Janusz Pikuła, Krzysztof Kwieciński, Grzegorz Porembski, Adam Pietras

FEM Simulation of Check Valve Ball FSW Process

Abstract: The article presents results of the FEM simulation involving friction stir welding (FSW) of check valve ball elements. The tests involved the obtainment of temperature fields, information about stresses and displacements during the process and about residual stresses and strains following the cooling of elements welded. The knowledge obtained concerning process thermal conditions and that concerning the field of stresses and strains was used to design clamps of a station intended for welding check valve ball elements.

Keywords: welding, FSW, simulation, FEM, numerical model

Introduction

Aluminium alloys are widely used as the basic low density structural materials in the aviation, automotive, transport, military, ship-building and other industries. Such a situation is the result of, among other things, increasingly strict requirements concerning a strength – mass ratio and high corrosion resistance. For the applications enumerated above, 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 hot crack formation risk 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].

Instytut Spawalnictwa in Gliwice collaborating with the CEMAD foundry and ZBUS Sp. z o.o. (Ltd) developed and industrially implemented a modern technology for making complex bimetallic elements utilising the FSW method. Within the tests conducted it was possible to know the principles of selecting welding conditions,

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, Department of Resistance and Friction Welding and Environmental Engineering

process thermal conditions as well as the field of stresses and strains.

The fields of temperature, stresses and displacement of objects having complicated shapes and complex properties can be determined using the Finite Element Method (FEM) [4]. In order to know thermal conditions and fields of stresses and strains during friction stir welding of a selected element it was necessary to create an FEM numerical model of this process and carry out a related simulation afterwards. The FEM results enabled the development of a proper tooling design as well as made it possible to properly design welding stations built on the basis of milling machines.

FSW Process Principle

The principle of the FSW process is presented in Figure 1. The preparatory stage involves pressing the edges of elements to be joined (against each other) and fixing them rigidly in the tooling. A welding device 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 softening 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) [6] 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.

FEM Used for Designing FSW Joints

There are many reference publications describing FEM-based modelling of FSW joints. The work [8] presents a model of the FSW process (Fig. 4) used by the authors to examine the effect of tool probe geometry and increased welding rate on the flow of material and grain size

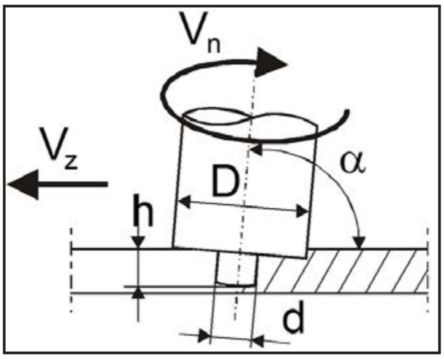

Fig. 1. Scheme of tool movement during FSW (v_n – rotation rate, v_z – tool linear velocity) [5]

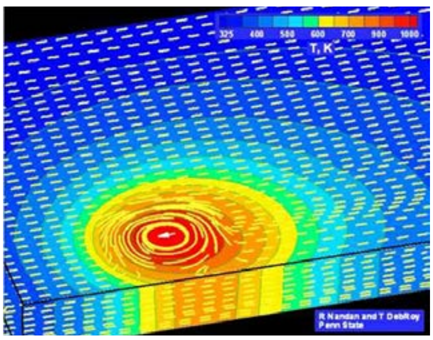

Fig. 2. FSW simulation results – the field of temperature and material flow directions [7]

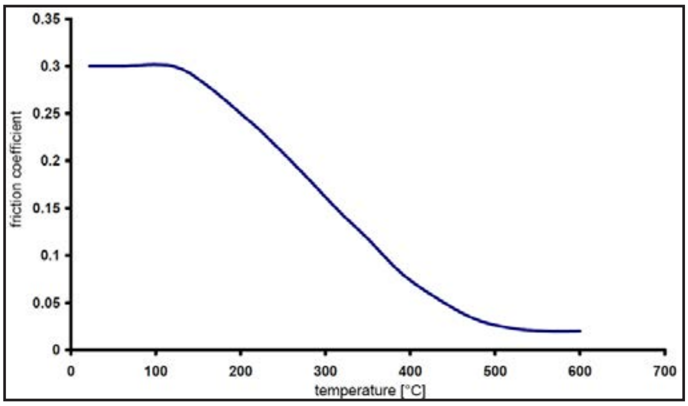

Fig. 3. Dependence of the friction coefficient in the 6061-T6 aluminium alloy temperature [6]

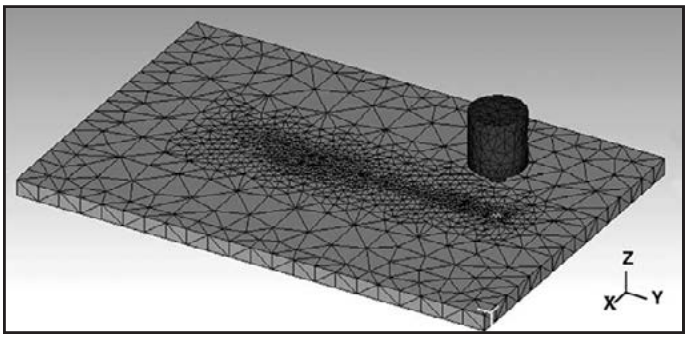

Fig. 4. FSW process FEM model – start of simulation [8]

in welded joints. The model proposed underwent calibration using results obtained during real FSW process measurements. Particular attention was given to the coefficient of friction between the tool and the material being welded in order to precisely represent welding process-accompanying forces and the field of temperature. The value of temperature in the function of time proved consistent at a selected point (Fig. 5).

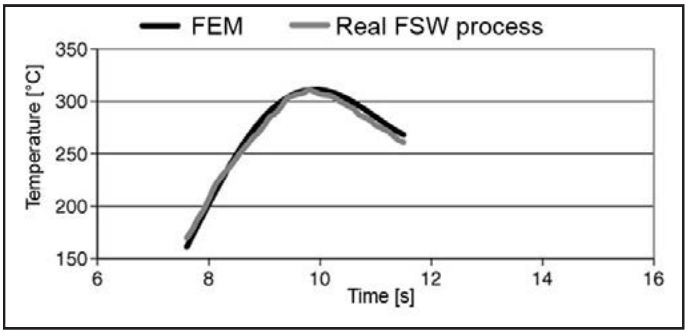

Fig. 5. Value of temperature in the function of time at a point 8 mm away from the joint axis, rotation rate 1040 rev./min, welding rate 100 mm/min [8]

The primary objective of the work whose results are presented in the article [9] was to develop an accurate model of friction during FSW of 6061 grade aluminium alloy. The model was developed using Forge3 F.E. software and involved both the element welded (plate) and the tool. Due to time-consuming calculations, the dimensions of the plate modelled were smaller than those of the plate used in the real FSW process. The tests involved two friction models verified experimentally. The Norton-Hoff model enabled determining that the values of welding-accompanying forces were consistent with those characterising the real process, yet the consistence of the numerically determined temperature field in the tool area was not ascertained. The numerical simulation revealed that the highest temperature was near the shoulder, whereas the measurements made using thermocouples revealed that the highest temperature during FSW was on the probe surface. Using the Coulomb friction model it was determined that the highest tool temperature was on the probe surface (Fig. 6) and that the temperature value was consistent with that obtained during experimentation. However, the temperature measured in the other tool areas was higher than that measured during the friction stir welding process.

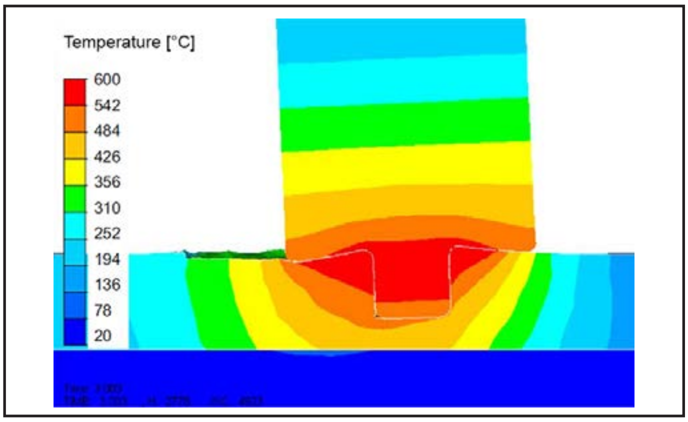

Fig. 6. Temperature field according to FSW process simulation [9]

The article [10] presents the simulation results concerning the friction stir welding of AA 7050-T7451 grade aluminium alloy. The field of temperature on elements joined and that on the tool (Fig. 7) indicate that the highest temperature was present on the shoulder surface. However, it was not possible to obtain the full conformity of FEM results with those obtained experimentally. The temperature values calculated were higher than the temperature values measured during the material welding process.

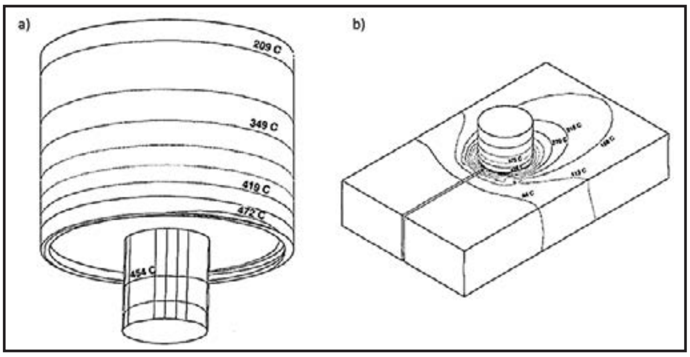

Fig. 7. Temperature field of a) tool, b) elements joined [10]

The reference publications analysis presented provides the results of tests conducted on flat elements. The numerical models were prepared for simulating processes in specified welding conditions. The model presented in this article was prepared for a specific task, i.e. the simulation of the FSW process conducted on a check valve ball. The simulation results (fields of stresses and strains) were used to design welding station clamps.

Numerical Model

This article presents a model and simulation results related to the friction stir welding of two check valve ball halves made of aluminium alloy; the ball half diameter was 110 mm and the wall thickness amounted to 9 mm. A 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 of a shoulder having a diameter of 16 mm (Fig. 8). The calculations performed enabled the determination of the field of temperature as well as that of stresses and strains during the FSW process. In addition, the tests resulted in determining the values of residual stresses and strains after welding and cooling to the ambient temperature. The numerical model and simulation were developed using Ansys Mechanical APDL 14.5 software.

In the real process the ball halves were pressed against each other using special clamps (Fig. 9). The welding process consisted in the rotation of elements to be welded; the tool rotated only around its own axis.

The first stage involved the development of a geometrical model representing the real ball geometry. Afterwards, the model underwent discretisation, i.e. its area was divided into finite elements (Fig. 10). The size of 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 spheres (the shape of the element welded) and circles (the shape of the heat source). The length of finite elements was limited to 6 mm, whereas along the welding path the mesh was concentrated with a length limited to 2 mm. The analysis involved the use of Solid226 three-dimensional thermo-mechanical solid elements (Fig. 11).

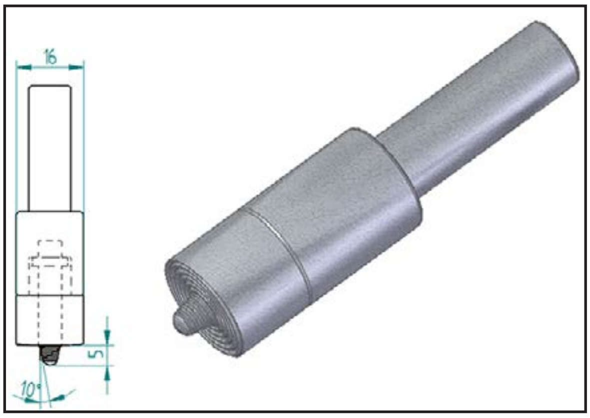

Fig. 8. Shape and dimension of the tool used in the real FSW process

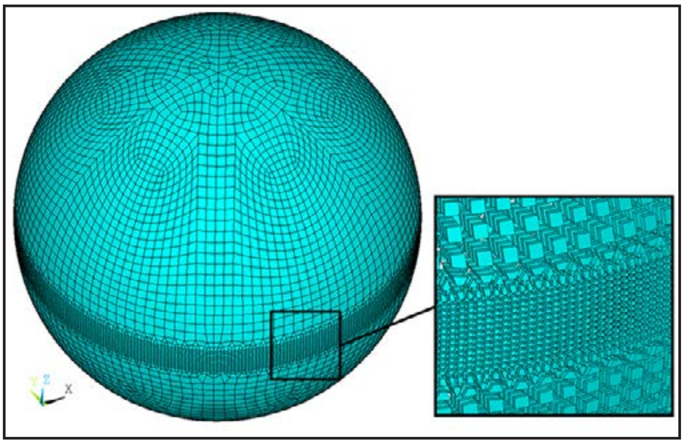

Fig. 10. Finite element mesh

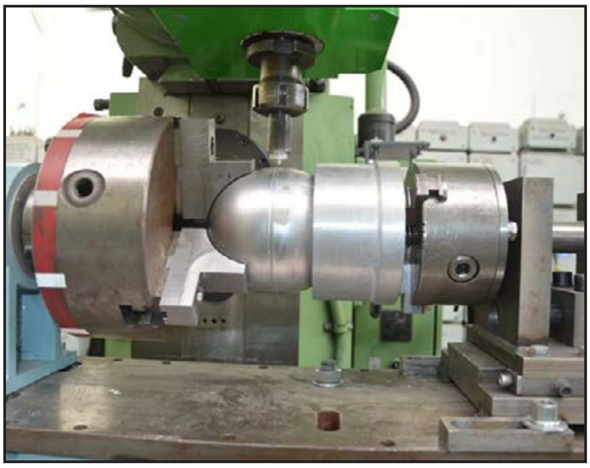

Fig. 9. Station for FSW of check valve ball elements

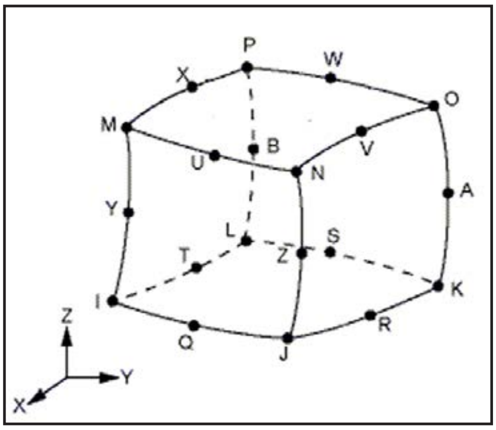

Fig. 11. SOLID226 three-dimensional thermo-mechanical solid element

Such elements are used in thermo-structural analyses in the elastic-plastic range.

Table 1 presents the welding process parameters used in the analysis. The value of the force in the direction of welding were 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 ball 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 is 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,
- M- turning moment.

The value of heat flux calculated amounted to 2229.4 W. The process of welding along the path was simulated as the heat source having the 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.

Table 1. Simulated welding process parameters

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

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 ball elements (Fig. 12) (reactions present

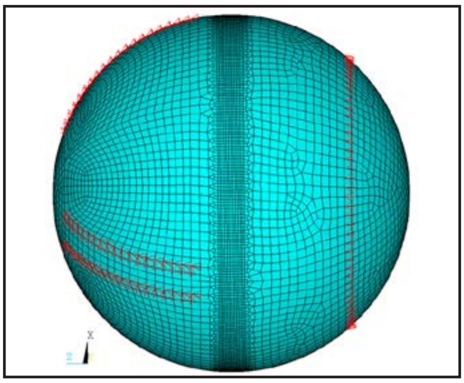

Fig. 12. Reactions of clams on elements being welded (marked red)

Table 2. Material data od 6061-T6 aluminium alloy [6]

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

in the real process). The numerical model assumed that there was not any ball element displacement at the clamp effect point (nodes were void of displacement freedom degrees).

Simulation results

The temperature of the ball elements welded during the simulated FSW process did not exceed 325°C (Fig. 13). The tool temperature during the welding process amounted to 300-325°C. 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. 14).

The greatest stresses during the FSW process conducted on the check valve ball elements were present at the clamp contact points and under the tool (Fig. 15). The value of residual stresses and strains were the highest along the welding path and at the welding process completion (i.e. tool removal) point (Figs. 16, 17).

Summary

The FSW FEM numerical model developed 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 the experimental temperature measurement results of the real friction stir welding of the check valve ball elements obtained using the thermographic camera. The correctness of calculated stress during the process and of the residual stresses as well as strains after element cooling to the ambient temperature were not verified.

Knowledge of process thermal conditions, stress fields and strains was utilised while developing the designs of clamps for the station used for welding check valve ball elements.

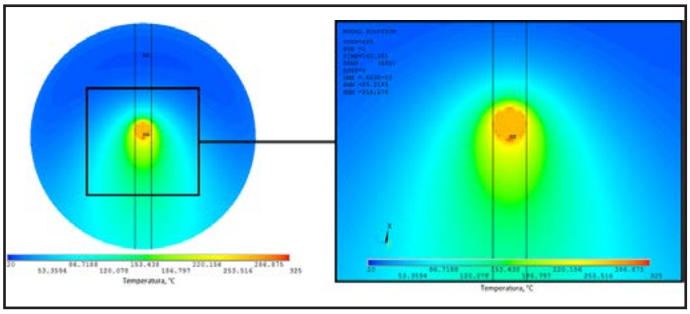

Fig. 13. Temperature field after 110 seconds folowing the welding process start

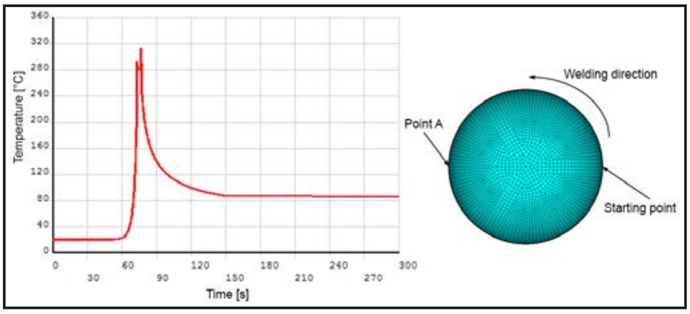

Fig. 14. Temperature changes in the function of time at point A

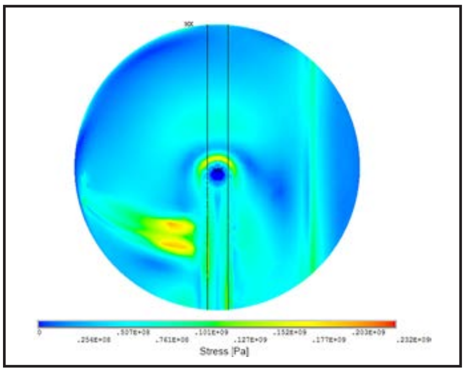

Fig. 15. Values of reduced stressed (according to the Huber - Mises – Hencky hypothesis) 110 seconds folowing the welding process start

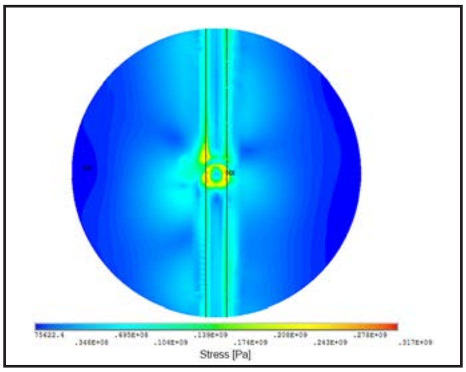

Fig. 16. Residual stress according to the Huber - Mises – Hencky hypothesis (5400 seconds following the welding process completion)

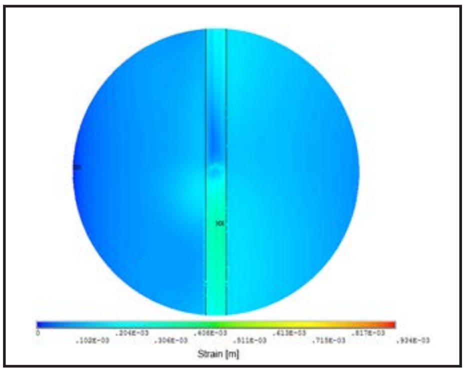

Fig. 17. Strain (5400 seconds folowing the welding process completion)

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 temperature values obtained were consistent with those obtained using a thermographic camera.

2. A welding thermal cycle affects residual stresses and post-weld element strains.

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

4. The values of residual stresses 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.

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

[3] Krasnowski K.: Wpływ procesu FSW na mikrostrukturę i wytrzymałość zmęczeniową złączy stopu aluminium 6082. Rozprawa doktorska, Akademia Górniczo-Hutnicza, Kraków, 2012.

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

[5] Pietras A., Zadroga L., Łomozik M.: Charakterystyka zgrzeiny utworzonej metodą zgrzewania z mieszaniem materiału zgrzeiny (FSW). Biuletyn Instytutu Spawalnictwa w Gliwicach, 2003, no. 3, pp. 34–38.

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

[7] 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, no. 3, pp. 883-895.

[8] 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, nos. 1-2, pp. 389–396.

[9] 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, no. 2, pp. 143–155.

[10] Ulysse P.: Three-dimensional modeling of the friction stir-welding process. International Journal of Machine Tools and Manufacture, 2002, no. 14, pp. 1549–1557.