Numerical simulation of friction stir welding of pure copper plates

M A Constantin$^{1,*}$, E L Niţu$^2$ and C Bădulescu$^3$

$^1$University Politehnica of Bucharest, Machine Building Technology Department, Splaiul Independenţei Street No. 313, Romania
$^2$University of Pitesti, Manufacturing and Industrial Management Department, Târgul din Vale Street No.1, Romania
$^3$ENSTA Bretagne, Mechanics and Systems Laboratory of Brest, 2 Rue François Verny, 29200 Brest, France

Email: constantin_marius.adrian@yahoo.com

Abstract. Friction Stir Welding (FSW) is a joining process which is performed at low temperatures, lower than the melting temperature of the base material, thus it is considered a solid state welding process. This feature makes it suitable for copper welding, material whose thermal diffusivity is higher than that of most steel alloys. Large heat losses identified at copper welding by fusion welding processes are thus reduced using FSW process. Because of the shown efficiency and the innovative character of this process, many actions have been initiated in order to optimize it. The aim of this paper is to develop a three-dimensional coupled thermo-mechanical finite element (FE) model of FSW process for pure copper plates using the Coupled Eulerian-Lagrangian (CEL) formulation given in the FE code ABAQUS V6.13. The CEL formulation is one of the few formulations that are capable of handling with such large deformations. The developed numerical model was validated by comparing its results related to the temperatures calculated in the process time with those measured in performed experiments using the same process parameters. This model was capable of simulating the FSW of copper plates and of anticipating the temperature distribution and burrs formation in the weld bead.

1. Introduction
Friction Stir Welding is a solid-state joining process based on friction and plastic deformation heating generated by the interaction between a non-consumable welding tool and the work-pieces. This process is most often developed in four phases: plunging, dwelling, translational phase and retreating. In the plunging phase, the welding tool is positioned perpendicularly on the work-pieces and rotates around its own axis as it progressively penetrates the work-pieces. When the welding tool’s pin is fully plunged and the tool’s shoulder is in contact with the work-pieces material, the dwelling phase begins, thus the material is brought to the temperature required to carry out this process. The translational phase involves the shifting of the welding tool (having the welding speed) on the welding direction in order to stir and transfer the plasticized material behind it, this way creating the welded joint, figure 1. The retreating phase is represented by the retreat of the tool from the welded material at the end of the welded joint.

Conventional fusion welding methods of copper do not have enough strength to be considered as load carrying joints, that is why scientific research of FSW process presents this increased interest. Moreover, the identified problems related to alloying elements, thermal conductivity of the material,
shielding gas, joint design, welding position and surface condition, that appear by using the fusion welding processes are avoided by using FSW process [1].

Figure 1. A schematic presentation of the friction stir butt welding process.

From the analysis of the scientific literature it can be noticed that the numerical simulation of the FSW process is performed using several methods. More methods were used due to the problematic characteristics of this process represented by the large deformations generated during the process and the material properties changes depending on the temperature. Using Lagrange method often leads to excessive mesh distortion and the Arbitrary Lagrangian Eulerian (ALE) formulation has difficulty in defining interactions and loads. However notable results were reached using the ALE formulation, in which most authors overcame these large deformations by continuously remeshing or local mesh refinement. In order to avoid these issues, in this paper the Coupled Eulerian-Lagrangian (CEL) method is investigated to model the FSW process. The CEL method uses a lagrange-plus-remap algorithm. When the mesh is distorted during a Lagrangian increment, it is restored by calculating the material flow between elements and subsequent remapping [2].

Most studies that are developed to simulate the FSW of aluminium alloys using CEL formulation are used to predict the distribution of the temperature and the occurrence of defects during the FSW process, including tunnel defects, cavities and excess flash formation [3,4].

In the case of the numerical simulation of FSW of copper, these studies are more limited. Pashazadeh et.al [5] developed a FEM using ALE formulation, from which it was concluded that closer to the pin the values of plastic strain are higher in the advancing side than in the retreating side, but away from the pin, it is the opposite, indicating a wider thermo-mechanical affected zone in retreating area. Karrar et.al [1] used CEL formulation to simulate the FSW of copper plates. The conclusion was that the CEL model was able to predict the temperature in the weld bead at an accuracy level of 3.7% and the axial forces at an accuracy level of 10%.

This paper describes in a clearer way the development of a valid CEL finite element model for FSW of commercial pure copper plates and the results obtained using this simulation.

2. Development of the numerical model
To develop a numerical model, the following activities must be carried out: reproducing the geometry of the parts, defining the material's behaviour and the type of contact between the friction surfaces, establishing the boundary conditions and the mesh of the work-pieces.

2.1. Reproducing the geometry of the parts
The four parts that make up the numerical model are the following: one tool, two work-pieces and one Eulerian domain, figure 2, to which the dimensions of the work-pieces and the properties of the work-pieces material are assigned, figure 3. The work-pieces are developed as 3D deformable solid and the tool is modelled as a rigid Lagrangian body, constrained to its specific reference point.
The work-pieces consist of two copper plates of 250 mm by 100 mm by 3 mm. The welding tool used in the experiments is made of P20+S (carbide of sintered tungsten), it has a mono-block structure with conical pin, the diameter of its shoulder is 20 mm and of the conical pin is 4 mm on its base and 3 mm on its top, presenting a pin length of 2.8 mm. The Eulerian domain dimension includes these two work-pieces and a void layer of 0.5 mm thick on the top of these in order to make visible the flash formation during welding, figure 3.

2.2. Defining the material’s behaviour

The material of the work-pieces is Cu-DHP, which is phosphorous de-oxidised copper that is 99.9% pure, presenting just 0.015 - 0.04 % phosphorus. The main chemical composition and the values of the temperature dependent material properties used to develop this numerical simulation were taken from material’s property table, table 1. And for Poisson’s Ratio was used a value of 0.33.

**Table 1. Temperature dependent pure copper properties.**

| Temperature [°C] | 20 | 200 |
|-----------------|----|-----|
| Thermal Conductivity [W/m°C] | 328 | 316 |
| Density [Kg/m³] | 8940 | 8840 |
| Young’s Modulus [GPa] | 120 | 113 |
| Thermal Expansion [10⁻⁶/°C] | 16,8 | 17,3 |
| Specific Heat [J/Kg °C] | 385 | 403 |

The plastic behaviour of the pure copper material from the numerical model is established by the Johnson-Cook’s empirical law integrated in Abaqus/Explicit, which characterizes the relationship between the flow stress \(\sigma\), strain rate \(\dot{\varepsilon}\) and temperature \(T\), according to the following formula:

\[
\sigma = [A + B \cdot (\dot{\varepsilon}_p)^n] \cdot [1 + C \cdot \ln \left( \frac{\dot{\varepsilon}_p}{\dot{\varepsilon}_0} \right)] \cdot [1 - \left( \frac{T - T_{ref}}{T_{melt} - T_{ref}} \right)^m] \]  

where: \(\sigma\) - the yield stress; \(\dot{\varepsilon}_p\) - the effective plastic deformation; \(\dot{\varepsilon}_0\) - the effective deformation speed; \(\dot{\varepsilon}_0\) - the normal deformation speed, which is typically 1 [s⁻¹]; A, B, C, n, T_{melt}, and m are material constants, which are listed in table 2; the n exponent takes into account the hardening of the material, while m depends on its melting; C is influenced by the speed of deformation; T_{ref} is the environmental temperature, which is 25 °C for these experiments and at which are determined the parameters A, B, n; T_{melt} is the material’s melting temperature [6].

**Table 2. Copper’s Johnson-Cook Parameters [1].**

| Material | T_{melt} [°C] | A [MPa] | B [MPa] | C  | n   | m   |
|----------|--------------|---------|---------|----|-----|-----|
| Cu-DHP   | 1083         | 90      | 292     | 0,025 | 0,31 | 1,09 |

2.3. Defining the type of contact between the friction surfaces

The FSW process relies on the heat produced by the friction between tool and work-pieces, which softens, stirs and welds the work-pieces material, that’s the reason why is so important for the numerical simulation of the FSW to well define the friction phenomenon. A "general contact" was defined between the welding tool and the work-pieces, in which tangential behaviour is described by a friction law, the normal behaviour is described by a "hard contact" enabling the parts to separate after contact and for heat generation it’s used an inelastic heat fraction of 0.9. This general contact is used in explicit dynamic analysis of the CEL Formulation [4].

To define the contact between Eulerian domain and the tool, it was used the general defined formulation of Coulomb’s law, as function of slip rate, contact pressure and surface temperature. In this numerical model the friction coefficient, \(\mu\), is assumed to be 0.35 between steel (tool’s material) and copper (work-pieces’ material).
2.4. Establishing the boundary conditions
There are two types of boundary conditions in the simulation of FSW process: mechanical boundaries and thermal boundaries.

Typically, the FSW joints are developed in 4 important steps: plunging, dwelling, translational step and retreating. In order to reduce computational time, in this study are analyzed the plunging and the translational step. The tool is the only moving part, having rotational movement around its own axis and translational movement in Y direction for plunging step and in X direction for translational step, movements that characterize the main process parameters. The lower surface of the Eulerian domain was fixed in all degrees of freedom to avoid its motion.

The heat exchange between environment and Eulerian domain is characterized by a heat transfer coefficient \( h = 30 \text{ W/m}^2\cdot\text{°C} \), which is assigned to all surfaces of the Eulerian domain.

In this numerical model of FSW process there are three controllable process parameters: the angular velocity, the plunge velocity and the welding velocity. The angular velocity was 800 rpm, the plunge velocity was 2.8 mm/s and the welding velocity was 2.5 mm/s. These values were constant during the step in which they were used. The process parameters and the dimensions of the parts were the same as those used experimentally.

Previous presented values of the angular velocity, of the plunge velocity and of the welding velocity are assigned to the tool at the reference point ‘RP1’, figure 2.

![Figure 2. Boundary conditions in Abaqus/Explicit.](image)

2.5. Mesh of the work-pieces
This numerical model uses the advantages presented above, given by the Coupled Eulerian-Lagrangian (CEL) formulation in simulating severe element deformation.

Biased seeding was used to generate a finer mesh in the weld seam line than in the other regions of the work-pieces, in order to use a minimum number of elements to reduce the calculation time. Eulerian domain was meshed using 8900 multi-material thermally coupled Eulerian elements (EC3D8RT8), the mesh being generated using a 0.7 mm thickness of each layer.

The Eulerian implementation in Abaqus/Explicit is based on the volume-of-fluid method. In this method, material is tracked as it flows through the fixed mesh by computing its Eulerian volume fraction (EVF) within each element. Thus, if an element is completely filled with material then it is defined as ‘full’ and its volume fraction is 1. If it is completely free of material, its volume fraction is 0 and it is filled by default with a material without mass and strength called ‘void’ material [7].

For the development of a CEL numerical simulation, one specific thing must be carried out after the elements are meshed. This specific thing is to fill the inside geometry of the Eulerian domain with the material listed in the Eulerian section definition, because by default, all Eulerian elements are initially void. This specific thing is done in two steps. First step is to define an initial volume fraction for the material, selectively using filling elements (in our case, the two Lagrangian work-pieces) and the other is to assign this initial volume fraction to the Eulerian domain, creating in this way the initial shape of the Eulerian materials, figure 3. These initial conditions are only applied at the beginning of
the analysis, when the loads are applied to the material it gets deformed and the volume fractions are accordingly recalculated.

After performing these activities, the work-pieces are removed from the assembly design, their dimensions and properties being memorized by the Eulerian domain, figure 3.

![Figure 3. Eulerian domain mesh.](image)

3. Results and discussions

For the validation of the numerical model, it was compared the resulted temperature distribution data from the numerical simulation with the temperature distribution data from the literature and from the experimental determinations made using the same material and process parameters.

It can be seen that in plunging step the temperature is symmetrically distributed to the weld centreline, figure 4, same results were found in the literature [8] and in the experimental data, figure 5.

![Figure 4. Temperature distribution in plunging step (step time 0.8 [s]).](image)

In the translational step, the temperature field shows a slight asymmetry, figure 6. Temperatures calculated by the numerical simulation on the upper surface of the work-pieces, in the transverse direction, at 20 and 50 mm from the beginning of the joint, exactly when the tool transits those areas, indicate the same asymmetry, figure 7. The temperature is higher in the retreating side of the tool compared to the advancing side, because of the higher deposit of material in this area [1,5,9]. This higher temperature on the retreating side during translational step matches the experimental results, where it can be noticed that the heat affected area is more prominent on the retreating side, figure 5.

![Figure 5. The aspect of the FSW joint that used the same process parameters.](image)

![Figure 6. Temperature distribution in translational step (step time 8 [s]).](image)
Figure 7. Temperature distribution in transverse direction.

Figure 8. Measured temperatures during FSW process.

The maximum nodal temperatures estimated by the CEL model after 20 mm (step time 8 [s]) and 50 mm (step time 20 [s]) from the beginning of the translational step are approximately 542 [ºC] and 530 [ºC], values that are close to the experimental temperatures measured behind the welding tool, using a thermographic camera, that indicates a value of approximately 525 [ºC] at 20 mm and 500 [ºC] at 50 mm, figure 8. The average temperature measured during this experiment was 506 [ºC]. The maximum identified temperatures are below the copper melting temperature and were located closer to the FSW tool pin.

4. Conclusions
The paper describes the activities needed to be performed in order to develop a valid finite element CEL model for FSW of commercial pure copper plates and the results obtained using this simulation.

The conclusions from this study can be summed up as it follows:
1. The developed CEL finite element model was capable of simulating the FSW of pure copper plates and to anticipate the temperature distribution in the weld bead, which is in accordance with the experimental results and literature.
2. The main advantages of the CEL formulation are represented by its capabilities of handling with large deformations and of tracking the material flow.
3. A fine mesh is important for a more accurate tracking of free surfaces like burrs or the formation of voids.

As future research directions, we want to improve the current numerical model in order to explore the aforementioned capabilities of the CEL formulation for the FSW analysis of pure copper plates.

Acknowledgements
The work was done with the support of the National Institute for Welding and Material Testing, ISIM Timișoara.

References
[1] Karrar G, Shuaib A N, Al-Badour F A, Merah N and Mahgoub A K 2014 Int. Mech. Eng. Congr. & Exp. (Montreal: ASME)
[2] Noh W F 1964 Methods in Computational Phy. 3 117-79
[3] Hossfeld M and Roos E 2013 Int. Conf. on Adv. Man. Eng. and Techn. (Stockholm)
[4] Iordache M, Badulescu C, Iacomi D, Nitu E and Ciucu C 2016 Mat. Sci. and Eng. 145
[5] Pashazadeh H, Teimournezhad J and Masoumi A 2014 Mat. and Des. 55 619-32
[6] Al-Badour F, Nesar M, Abdelrahman S and Bazoune A 2014 Int. J. of Adv. Man. Tech. 72 607-17
[7] ABAQUS Inc. 2014 ABAQUS analysis user’s manual
[8] Li W, Zhang Z, Li J and Chao Y 2012 J. of Mat. Eng. and Perf. 21 1849-56
[9] Jabbari M 2014 Computational Mat. Sci. 81 296-302