

Modelling simulation of the thermal field for the heat transfer in environments with discontinuities

B Dumitru, I Severin and G M Dumitru

Department of Materials Technology and Welding, Faculty of Engineering and Management of Technological Systems, University “Politehnica” of Bucharest, Splaiul Independentei, no. 313, RO-060042, Bucharest, Romania

E-mail: bogdan_dmt@yahoo.com

Abstract. The paper proposes a simulation model made using the method of finite element analysis for the case of thermic transfer by infrared thermographic control with impulses for specific sample parts with induced defects. Results showed a correlation between the finite element analysis and the images obtained by infrared thermography with a maximal error of 5-6%. This correlation proves the viability of the model and could represent the basis for the developing of a working instrument for non-destructive testing of parts that are conditioned or reconditioned by thermal spraying technology.

1. Introduction

The conditioning and reconditioning process by thermal spraying is complex and involve a multitude of parameters [1, 2]. Any deviation of these parameters can cause the failure of the process, that is non-conform properties for the layers deposited by thermal spraying. The test samples can at most attest the validity of the proposed thermal spraying procedure, but they cannot ascertain that the parts conditioned or reconditioned by thermal spraying are on the same level, non-conformity risks being rather significant.

The analogies of type model – reality are important instruments for the study of different phenomena taking place in environments with discontinuities [3, 4]. The present paper proposes the finite element analysis of the thermic transfer in case of the infrared thermographic control with impulses for test samples covered by thermal spraying technologies, samples that have induced shape and dimension defects. The finite element analysis of the test samples or of the real parts, correlated with the images obtained by infrared thermographic control with impulses has as objective the proposal of a working instrument for the non-destructive control of the parts conditioned or reconditioned by thermal spraying.

2. Modelling with the method of finite element analysis

Finite element analysis of the thermic transfer phenomena is a complex process containing a geometric model, a numerical model with finite elements and the software of software packages used to solve the problem. In order to simulate the thermic field for the heat transfer in environments with discontinuities, we have chosen ANSYS 14.5, software allowing the usage of complex mathematical models with multiple analysis possibilities.

The model proposed for the finite element analysis of the thermic field in case of heat transfer in environments with discontinuities uses the following calculus hypothesis [5]:

- Material hypothesis;



- The dependence of the thermo-physical properties of the temperature;
- The existence of thermic exchange with the environment by radiation;
- The existence of thermic exchange with the environment by convection;
- Perfect contact between the base material and the deposited layer.

The geometric model used by the finite elements analysis of the heat transfer in environments with discontinuities is presented in Figure 1 and corresponds to test samples P1, P2 and C1, with defects induced artificially for the purpose of the study. The geometric configuration of the used model is presented in Figure 1 with the appropriate meshing volumes in Figure 2.

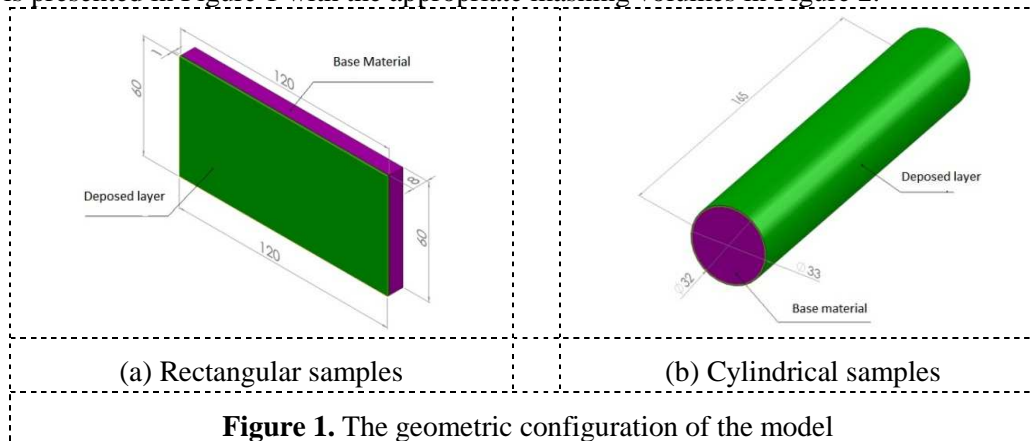


Figure 1. The geometric configuration of the model

The analysis with finite elements of the thermic transfer in environments with discontinuities used the software module *Transient Thermal* of ANSYS software, the main meshing element being the element SOLID227. To simulate the losses by radiation during the heat transfer process was used the overlaying on the surfaces that emit radiation of elements of type SURF152. The superficial emission coefficient is considered 0.5 and the constant value Stefan-Boltzmann is $5,667 \text{ E-}8$.

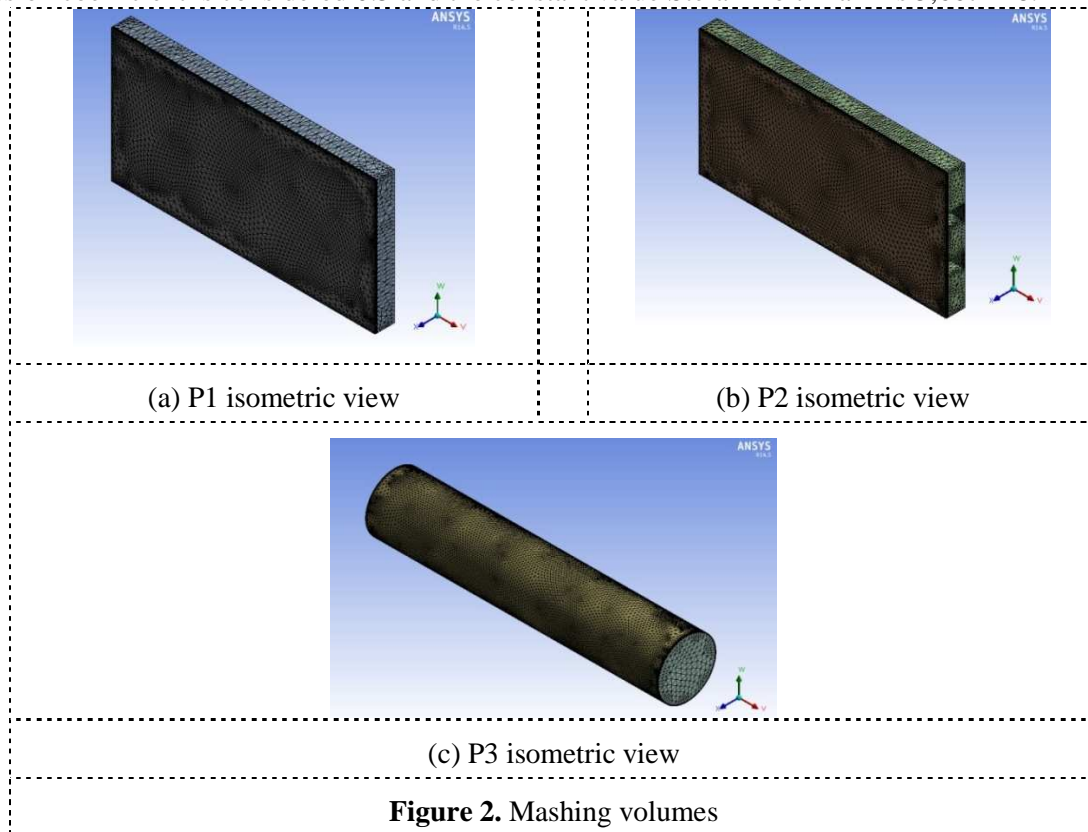


Figure 2. Meshing volumes

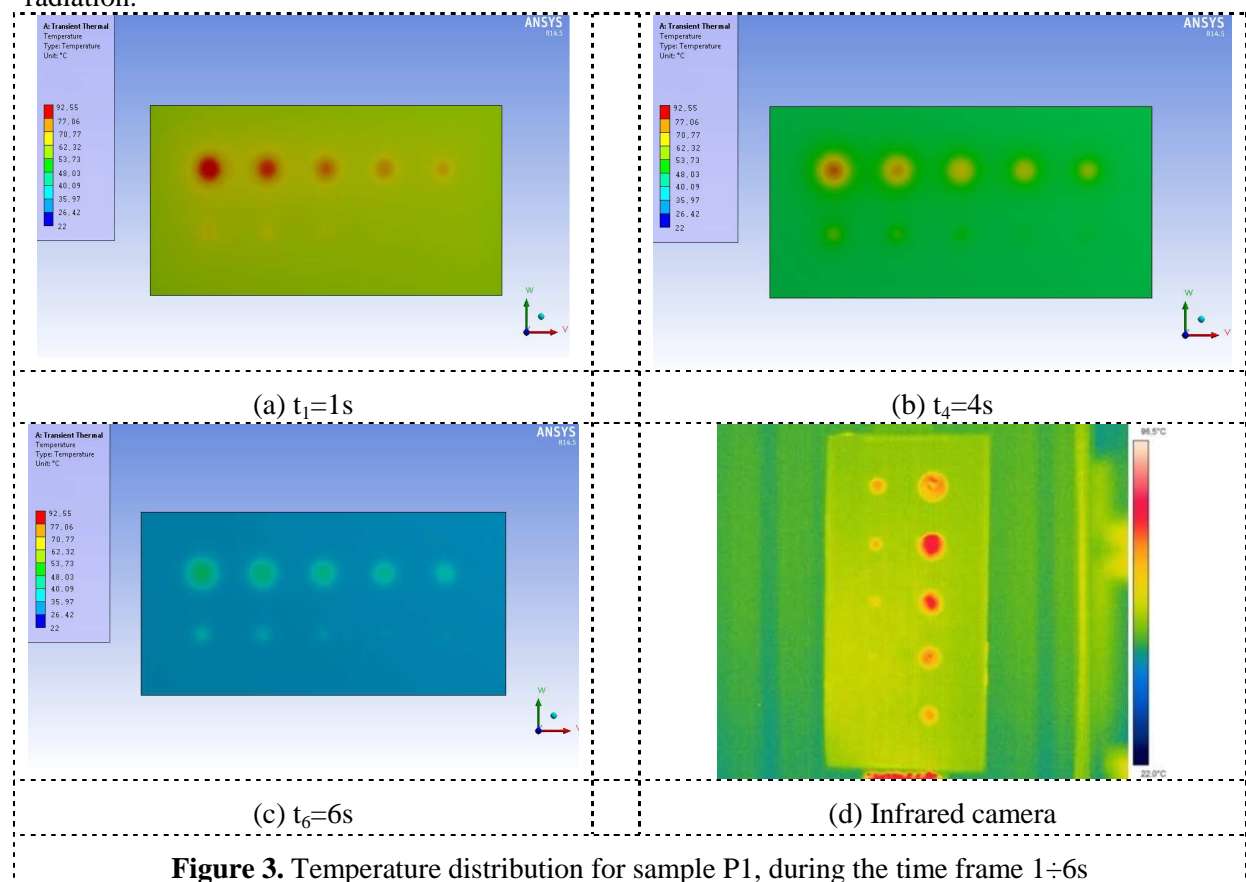
The thermal analysis in transient regime is based on the particularity that the values of the temperature and / or thermic flow loadings vary over time, this variation being given by an equation or a loading curve with finite number of steps [6].

Table 1 presents the material and loading parameters used in the analysis process.

Table 1. Material and loading parameters used in the analysis

Material properties	Thermic conductivity for deposited layer	45 W/m·°C
	Thermic conductivity for base material (test samples P1, P2)	60.5 W/m·°C
	Thermic conductivity for base material (test sample C1)	55.5 W/m·°C
Loads	Emissivity within environment	0.50
	Thermic flow	300 W/cm ²
	Temperature of the environment	22 °C
Stefan Boltzmann constant		5.667 E-8 Wm ⁻² K ⁻⁴

Figure 3 lists the results obtained by the analysis of sample P1, having as induced defects holes of different diameters filled with epoxy resin. The thermic field was recorded at 1 second intervals. The maximum temperature observed during the thermographic process was 92.55°C. As time passed was observed that the extremities start to participate at the heat exchange with the environment by radiation.



The model of the finite element analysis presented in figure 3.a) is almost identical with the thermographic image presented in figure 3.d). The median values of the temperatures are 42.44°C for the infrared thermographic examination with impulses and 42.01°C for the finite element analysis, which leads to a model error of 1.01%. The temperature evolutions for sample P1 presented in Figure

4 show that the model chosen follows closely the real sample behavior, thus the model being considered adequate for the situation.

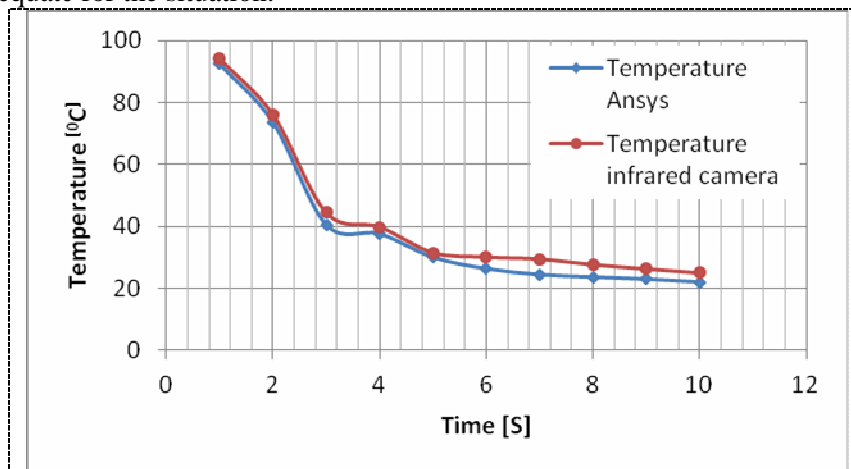


Figure 4. Temperature evolution on the surface of the sample P1

Figure 5 presents the results for sample P2, in form of isotherms of the transient thermic field, in frontal view. The sample P2 has induced as defects two channels that are filled with epoxy resin, one with constant depth, and one with variable depth. The thermic field was recorded at 1 second intervals. The maximum temperature reached during the thermography was 95.19°C. The heat exchange with the environment by radiation is made on the posterior side and through the lateral sides. This can be observed graphically as time is elapsing, by gradual heating of the base materials and the cooling of the deposited layers [6].

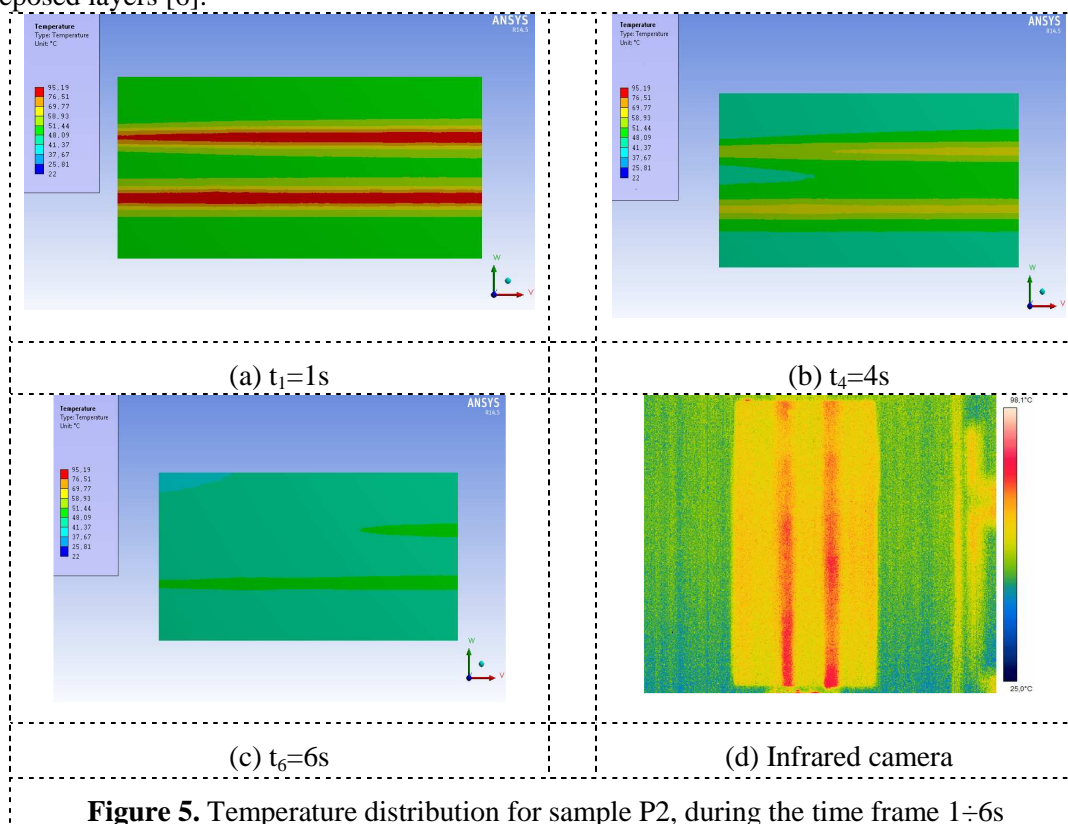


Figure 5. Temperature distribution for sample P2, during the time frame 1÷6s

A minimum difference between the results obtained by thermography and the finite element analysis was observed. The average temperature value was 53.06°C for the examination with infrared

camera and 49.81°C for the finite element analysis. Thus, the error of the finite element, for the medium temperature, is 6.12%.

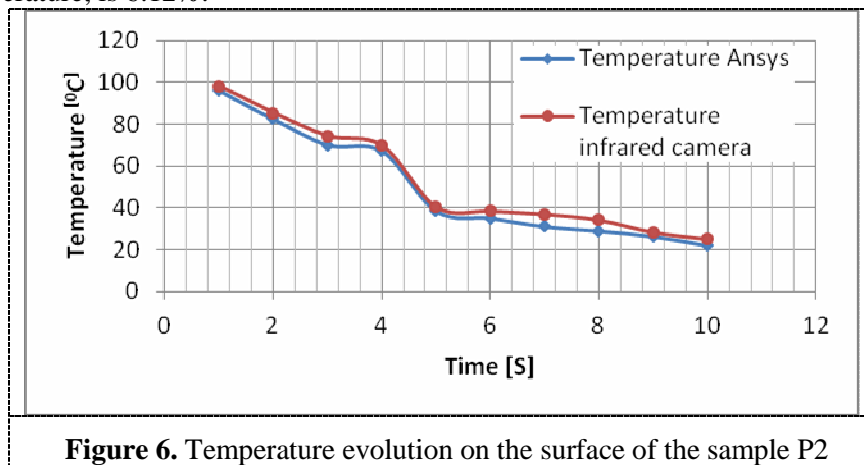


Figure 6. Temperature evolution on the surface of the sample P2

Figure 7 lists results for sample C1, as isotherms of the transient thermic field, in isometric view. The thermic field was recorded at 1 second intervals. The maximum temperature observed during thermographic control process was 60.21°C .

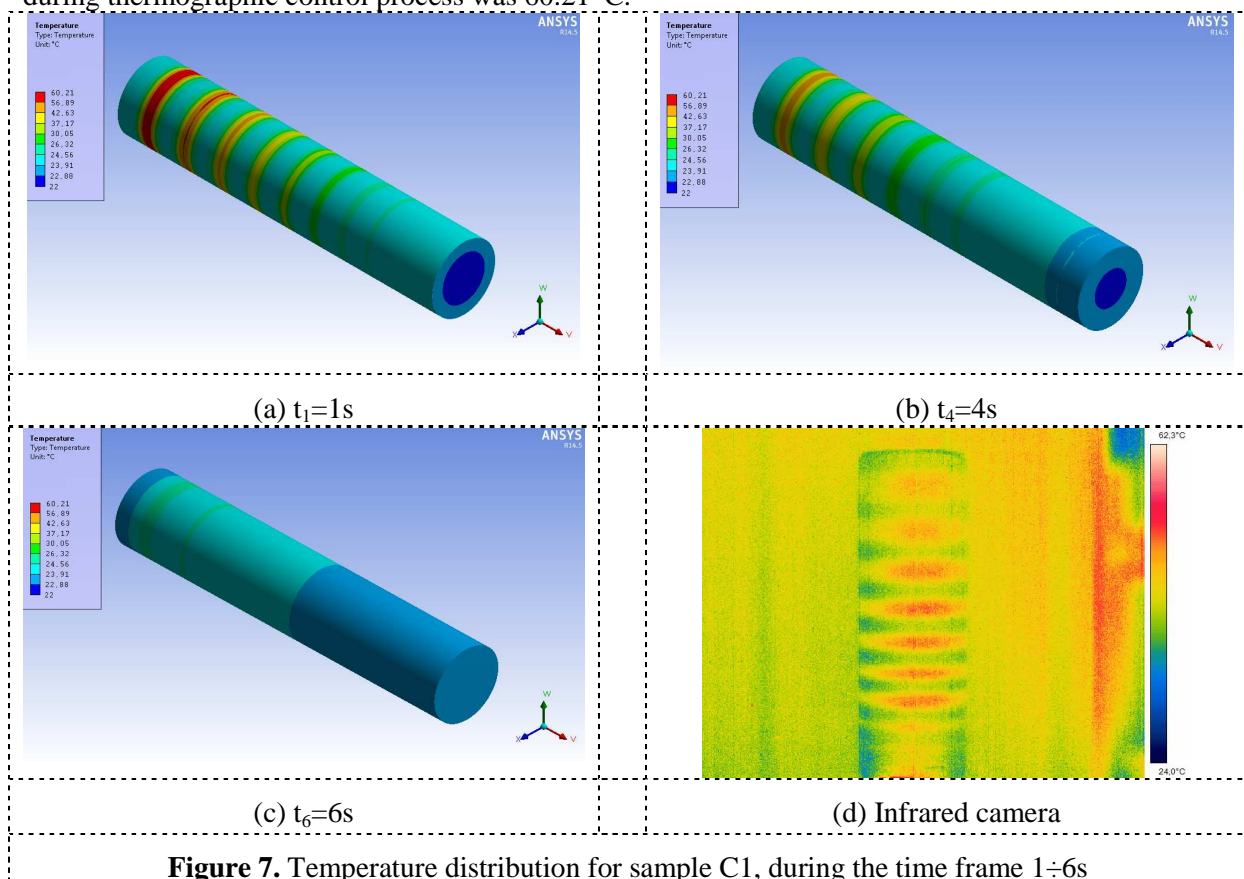


Figure 7. Temperature distribution for sample C1, during the time frame 1÷6s

It was observed a minimum difference between the results obtained by thermography and the finite element analysis. The medium value of the temperature is 36.38°C for the examination with infrared camera and 34.20°C for the finite element analysis. Thus, the error of the finite element, for the medium temperature, is 5.99%.

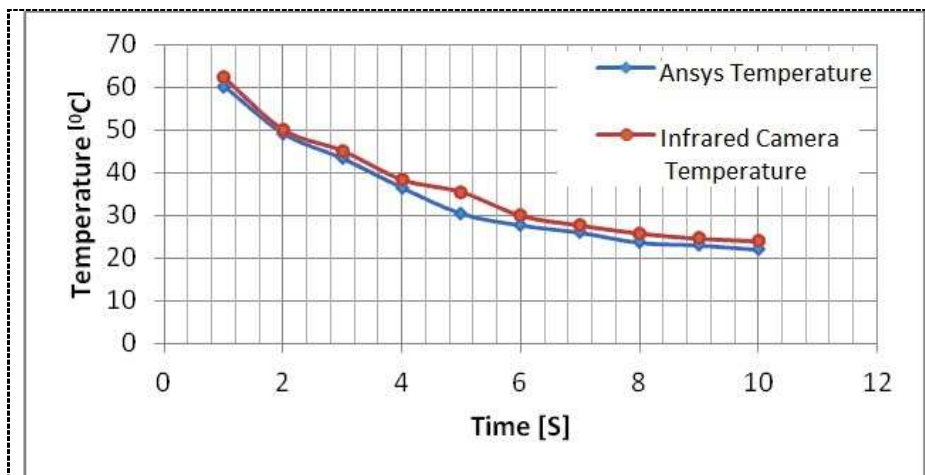


Figure 8. Temperature evolution on the surface of the sample C1

3. Conclusions

The finite elements analysis of the thermic transfer process for the infrared thermographic control process with impulses emphasises for the studied cases the following conclusions:

- The righteousness of the model with reference to the circular defects of different shapes and dimensions on the surface of the base material or on the interface deposited layer by thermal spraying – base material;
- The finite element model fidelity is acceptable (maximum 5-6% error);
- In case of defects with dimensions comparable to the part itself, the heat transmission is tridimensional, which makes the images for defects that are close to be slightly blurred;
- In order to obtain a better fidelity with cylindrical samples, the usage of a uniform circumferential heating system is recommended.

4. References

- [1] Pawlowski L, 2008 *The science and engineering of thermal spray coatings*, Ed.2 (New York J. Wiley & Sons)
- [2] Dumitru G M, Constantin R, Dumitru B and Mihăilescu A, 2013 *Reconditioning and repairing of products* (Bucharest - Printech Editure)
- [3] Mihai A, 2009 *Nondestructive Examinations* (Bucharest - Printech Editure)
- [4] Ghelmez M, Lazar B, Toma C, Dumitru B and Slavnicu E, 2006 Bulk and Surface Ultrasonic Waves Generated in Metallic Solids by Pulse Shaped Sources, *Rev.Chim.* **57** P722
- [5] Nitoi D, Dumitru G M, Dumitru B, Mihăilescu A and Cismas S C, Reconditioning by Metal Deposition Technology - Temperature Finite Element Modeling, 2014 *Advanced Materials Research* **09 Vol. 1029**, P130-133
- [6] Botez S C, Dina V C, Dumitru G M, Dumitru B, Iacobescu G, 2014 Characterization of Metallic Layers Deposited through Thermal Spraying, *UPB Sci. Bulletin, Series D, Vol. 76*