Numerical simulation of Friction Stir Welding of three dissimilar aluminium alloys

A Boșneag¹*, M A Constantin¹ and E L Nitu²

¹University Politehnica of Bucharest, Machine Building Technology Department, Splaiul Independenței Street No. 313, Romania
²University of Pitesti, Manufacturing and Industrial Management Department, Târgul din Vale Street No.1, Romania

Email: ana.bosneag89@yahoo.com

Abstract. FSW process is increasingly required in industrial environment. This innovative process has a lot of advantages: can be performed without melting of material, uses a non-consumable tool, performed good mechanical properties, can weld dissimilar materials and have a low environmental influence. On the other hand, FSW process has several disadvantages: an excessive tool wear, weld speeds slower, expensive equipment and limitations for high melting temperature materials. To achieve the expected results during industrial applications, welding process behaviour must be properly understood. The Finite Element Simulation is a technique which would help to understand the process variation during the welding and the result could be used to notice the effect of process parameters on weld quality. In this study an attempt is made to develop a FE model using ABAQUS V6.13/Explicit with help of CEL formulation, Johnson-Cook material law and Coulomb’s Law of friction. The model will obtain a perspective on circumstance of processing before carrying out the welding process, would predict the effect of input process parameters on outputs parameters like: temperature. The simulation results are used to obtain the optimized process parameters. Numerical model validation is performed by comparing the value of temperatures from FE simulation with the experimentally recorded temperatures.

1. Introduction

Friction Stir Welding process was invented by Wayne Thomas at The Welding Institute, Cambridge from U.K. at the end of 1991 year. Until now was considered a new and innovative welding process [1]. At this moment, this process is more and more used to weld similar and dissimilar alloys in industry such as: transportation where more than half of industrial application is involved in automotive, aeronautics (especially this three studied aluminium alloys), railway and aerospace. The other half is split in: electronic industry, metal working, machinery and research [2, 3].

This welding processes are performed in three different steps: in 1st case, the rotating cylindrical tool makes a plunging in the joined plates; after that, it starts with a moving along the welding seam, generates heat and plasticize the material to create a plastically-mechanical joint and, at the end, the tool is extracted from the welded plates [4]. To perform this weld in good conditions and with a quality welding seams, it is important to control the temperature distribution and the material flow in the process time. These characteristics have direct influence in formation of the welding joint [5].
Friction Stir Welding process have a lot of advantages, but, in the same time can have some disadvantages. One of the major disadvantages can be considerate the cost of the process and the cost of the experiments because in this time are spend resources like: materials, energy, tool wear and machine wear. This disadvantage can be minimised using the finite element modelling because it is a helpful tool to analyse and understanding the effect of the process parameter on the proprieties of the welding seams and to the welding process [6]. In the same time, a good finite element modelling can help to achieve information regarding material flow and temperature distribution in the process time. This is important fact to optimise input parameters and to avoid occurrence of defects in welding seams.

For Friction Stir Welding process it is not easy to perform a numerical simulation because it is a complex process with large deformation and heat variation.

In recent years, a lot of researches had a subject the study of temperature evolution during the Friction Stir Welding process using a wide range of finite element software. In this context is necessary to model this process as a fully thermomechanical analysis. The greater potential to achieve the actual requirement is performed by CEL method. Through CEL method it is possible to predict defects in welding seams for given process conditions, this can be helpful to select the appropriate process parameters prior to performing welding process. The CEL method helps in reducing simulation time, making finite element simulation economical [7]. To simulate the Friction Stir Welding process between aluminium alloys, the most studies are developed to using CEL method [8].

In this paperwork, it is studied the finite element model of the FSW process for three dissimilar aluminium alloys welded in overlap position, these three alloys are: AA7075, AA2024 and AA6061. For validation of the numerical model, it was compared with experimental study performed in the identical conditions.

2. Experimental procedures
The Friction Stir Welding process was performed by using a dedicated welding machine. On this machine was mounted a cylindrical threaded M6 tool presented in figure 1. All three plates for welded, AA7075, AA2024 and AA6061 was fixed on the machine table with a special device.

The temperature generate in process step was recorded using a thermographic infrared camera, type FLIR A40M. This camera has a range between -40 [°C] and + 2000 [°C] and an accuracy of ±2 [°C]. Viewing position of the camera, in process time, was behind the tool, very close of it, on the middle of the welding seam. The details of the experimental stand are presented in figure 2.

![Figure 1. Welding tool](image1)

![Figure 2. Experimental stand](image2)

The welding parameters used to join these three different alloys are: welding speed set at 100 [mm/min] and the second parameter is rotation speed set at 650 [rpm].
3. Numerical modelling

The numerical model of Friction Stir Welding process for three aluminium alloys, was performed using Coupled Eulerian Lagrange (CEL) simulation technique in software ABAQUUS 6.13. This technique was used to avoid mesh problems for this welding process simulation that involve major deformations. The CEL analysis combined in the same step of the process, has two different approaches: the Lagrangian one and the Eulerian one. This simulation technique has been chosen because it has several advantages such as: can predict the material distribution and the formation of voids through the joining seams, is possible to use complex constitutive laws.

3.1. Parts geometry

All three aluminium alloys used in this study, AA7075, AA2024 and AA6061 were modelled like Lagrangian 3D deformable parts. Dimensional geometry of plates is 80 [mm] x 100 [mm] and thickness of it are identical with experimental study, equal with 2 [mm].

The Eulerian part was modelled using 3D Eulerian sets with length and width identical with Lagrangian dimensional. The thickness of it is 7 [mm], bigger with 1 [mm] than the whole package made by welding parts overlapped. The dimensions of Eulerian part was considerate bigger in order to cover fluid parts, flow and deforms of welding parts. Geometry of the welding tool is identical with the tool used in the experimental study: shape of the pin is cylindrical threaded M6 with the high of the pin of 5.3 [mm] and diameter of tool shoulder of φ22 [mm]. In Abaqus the welding tool is modelled as a rigid part.

One Abaqus extras of Lagrangian elements (welding plates), Eulerian element and welding tool, are represented in figures 3-5.

3.2. Property of materials

The materials used in this paper are: AA7075, AA2024 and AA6061. The elasticity is based on the Hook's law and put forward from the Young's Modulus and Poisson’s Ratio. The parameters of these three aluminium alloys such as: density, Young’s Modulus and other are presented in table 1.

| Material  | Melting point [°C] | Density [Kg/m³] | Young’s modulus [MPa] | Poisson’s Ratio | Inelastic Heat Fraction | Conductivity [W/m°C] | Specific Heat [J/kg°C] |
|-----------|--------------------|-----------------|------------------------|----------------|------------------------|----------------------|----------------------|
| AA 7075   | 635                | 2810            | 72000                  | 0.33           | 0.9                    | 134                  | 960                  |
| AA 2024   | 500                | 2773            | 73000                  | 0.33           | 0.9                    | 121                  | 875                  |
| AA 6061   | 582                | 2700            | 67000                  | 0.33           | 0.9                    | 167                  | 960                  |

For plasticity it is used the Johnson-Cook model, it achieves real deformations, deformation speeds and temperature changes. These three parameters are found in the empirical equation that is represented in bellow sentence (1):

\[
\bar{\sigma} = [A + B \cdot (\varepsilon^{pl})^n] \cdot [1 + C \cdot \ln \left( \frac{\varepsilon^{pl}}{\varepsilon_0} \right)] \cdot \left[1 - \left( \frac{T - T_{ref}}{T_{melt} - T_{ref}} \right)^m \right] [\text{GPa}]
\]  (1)
The coefficients of equation are: $\bar{\sigma}$ - the yield stress; $\bar{\varepsilon}^{pl}$ - the effective plastic deformation; $\dot{\varepsilon}^{pl}$ - the effective deformation speed; $\varepsilon_0$ - the normal deformation speed; $A$, $B$, $C$, $n$, $T_{\text{melt}}$, and $m$ are material constants, which are listed in Table 2; the $n$ exponent takes into consideration the hardening of the material, while $m$ depends on its melting; $C$ is influenced by the speed of deformation; $T_{\text{ref}}$ is the ambient temperature, which is 20 [°C] in this case and at which we determine the parameters $A$, $B$, $n$; $T_{\text{melt}}$ is the material’s melting temperature. In this law, the first term represents the hardening of the material, the second term represents the influence of deformation speed and the third represents the effect of temperature [12].

Table 2. Constants of Johnson-Cook material model [9, 10, 13]

| Material | A (MPa) | B (MPa) | C   | n    | m    | $T_{\text{melt}}$ (°C) |
|----------|---------|---------|------|------|------|------------------------|
| AA 7075  | 546     | 678     | 0.002| 0.71 | 1.56 | 630                    |
| AA 2024  | 265     | 426     | 0.002| 0.34 | 1    | 500                    |
| AA 6061  | 324     | 114     | 0.002| 0.42 | 1.34 | 583                    |

In software Abaqus, the materials are not assigned directly to the body, it is necessary to create a section that includes the material and after that, to assign this section to the body. The Eulerian body was split in three parts in height, corresponding for each materials plate, and received all sections of the characteristics of the three materials.

3.3. Assembly

For assembly step was respected experimental study, presented in figure 6, the Lagrangian parts were positioned in overlap, with AA7075 in upper position, AA2024 in middle position and AA6061 in lower position. The Eulerian part was to include all three Lagrangian parts, start by the lower limit and to overcome with 1 [mm] in up the first alloy, AA7075. The welding tool was positioned in contact with the upper Lagrangian part, at the middle of welding plates, figure 7.

![Figure 6. Real welding system](image)

![Figure 7. Assembly welding system in Abaqus](image)

3.4. Process steps

This welding numerical simulation is performed in two different steps. First of it is plunging and the other one is advanced. The time spent of this step is approximatively similar with performing experiment.

In experimental situation the plunging step is performed in 90 [s], with a dwelling period and in simulation this step is performed in 5 [s] from time consumption. This step has a big influence in sparing of temperature at the start of the welding seam. The second step it is represented by the advanced, considerate a welding speed equal with 100 [mm/min] and a length of the welding seam equal with 50 [mm] this step is completely performed in 30 [s].
3.5. Interaction and boundary conditions
For this simulation, the contact between deformable parts and welding tool was set as “general contact” and the friction tangential factor was set at 0.3 [6]. The second interaction is created between the welding system and the environmental temperature set at 20 [°C].

For numerical simulation of FSW process it is necessary to create three boundary conditions, first of it is represented by blocking of the welding system. The second one is represented by plunging of the welding tool. This movement of the tool is composed by vertical plunging speed set at 1.08 [mm/s] and rotation speed set at 68.033 [radians/s] similar with 650 [rpm]. The last boundary condition it is the advance of the welding tool, this is composed by welding speed and rotation speed. The welding speed was set at 1.666 [mm/s] similar with experimental 100 [mm/min] and rotation speed was kept at 68.0333 [radian/sec].

3.6. Mesh of the workpieces
The creation of mesh is the last step, before launching the simulation job in the same time, is the most difficult of it. In this simulation step, important to define the correct dimension of mesh, a gross mesh cannot give quite accurate results, on the other hand a smaller mesh can be very costly. To optimize and to reduce number of nodes is necessary to create partitions and to create meshes according to the position of partition in welding system. In CEL simulation is necessary to create mesh just at Eulerian part and at welding tool, but the last one is a rigid body and number of meshes can be limited.

For actual numerical model the mesh grain has the same shape, parallelepiped rectangular, and the dimension of it is different according to the plates position. These are split as follows: 1 [mm] x 1[mm] x 1.5 till 7 [mm] on the welding seam; 1 [mm] x 1 till 3 [mm] x 1.5 till 7 [mm] in the right lateral and the left lateral of the welding seam. Collectively, this mesh of the Eulerian part has in total 7840 elements, the welding tool is composed of 2195 elements. In figure 8 and figure 9 are represented finite element mesh for Eulerian part and for welding system.

![Figure 8. Finite element mesh of Eulerian part](image)

![Figure 9. Finite element mesh of welding system](image)

4. Results and discussions
For finite element method is important to validate the numerical model comparing of it with real one. The most common criteria used to validate a model, are: to compare the temperature evolutions and distribution in the welding seams; to compare axial force applied to the tool in process step, analyse the residual stress and to compare the deformation and the shape of the welding seam [12]. In this study will be analysed the temperature evolution and deformation of the material in the process time. For Friction Stir Welding process of aluminium alloys the temperature in process time are between 0.7 – 0.9 [%] by the melting point of the joined alloys. In this experiment the measured temperature is recorded in upper surface, on the AA7075 alloy, for this alloy the value of melting point is the bigger one. Consequently, the peak temperature is necessary to be lower than 572 [°C] (0.9 * melting point of AA7075).

For temperature analyse module “TEMPMAVG” was used, this module interprets the temperature as a mass fraction weighted average of all materials in the element.

The simulate temperature in plunging step was analysed like an average of temperature behind of the of the welding tool, the average was calculated using the temperature from the nodes selected in figure 10. As well as in experimental study, the maximum value of temperature was achieved at the end of
plunging step and it is equal with 480 [°C], presented in figure 11. In numerical simulation, the average temperature at the end of the plunging step (after 4.9 s) is equal with 523 [°C]. This difference can be explained by the time spent within the plunging step, in experimental situation this was bigger by 18 times than the numerical simulation.

![Figure 10. Selected nodes used to calculate the temperature average in plunging step](image)

![Figure 11. Temperature evolution in experimental study](image)

![Figure 12. Capture made with a thermographic camera in process time](image)

The temperature evolution for advancing step was calculated on three different lines, perpendicular on the welding seam, to the joint surface. This method has been chosen to be similar with special thermographic camera used in experimental study. All temperature dates recorded with special camera, was converted in a graph presented in figure 11.

These three lines were positioned at 5 [mm] to the welding start, at 10 [mm] and near the welding tool, at 15 [mm]. For all three sections, the temperature was measured in the same process step, while tool performed the 15 [mm] of the welding seam. Position of measured values on the welding system are presented in figure 14.

The effective value of temperature in measured areas are presented in graphic from figure 13. The bigger temperature is achieved in the middle of the welding seam, followed by the shoulder of the tool. For all three line was identified the same pattern and the similar diagrams, the difference between them are represented by the maximum value of the temperature. The bigger value was measured for the section near the welding tool.

In experimental procedure, the average temperature achieved on the first 15 [mm] of welding seam is around at 475 [°C]. In numerical model, the average of temperature calculated nearby welding tool (15 mm in both sides, by the middle of the welding join) is presented in table 3. All three values recorded are very close to the experimental result.
Table 3. Average of temperature nearby welding tool

| Position on the welding seam | Average of temperature [°C] |
|-----------------------------|----------------------------|
| 5 [mm] to the start          | 471                        |
| 10 [mm] to the start         | 493                        |
| 15 [mm] to the start         | 532                        |

In this model, the visual aspect of the welding seam is similar with experimental study. In the plunging step, tool displaces the material and penetrates all three plates. In figure 16 is captured the end of the plunging step, here it is present the hole performed by the tool pin. This hole is moving along the welding seam with the tool and at the end of the join remains present, figure 15.

At the start of the welding seam the welding materials is pushed to the outside of the tool shoulder, this aspect is similar in figure 15, numerical model, and figure 16, experimental study.

Deformation of the plates in the advancing step are approximate similar with the experimental study. In the numerical simulation, the welding seam is not linear along of the join, but this shape can be influenced by dimension of mesh. The visual aspect of the welding seam, after welding can be observed in figure 17, as well as in these figures can be observed the welding seam performed by the numerical model.
5. Conclusions
The present paper describes the following steps: to realise a numerical model for Friction Stir Welding process and compare this model with an experimental study performed with the same value of input parameters. The goal of simulation is to minimise cost, time spent and resources used with the experimental study.

1. Comparing thermal distribution for numerical model and real results, in plunging and advancing step, it is proved that differences are smaller. In both steps temperature simulated is bigger than temperature recorded in experimental process, for plunging step is bigger with 9% and for advancing step is bigger with 5% (calculated like an average for all three analysed sections). This aspect confirms that the presented model is developed using the correct conditions.

2. The visual aspect of the welding seam it is approximate similar in simulation and experimental study. This characteristic of the welding seam can be improved using a smaller mesh dimension and creating a bigger Eulerian, but all of this involves more time spent to simulate all process.

3. In future, this numerical model can be improved using the next hints: realising the smaller mesh in welding zone because this can improve visual aspects, realising a bigger Eulerian to increased space necessary for deformation and using characteristic materials, variated by temperature.

The general conclusion is the next: in this study a numerical model was created, by 7840 mesh, that simulates a FSW process for three different aluminium alloys: AA7075, AA2024 and AA6061, using CEL method and was validated comparing the distribution temperature and the visual aspect of the welding seam.

References
[1] Mohamed M A E, Osman T A, Mokadem A and Elshalakany A B 2018 Adv. J. of Grad. Res.4 1
[2] Bosneag A, Constantin M A, Nitu E and Iordache M 2017 Mat. Sci. and Eng. 252 012041
[3] Bosneag A, Constantin M A, Nitu E and Iordache M 2018 MATEC Web of Conferences 178 03003
[4] Andrzej K, Rafal K, Krzysztof O, Dawid W and Tomasz T 2018 MTAEC9 52 283
[5] Xiawei Y, Wuyuan F, Wanya L, Yaxin X, Qiang C, Tiejun M and Weibing W 2018 Sci.and Tech. of Weld. and Join. 23 704-14
[6] Iordache M, Nitu E, Badulescu C, Iacomi D, Botila L and Radu B 2016 Adv. Mat. Res. 1138 113-8
[7] Sanjeev N K and Ravikiran B P 2015 SIMULIA Community Conference 1062-79
[8] Iordache M, Badulescu C, Iacomi D, Nitu E and Ciuca C 2016 Mat. Sci. and Eng. 145
[9] Flores-Johnson E A, Luming S, Irene G and Giang D N 2014 Comp.Sci. and Tech. 96 13-22
[10] Hassan I, Waqas S, Muhammad A, Ahmed A and Tarek M 2018 Int. J. of Eng. & Tech. 7 37-41
[11] Chao Y and Qi Y 1998 J. of Mat. Proc.g & Man. Sci. 7 215-233
[12] Constantin M A, Bosneag A, Iordache M, Badulescu C and Nitu E 2016 Appl. Mech. and Mat. 834 43-8
[13] Adibi-Sedeh A, Madhavan V and Bahr B 2003 J. of Manu. Sci. and Eng. 125 656-66