The article contains selected results of tests conducted within the confines of Project no. Innotech/28/150092/2012 financed by the National Research and Development Centre.

References

[1] Krasnowski K., Dymek S.: *A comparative analysis of the impact of tool design to fatigue behavior of single-sided and double-sided welded butt joints of En aw 6082-t6 alloy*. Journal of Materials Engineering and Performance, 2013, no. 12, , pp. 3818–3824. http://dx.doi.org/10.1007/s11665-013-0711-z

[2] Kalemba I.: *Mikrostruktura i własności połączeń stopów aluminium wykonanych metodą zgrzewania tarciowego z mieszaniem materiału spoiny*. Ph.D. Dissertation, Akademia Górniczo-Hutnicza, Kraków, 2010.

[3] Krasnowski K.: *Wpływ procesu Fsw na mikrostrukturę i wytrzymałość zmęczeniową złączy stopu aluminium 6082*. Ph.D. Dissertation, Akademia Górniczo-Hutnicza, Kraków, 2012.

[4] Rakowski G., Kacprzyk Z.: *Metoda elementów skończonych w mechanice konstrukcji*. Oficyna Wydawnicza Politechniki Warszawskiej, Warszawa, 1993.

[5] Pikuła J., Kwieciński K., Porembski G., Pietras A.: *Fem Simulation of Check Valve Ball Fsw Process*. Biuletyn Instytutu Spawalnictwa, 2014, nr 6, s. 23-29 http://bulletin.is.gliwice.pl/index.php?go=current&ebis=2014_06_03

[6] Pietras A., Zadroga L., Łomozik M.: *Charakterystyka zgrzeiny utworzonej metodą zgrzewania z mieszaniem materiału zgrzeiny (Fsw)*. Biuletyn Instytutu Spawalnictwa, 2003, nr 3, s. 34–38.

[7] Awang M.: *Simulation of friction stir spot welding (Fssw) process: study of friction phenomenon*. West Virginia University, 2007.

[8] Nandan R., Roy G. G., Lienert T. J., Debroy T.: *Three-dimensional heat and material flow during friction stir welding of mild steel*. Acta Materialia, 2007, vol. 55, no. 3, pp. 883–895. http://dx.doi.org/10.1016/j.actamat.2006.09.009

[9] Buffa G., Hua J., Shivpuri R., Fratini L.: *A continuum based Fem model for friction stir welding - model development*. Materials Science and Engineering: A, 2006, vol. 419, no. 1–2, pp. 389–396. http://dx.doi.org/10.1016/j.msea.2005.09.040

[10] Assidi M., Fourment L., Guerdoux S., Nelson T.: *Friction model for friction stir welding process simulation: Calibrations from welding experiments*. International Journal of Machine Tools and Manufacture, 2010, vol. 50, no. 2, pp. 143–155. http://dx.doi.org/10.1016/j.ijmachtools.2009.11.008

[11] Ulysse P.: *Three-dimensional modeling of the friction stir-welding process*. International Journal of Machine Tools and Manufacture, 2002, vol. 42, no. 14, pp. 1549–1557. http://dx.doi.org/10.1016/s0890-6955(02)00114-1