

DEVELOPMENT OF STRATEGIES TO REDUCE THE COMPUTATIONAL COST IN SIMULATIONS OF WELDING PROCESS

Rafael de Souza Pereira
José Dásio de Lira Junior
Pablo Batista Guimarães
Tiago de Sousa Antonino

Federal Institute of Education, Science and Technology of Pernambuco. Av. Professor Luiz Freire, 500, Cidade Universitária, Recife, Pernambuco.

josedasio@recife.ifpe.edu.br, rafaeldesouza51@hotmail.com, pabloguimarães@recife.ifpe.edu.br, tiagoantonino@recife.ifpe.edu.br

Abstract: The finite element method (FEM) is a numerical method that aims to obtain approximate solutions to engineering problems. Thermal problems may have approximate solutions, however, in most cases, the resolution of the problem generates a large computational cost. Thus, the development of strategies to reduce the computational cost is very important. This work developed strategies to improve the quality of mesh in computer simulations of welding process based on FEM. The methodology is applied in simulations of the TIG welding process (Tungsten Inert Gas), also known as GTAW (Gas Tungsten Arc Shielded-Welding). In welding problem, geometric models in two and three dimensions were developed and compared. The graphics of thermal cycles were obtained. The results showed the need for strategies to reduce the execution time of welding simulations based on FEM.

Keywords: Welding Simulation, Tig Welding Process, Finite Element Method

1. INTRODUCTION

The finite element method (FEM) is a numerical method that aims to achieve approximate solutions to engineering problems. Welding simulation problems may have approximate solutions, however generate a large computational cost. For this reason, the development of strategies to reduce the computational cost is critical in numerical problems. One possible strategy is to use geometric models with only two dimensions to represent three-dimensional problems (Carmo e Faria, 2014).

This work developed strategies to improve the quality of mesh in computer simulations of welding process based on FEM. The methodology is applied in simulations of the TIG welding process (Tungsten Inert Gas), also known as GTAW (Gas Tungsten Arc Shielded-Welding). In welding problem, geometric models in two and three dimensions were developed and compared. The work is developed in Abaqus software - CAE (Hibbit, 2012), which is a software based on FEM. The Goldak heat source was used to simulate the welding process (Goldak, *et al.*, 1984; Goldak and Akhlaghi, 2005; Fu, *et al.*, 2011). Thermal cycles to the same problem are obtained using models in two and three dimensions, the execution times are evaluated and compared.

2. MODELING OF THE WELDING PROCESS

In the simulated welding process, a concentrated heat source was applied along the butt joint. The heat conduction of a solid, considering the time t , is related to cartesian orthogonal system (x, y, z) and can be expressed by Eq. (1) (Antonino, 2014).

$$\frac{\partial}{\partial x} \left(K_T \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left(K_T \frac{\partial T}{\partial y} \right) + \frac{\partial}{\partial z} \left(K_T \frac{\partial T}{\partial z} \right) + q_0 = \rho c \frac{\partial T}{\partial t} \quad (1)$$

Where:

T – temperature [$^{\circ}\text{C}$];

x, y, z – orthogonal Cartesian coordinates [m];

t – time [s];

K_T – thermal conductivity [W/m. $^{\circ}\text{C}$];

ρ - density of the material [Kg/m³];

c – specific heat of the material [J/Kg.°C]
 q_0 – heat source [W/m³].

The heat source used was modeled by Goldak (Fig. 1). This heat source is defined by equations Eq. (2) and Eq. (3) (Goldak and Akhlaghi, 2005; Guimarães, 2010).

$$q_f(x, y, z) = f_f \frac{\eta UI}{a_f b c \pi \sqrt{\pi}} \cdot 6\sqrt{3} \cdot \exp\left(\frac{-3x^2}{a_f^2}\right) \cdot \exp\left(\frac{-3y^2}{b^2}\right) \cdot \exp\left(\frac{-3z^2}{c^2}\right) \quad (2)$$

$$q_r(x, y, z) = f_r \frac{\eta UI}{a_f b c \pi \sqrt{\pi}} \cdot 6\sqrt{3} \cdot \exp\left(\frac{-3x^2}{a_f^2}\right) \cdot \exp\left(\frac{-3y^2}{b^2}\right) \cdot \exp\left(\frac{-3z^2}{c^2}\right) \quad (3)$$

where:

q_f – Volumetric power distribution before the torch [W/m³];
 q_r – Volumetric distribution of energy after the torch [W/m³];
 f_f – Distribution of energy before the torch;
 f_r – Distribution of energy after the torch;
 a_f – Length of the weld pool – energy distributed before the torch [m];
 a_r – Length of the weld pool - energy distributed after the torch [m];
 b – Parameter associated of the heat source model [m];
 c – Parameter associated of the heat source model [m].

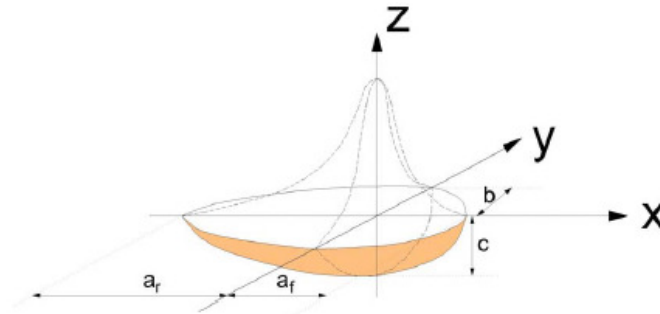


Figure 1. Volumetric source on the Double Ellipsoidal

The parameters U , I , η are connected directly to the welding procedure, while b and c are the geometric parameters of the source and can be determined by metallographic examination. The parameters f_f , e f_r are obtained according to the following equations Eq. (4), Eq. (5) e Eq. (6).

$$f_f + f_r = 2 \quad (4)$$

$$f_f = \frac{2 \cdot a_f}{a_f + a_r} \quad (5)$$

$$f_r = \frac{2 \cdot a_r}{a_f + a_r} \quad (6)$$

3. CASE STUDY

The welding problem was modeled in Abaqus with two and three dimensions (2D and 3D). The case study is a plate with 50 x 60 mm (Fig. 2). Three different cases were developed with a thickness of 3 mm, 4 mm and 5 mm.

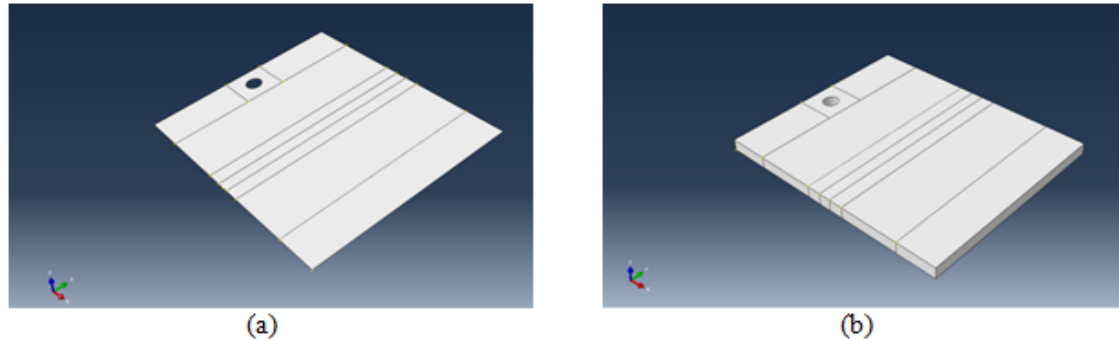


Figure 2. (a) 2D geometric model, (b) 3D geometric model

To identify simulated cases the T XX Y CZ legend was created. Where T is the temperature, XX represents the size of the model (2D or 3D), Y is the thickness of the model, C is the simulated case, and Z is the case number. The simulated cases were named as T2D3C1, T3D3C1, T2D4C2, T3D4C2, T2D5C3, T3D5C3.

The plate material is API 5L X80 steel, this material is very used in the oil and gas industry (Antonino, 2013). The thermophysical properties were used according to Antonino. *et al.* (2014). In this work were used Abaqus elements with linear shapes functions (Fig. 3).

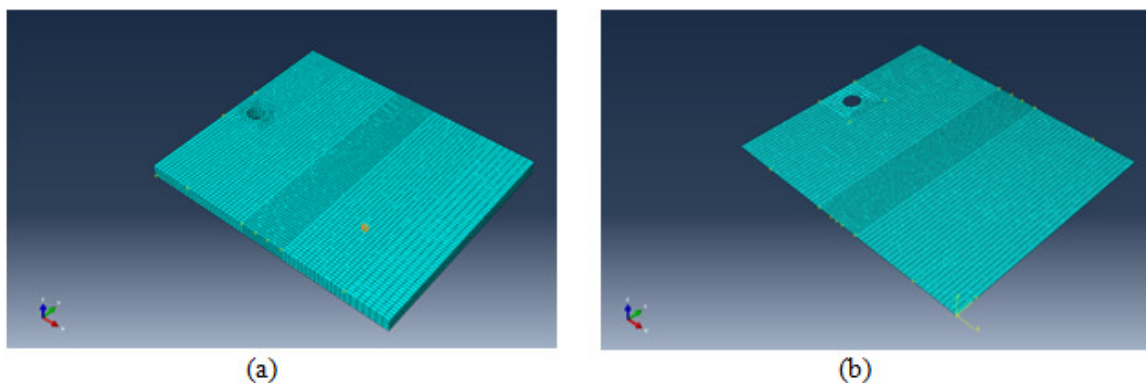


Figure 3. (a) Mesh with 3D elements, (b) Mesh with 2D elements

For more accurate results, the mesh elements of the fusion zone (FZ) and the heat affected zone (HAZ) have edge with dimensions of 0.5 mm. These regions are very important in welding, they occur phenomena as changes in microstructure and change in the physical state of the material (Antonino, 2013; Marques, *et al.*, 2009). However, in the base metal region, the mesh elements have edge dimensions of 1 mm (Fig. 4).

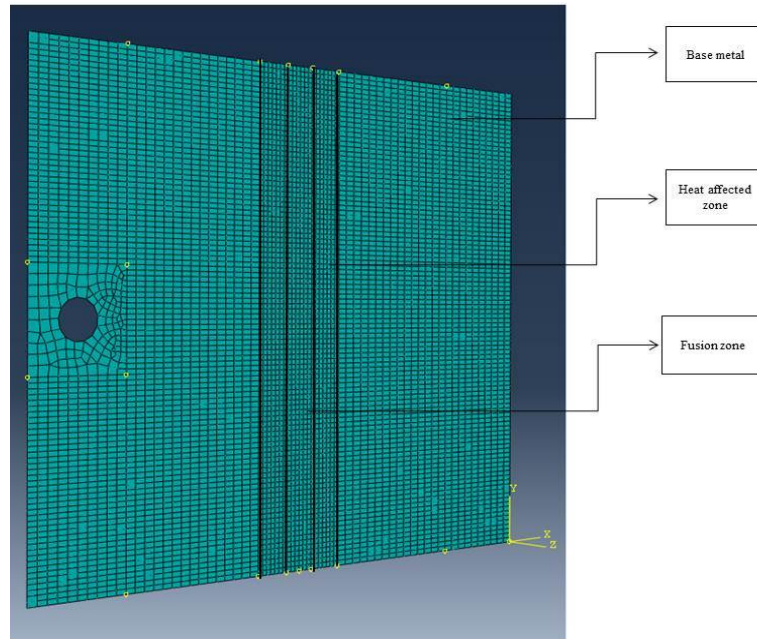


Figure 4. Finite element mesh

The case study was the simulation of the TIG welding process in a butt joint. The welding parameters were added in the program through a abaqus subroutine called DFLUX. The parameters were as follows: electric current of 90 A, voltage of 24 V, pass speed of 1 mm / s. The process energy efficiency is 85% (Antonino, 2013). Thermal cycles were obtained at the same point on the plate for each model. The selected point is represented in Figure 5

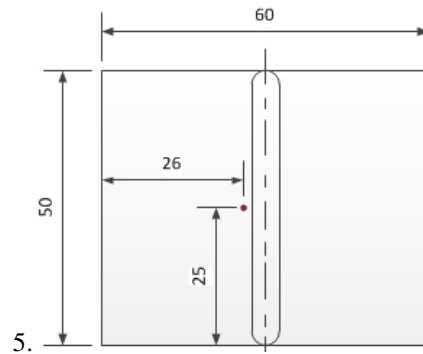


Figure 5. Point selected for the thermal cycle

To check the temperature difference between 2D and 3D models, thermal cycles was obtained and compared. Using the data obtained in the thermal cycles, graphics with the temperature difference versus time between 2D and 3D models have been developed. Three regions were created in thermal cycling graphs: heating zone, the transformation zone of the material microstructure and slow cooling zone.

The heating zone is the region of the thermal cycle in the period between the start of welding until the time when the welding torch touches the selected point of Figure 5. The transformation of microstructure zone represents the first cooling region of the thermal cycle at which the material will suffer microstructural transformations. The minimum temperature in this region is 500 ° C. The slow cooling zone represents the second cooling region of the thermal cycle at which the material will cool down to the minimum temperature of the case. These regions are shown in Figure 6.

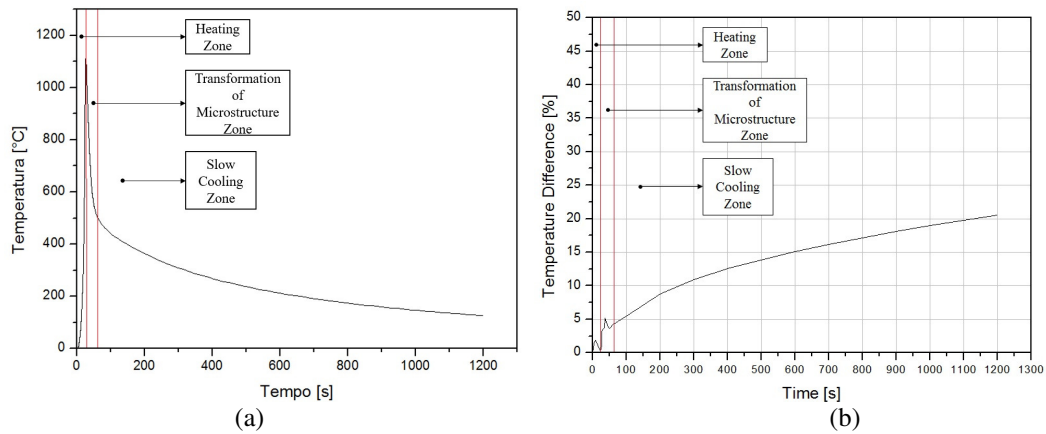


Figure 6. (a) Graphic of thermal cycle, (b) Graphic of temperature difference

4. RESULTS

The temperature fields at time 25 s of the 2D and 3D models with 3 mm thickness are shown in Figure 7.

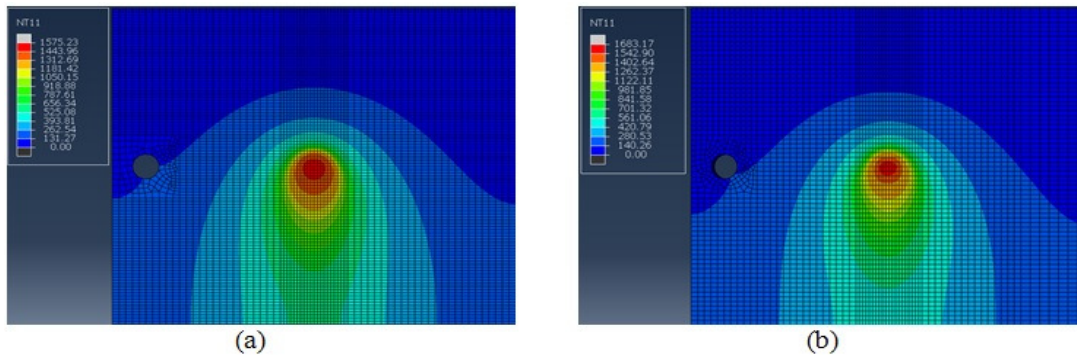


Figure 7. (a) Temperature field for 2D model (T2D3C1), (b) Temperature field for 3D model (T3D3C1)

The comparison of cases with different thicknesses were prepared using the graphs of the thermal cycle and temperature difference graph. The graphs are shown in Figure 8.

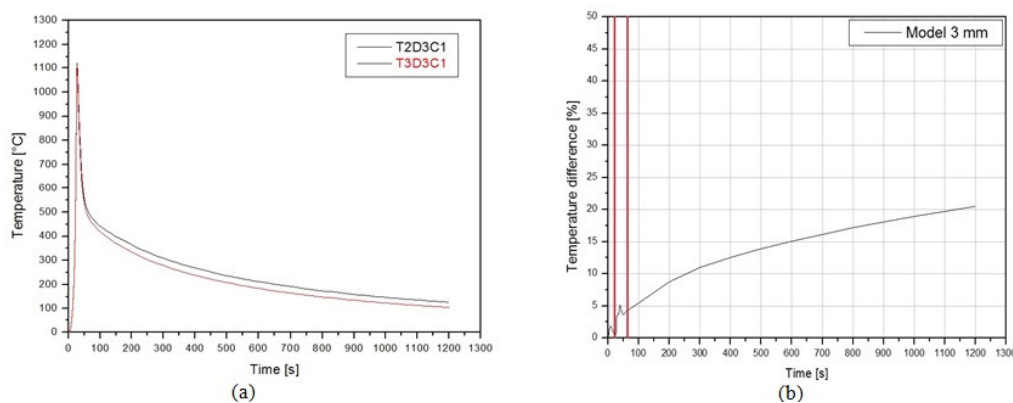


Figure 8. (a) Thermal cycles of the cases T2D3C1 and T3D3C1, (b) Temperature difference graph of the case 1

The results for 3 mm thickness case had a difference of approximately 5% in the heating step between 2D and 3D models. While the phase transformation of the microstructure, the difference in temperature also achieves the value of 5%. However, in slow cooling phase, the temperature difference reaches values near to 21%. The 3D model has 13293 elements, while the 2D model has 6945 elements.

Due to this difference of elements and the number of degrees of freedom, the processing time of 2D and 3D models were, respectively, 1h 43min e 2h 8min. Therefore, the 2D model indicated a 19% reduction in processing time. The values can be found in Table 1.

The comparative graphs of 4 mm thickness model are shown in Figure 9.

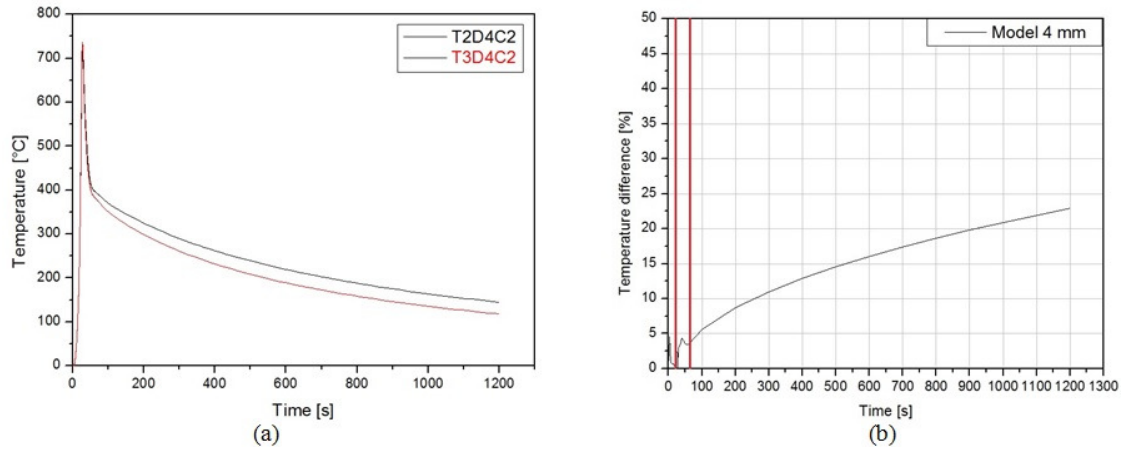


Figure 9. (a) Thermal cycles of cases T2D4C2 e T3D4C2, (b) Temperature difference graph of the case 2

The model with 4 mm thickness showed in the heating phase and transformation of the microstructure less than 5% difference between 2D and 3D models. However, temperatures of slow cooling phase showed 23% difference. The mesh of the 3D model has 18564 elements. The 2D model obtained an 53.68% reduction in processing time. The processing time of the two models is shown in Table 1.

The comparative graphs of 5 mm thickness model are shown in Figure 10.

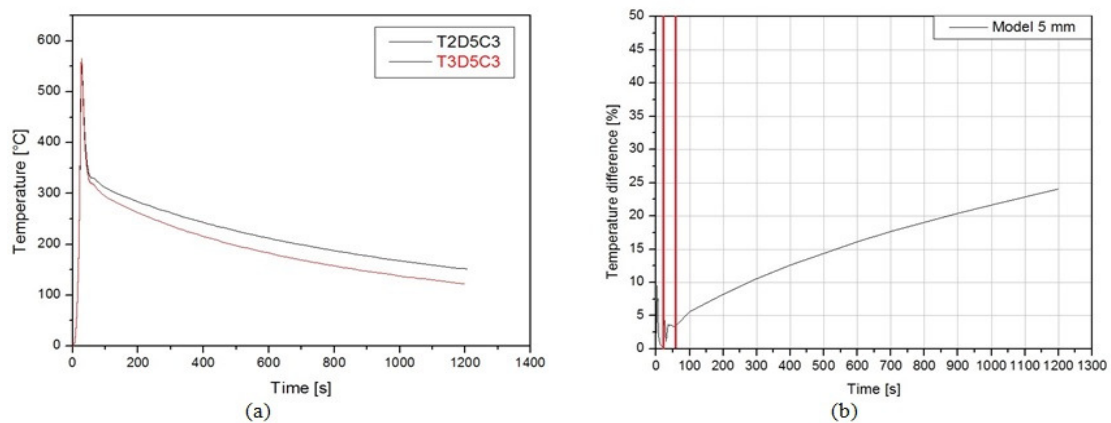


Figure 10. (a) Thermal cycles of the cases T2D5C3 e T3D5C3, (b) Temperature difference graph of the case 3

In this case, the temperature difference showed below 5% between 2D and 3D models of 5 mm thickness. As in previous cases, the highest temperature difference occurs in the slow cooling phase. In this case, the difference reaches the value of 24% approximately. The 3D model has 23205 elements. In this case, the 2D model reduction of processing time was 68.41%. The processing time of the two models is shown in Table 1.

Table 1. Results for the analysis of computational cost

Cases		Number of elements	Processing Time (s)	Processing Time ⁽¹⁾	Time difference (%)
Case 1	T2D3C1	6945	6216.9	1h 43min	19,16
	T3D3C1	13923	7690,7	2h 8min	
Case 2	T2D4C2	6945	6123.5	1h 42 min	53,68
	T3D4C2	18564	13221	3h 40min	
Case 3	T2D5C3	6945	6284.2	1h 44min	68,41
	T3D5C3	23205	19895	5h 31min	

⁽¹⁾ approximate values

5. CONCLUSIONS

In this work, the 2D and 3D models in all cases show a 5% maximum temperature difference in the heating phase, as well as in the cooling phase until the temperature transformation microstructure. In slow cooling phase, the error tends to increase with time of process. For the cooling phase, the approximate temperature differences were 20.5% for the case of 3 mm thickness, 22.85% for the case of 4 mm, and 24% for the case of 5 mm. The results show that the temperature difference tends to increase with increasing plate thickness.

6. REFERENCES

- Antonino, T.S., 2013. "Simulação computacional da influência da velocidade de soldagem na distribuição de temperatura de uma junta de topo do aço API 5L X80". In *XI Congresso Ibero-Americano de Engenharia Mecânica – CIBEM2013*. La Plata, Argentina.
- Antonino, T.S., Guimarães, P.B., Alécio, R.A., Yadava, Y.P. and Ferreira, R.A.S., 2014. "Measurements of the Thermophysical Properties of the API 5L X80". *Materials Sciences and Applications*. Vol. 5, n. 8, p. 617-627.
- Antonino, T.S., 2014. *Estudo numérico-experimental para avaliação do campo de temperatura de uma junta soldada do aço API 5L X80*. Doctoral thesis, Universidade Federal de Pernambuco, Recife.
- Carmo, D.A. and Faria, A.R., 2014. "A 2D finite element with through the thickness parabolic temperature distribution for heat transfer simulations including welding". *Finite Elements in Analysis and Design*. Vol. 93, p. 85-95.
- Fu, G.,Souza, M.I., Estefen, S. F., 2011. "Transient temperature distribution analysis using Goldak's double ellipsoidal moving heat source". *6º Congresso Brasileiro de P&D em Petróleo e Gás*, Florianópolis, Brasil.
- Goldak, J., Chakravarti, A., Bibby, M., 1984. "A new finite element model for welding heat source", *Metall.Trans. B*, Vol. 15, n. 2, p. 299-305.
- Goldak, J.A., Akhlaghi, M., 2005. *Computational Welding Mechanics*. Springer, New York.
- Guimarães, P.B., 2010. *Estudo do campo de temperatura obtido numericamente para posterior determinação das tensões residuais numa junta soldada de aço ASTM AH36*. Doctoral thesis, Universidade Federal de Pernambuco, Recife.
- Marques, P.V., Modenesi, P.J., Bracarense, A.Q., 2009. *Soldagem: Fundamentos e Tecnologia*. UFMG, Belo Horizonte, 3th edition.
- Hibbit, K. & S. Inc., 2012. *Abaqus/CAE User's Manual*. Dassault Systèmes, Rhode Island.