



HAL
open science

FEM Prediction of Temperature and Residual Stresses Distribution During Friction Stir Welding of 2017A Aluminum Alloy

Oussama Mimouni, Riad Badji, Afia Kouadri-David, Mohamed Hadji, Nabil
Chekroun

► **To cite this version:**

Oussama Mimouni, Riad Badji, Afia Kouadri-David, Mohamed Hadji, Nabil Chekroun. FEM Prediction of Temperature and Residual Stresses Distribution During Friction Stir Welding of 2017A Aluminum Alloy. ESAFORM, Apr 2017, DUBLIN, Ireland. hal-02561174

HAL Id: hal-02561174

<https://hal.science/hal-02561174>

Submitted on 3 May 2020

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.

FEM Prediction of Temperature and Residual Stresses Distribution During Friction Stir Welding of 2017A Aluminum Alloy

Oussama Mimouni^{1,a}, Riad Badji^{2,b}, Afia Kouadri-David^{3,c}, Mohamed Hadji^{1,d},
Nabil Chekroun^{4,e}

¹Laboratory of Aircrafts, University of SaadDahlab, Blida 1, Blida, Algeria.

²Center of Industrial Technologies CRTI, P.O Box 64, Cheraga, Algeria

³Laboratory of Civil and Mechanical Engineering INSA Rennes, France.

⁴Research Laboratory of Manufacturing Technology Mechanical / ENPO- Algeria.

^aoussama.mimouni@hotmail.fr

^br.badji@csc.dz

^cafia.kouadri-henni@insa-rennes.fr

^dhadji.mohamed@gmail.com

^echakroun1982@yahoo.fr

Abstract. The structures integrities containing welds are evaluated using numerical methods to characterize the residual stresses field induced during welding. These methods are mainly based on coupled thermal and mechanical analysis using the finite element method. The present paper deals with the case study of a straight welding of two plates in 2017A aircraft aluminum alloy.

Keywords: welding, residual stress modeling, finite element, aluminum alloy.

1. Introduction

A method such as Fiction Stir Welding (FSW) generates a distribution of residual stresses in the vicinity of the welding join. The presence of these residual stresses should have a serious impact on the mechanical behavior. In fact, there are several techniques used to measure the residual stresses resulting from welding, which are however difficult to implement and less preciseness. Using numerical methods such as the finite element method, it is possible to establish the residual stresses state in a sheet and their distributions. Indeed, any welded joint shows both deformation and permanent stresses [1]. Residual stresses and shrinkage are due to thermal and metallurgical phenomena involved in the welding. The main strain types are [2]: the longitudinal and transversal shrinkage, bending, buckling, V-distortion and the rotational distortion (see Figure 1). It is clear that the knowledge of residual stresses distribution in a sheet is very important, since they seriously affect its behavior [3]. In fact, the residual stresses formation is associated with various phenomena and interactions such as [4]:

- Localized impact on the flow constraints affecting the sign and direction of residual stresses;
- Residual stresses are combined based on load superposition;
- Formation of cracks even without any loading;
- Increased risk of brittle fracture caused by the high local stressing;
- Residual stresses affect strength, fatigue, buckling and corrosion sensitivity of the sheet.

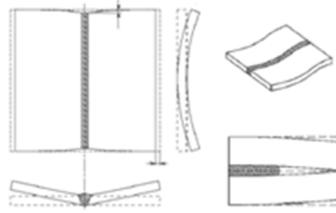


Figure.1 Types of shrinkage due to welding.

2. Modeling and Methodology

Welding analysis has to consider three major coupled fields [2], interacting each other more or less strongly (see Figure 2):

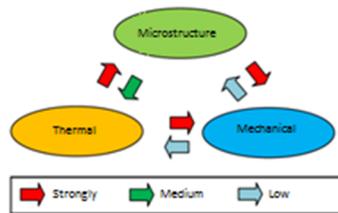


Figure.2 Coupling of the different types of welding analysis.

2.1. Welding Modeling

The objective is to present the different procedures and assumptions that lead to the development of the welding simulating model in order to predict the residual stresses generated in the welded sheets. A thermo-structural transient analysis is performed to simulate this process, using a finite element model developed, with Ansys® commercial software to model the welding of two 2017A aluminum alloy plates.

2.1.1. Modeling of the material

We assume that the material thermomechanical properties in the welding case are the same as those obtained through heat treatments, in order to take into account phenomena of latent heat and convection in the stirring zone. The thermomechanical properties of 2017A material are given in table1 [5].

Table 1 - Properties of aluminum alloy “2017A”

Heat capacity C_P (J / k g ° K)	Thermal expansion α (microns m ⁻¹ K ⁻¹)	Modulus of elasticity E (GP)	Modulus of rigidity G (GP)	Poisson coefficient n	É density of the material r (kg m ⁻³)
899	23.5	70	26.4	0.3	2700

2.1.2. Thermal properties

- **Thermal conductivity**

The thermal conductivity decreases when the temperature rises until reaching its minimum value, after which it stabilizes slightly (see Figure 3).

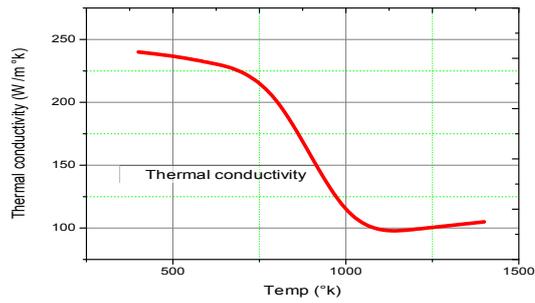


Figure.3 Evolution of the thermal conductivity (λ) as a function of temperature.

Most authors consider the material density (ρ) as constant throughout the welding process, even it expected to decrease slightly with increasing temperature [2], in particular because of the metal expansion [6]. The density value of 2017A aluminum is equal to 2776 kg m⁻³.

2.2. Welding process parameters

The pin rotate with a constant angular velocity of 1250 rpm and welding speed velocity of 36 mm/min. The limits of flow domain are modeled as walls with convective heat transfer to, a convective heat transfer coefficient of 50 W.m⁻².K⁻¹. The environmental temperature is assumed as 25°C.

3. Workpieces geometry

The geometry of the two parts to be welded is selected as simple as possible, ie a flat plate, to eliminate any risk of non-convergence due to the geometric complexity. The dimensions selected for each plate are 0.3 x 0.2 m².

3.1. Element Type

The plate meshing is achieved using shell elements. This type of element allows resolutions with only one element layer along the thickness which significantly reduces the execution time of this model (see Figure 4 and 5). [7]

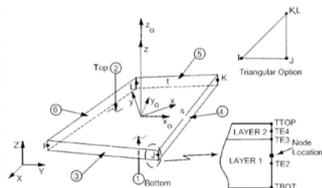


Figure.4 thermal shell element.

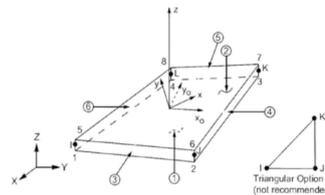


Figure 5: structural shell element.

3.2. Meshing types

Since the large temperature fluctuations are essentially localized in the welding zone, it should be necessary to refine smoothly the meshing in this area (see figure 6).

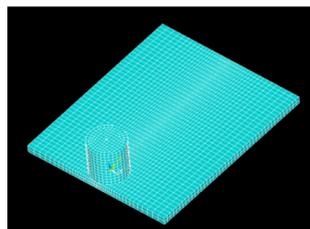


Figure.6 Mesh structured.

4. Structural Analysis

4.1. Geometry, meshing and boundary conditions

The same model (geometry and meshing) set in the thermal analysis is reused in the structural analysis by replacing only the thermal elements by structural ones and changing the boundary conditions (see Figure 7).

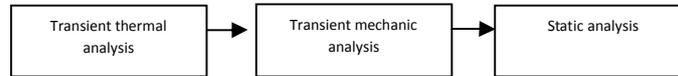


Figure.7 Programming flowchart.

4.2. Cooling

Cooling is achieved through free convection on the upper and lower plate faces (see Figure 8-2).

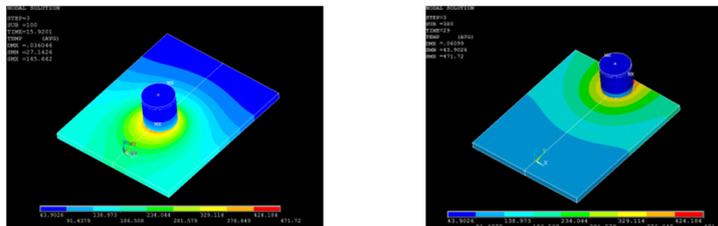
5. Results analysis

5.1. Temperatures distribution

As shown in figure 8-1, the solution reaches the stationary state when the mechanical equilibrium is attained after a number of additional iterations. It also shows the evolution of minimum, maximum and average temperatures calculated on the entire field of study.

The figure (see Figure 8-1) shows that the point of maximum temperature following the displacement of the center of the tool to the tip of the plate. At the end of cooling, the temperature at any point of the plate becomes substantially equal to the initial temperature $T = 25\text{ }^{\circ}\text{C}$ (see Figure 8-2).

welding



**Figure.8-1 surface temperature distribution.
cooling**

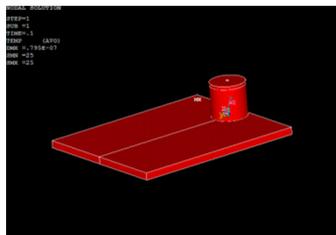


Figure.8-2 Surface distribution of the cooling temperature.

5.2. Distribution of surface residual stress

The figure (see Figure 9) represents the residual stresses distribution of a theoretical point of view.

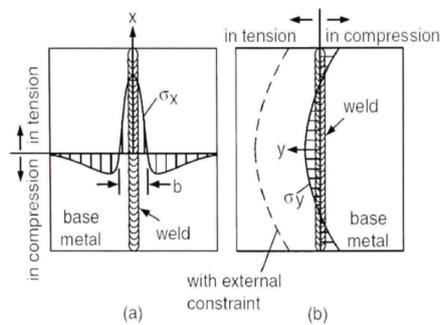


Figure.9 Distribution of residual stresses due to welding.

5.2.1. Longitudinal residual stresses

The figure (see Figure 10-a) shows the distribution over the entire plate, longitudinal residual stresses obtained. Along the weld seam is seen to be the most stressed area, tensile stresses act. In the rest of the plate, compressive stresses are created for balancing the first to reach substantially zero at the edge of the plate with an asymmetric tendency.

5.2.2. Cross residual stress

Transversal residual stresses appear in areas of tension and compression which are alternated along the weld line (see Figure 10-b). The central zone of the weld seam undergoes transversal tensile and both areas surrounding the ends undergo a principle of equilibrium, compressive stresses. These are the largest. The rest of the plate undergoes tensile whose value tends to zero at the edge of the plate with an asymmetric tendency.

5.2.3. Von Mises Stresses

The figure (see figure 10-c) shows the distribution of Von Mises equivalent stresses. The constraints are mainly located in the weld line and its vicinity.

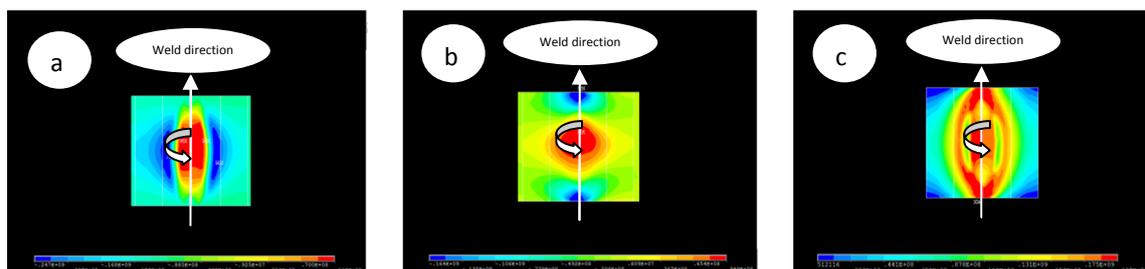


Figure.10 Residual stresses: (a) longitudinal (b) Transversal. (C) Von Mises.

About the longitudinal stresses (see Figure 10a), the maximum values are within the tool zone zone (Nugget) and the heat affected zone (HAZ) with values between -25 MPa and 100 MPa. Also, the values of the transversal stresses (see Figure 10-b) vary between -20 MPa and 50 MPa and located mainly along the weld line.

6. Conclusion

The objective of this work has focused on the numerical prediction of residual stresses generated by the FSW welding process of two flat sheets of 2017A aerospace aluminum alloy, using the finite element method. The analyzes were performed using the software Ansys®, using the double transient coupled-fields method. The approach proposed in this study is the realization of a thermostructural modeling. In conclusion, this study demonstrates that the finite element model used provides very good results for the residual stresses.

REFERENCES

1. Lu Jian. (1996). Handbook of measurement of residual stresses. Lilburn, GA: Fairmont Press Inc.
2. Pilipenko, Artem. (2001). Computer simulation of residual stress and distortion of thick plates in multielectrode submerged arc welding: Their technical mitigation Dr.ing, Norway, Norgesteknikskaturvitenskapeligeuniversitet (Norway)...
3. Zinn, W., and B. Scholtes. (2002). Residual Stress Formation Processes Pendant Welding and Joining. Handbook of Residual Stress and Deformation. p. 391-396. ASM International.
4. D. Kaplan and G. Murry (2001) .The name ph é è thermal nes, m e m e chanical tallurgiques and the affect area é e heat welding. M e m e mechanical metallurgy and welding, Lavoisier .
5. Perry's Chemical Engineers' Handbook 8th Edition, Section 2, physical and chemical data.
6. D. Berglund, A. Lundb â ck THE Lindgren (2001) Three-dimensional finite element simulation of laser welded stainless steel flat NUMIFORM'01, The Seventh International Conference on Numerical Methods in Industrial Forming Processes, Japan, Toyohashi, pp. 1119-1123.
7. ANSYS 12.1, section help, Mechanical APDL Documentation Descriptions Part I, Element Library.