Investigation of the Temperature Distribution and Residual Stresses in Welded Structures

Jiří Černý*[1]

[1] ČVUT v Praze, Fakulta strojní, Ústav mechaniky, biomechaniky a mechatroniky, Technická 4, 166 07 Praha 6, Česká republika

Abstract

A numerical model of a welded plate was created in ABAQUS software. The distribution of temperature depending on time was compared with experimental data. The stainless steel plate (type 321) was welded by shield metal arc (SMAW) and tungsten inert gas (TIG) welding methods. The temperature on the surface of the plate was recorded by several thermocouples and thermal imaging camera.

Key-words: Welding; ABAQUS; Plate; Numerical simulation; FEM; AWI Plug-in

1. INTRODUCTION

Numerical simulation of a welding process is a complex issue and it has been a major topic in welding research. There are many methods to describe a welding process in ABAQUS software, e.g. articles [1] and [2]. This paper describes creating a numerical model of welding plate in ABAQUS by using the Abaqus welding interface (AWI) plugin. Thermal analysis of this model was done and numerical results of the thermal analysis were validated with experimental data. Thus the temperature distribution in the whole welded plate was obtained.

2. EXPERIMENT

The plate was joined by a double-V butt weld. The size of the plate was 350 x 324 x 20, the scheme of the plate is illustrated in figure 1.

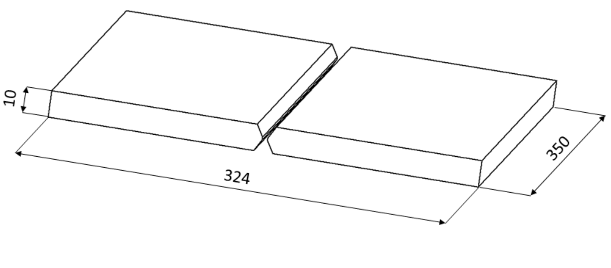

Fig. 1. *Scheme of welded plate with dimensions.*

The base material of the plate was stainless steel 1.4541 (type 321). The plate was welded by two methods. First four beads were welded by TIG method and the other beads were welded by SMAW method (fig. 2). Twelve beads were made in total.

*Kontakt na autora: Jiri.Cerny2@fs.cvut.cz

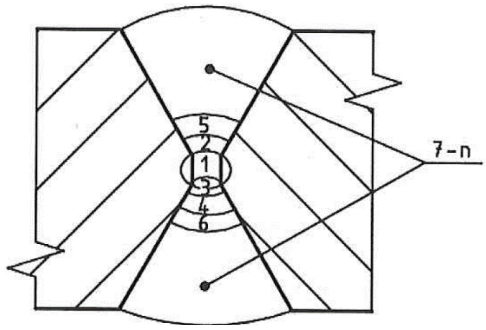

Fig. 2. *Structure of welding beads.*

Welding process was recorded with five thermocouples and thermal imaging camera. Thermocouples were installed into small hollows, which were drilled into the one side of the specimen (fig. 3). Thermocouples were in contact with the specimen surface only when the specimen side with hollows was on a top. Thermal imaging camera was recording surface temperature during whole welding process.

Fig. 3. *Installation of thermocouples on specimen surface.*

3. NUMERICAL SIMULATION

Numerical analysis was made in ABAQUS software. The dimensions of the plate model were created according to the specimen. Base and filler metal were created as one part with uniform material properties. The model was partitioned into several cells. More precisely, the filler metal was divided into twelve beads and each one of beads was divided into 28 chunks (fig. 4).

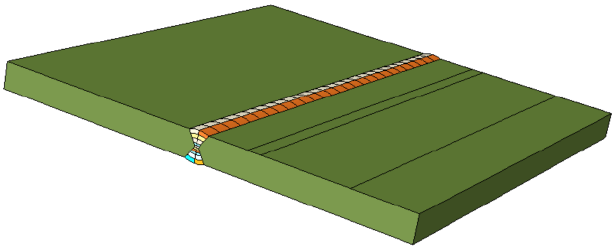

Fig. 4. *The Partition model, green is base metal, each one of beads is colored and is divided into 28 chunks.*

All elements which represent filler material were deactivated in the first time step. Boundary condition (heat transfer) was applied on elements which are in contact with the first chunk in the second step. The first chunk was reactivated and boundary condition was simultaneously deactivated in the third step. The second and the third steps were repeated for each chunk. There are 684 steps in the model in total. Time of each step was adjusted according to travel speed of a torch. The deactivation and reactivation were made by using MODEL CHANGE in the INTERACTION MODUL. The boundary condition magnitude was set to a melting temperature of the material 1400 °C [3].

The predefined initial temperature for base metal was set to 24 °C and for filler metal was set to 1400 °C. Surface film condition and surface radiation were set in each step. The material emissivity was set to 0.9. Film coefficient was adjusted according to recorded temperature by thermocouples. Ambient temperature was set to 24 °C. Creating steps and setting parameters were made via AWI plug-in, which is available on SIMULIA official website.

Temperature dependent material properties like conductivity, density and specific heat were obtained from the database of materials properties for stainless steel 1.4541 (type 321) [3].

4. RESULTS OF CALCULATION

The temperature distribution in the whole plate during welding process was calculated. The temperature distribution, when first bead is being added, is showed in figure 5.

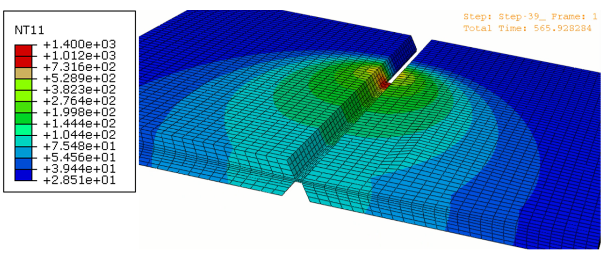

Fig. 5. *The temperature distribution, when first bead is being added.*

The film coefficient was adjusted to 25 [W/m^{2}K]. The temperature development at the local point, where one of the thermocouples was installed, is presented in figure 6. The green line is the recorded temperature from experiment and the red line is calculated temperature.

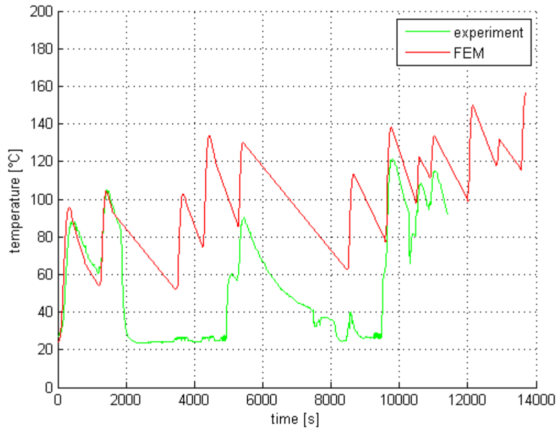

Fig. 6. *The temperature development at place, where one of the thermocouples was installed.*

5. CONCLUSION

The film coefficient was adjusted as constant, the value of constant was 25 [W/m^{2}K]. Calculated temperature distributing in plate fits recorded temperature by thermocouples, but better correspondence between numerical result and experiment data would be probably obtained, if the film coefficient was temperature dependent. Temperature distribution will be used to calculate residual stresses in the welded plate.

References

[1] Wenya Li et al. "Numerical Simulation of Linear Friction Welding Based on ABAQUS Environment: Challenges and Perspectives". en. In: *Journal of Materials Engineering and Performance* 23.2 (Feb. 2014), pp. 384–390. ISSN: 1059-9495, 1544-1024. DOI: 10.1007/s11665-013-0776-8. Available at: https://link.springer.com/article/10.1007/s11665-013-0776-8 (visited on 09/11/2017).

[2] S Fricke, E Keim, and J Schmidt. "Numerical weld modeling — a method for calculating weld-induced residual stresses". In: *Nuclear Engineering and Design* 206.2 (June 2001), pp. 139–150. ISSN: 0029-5493. DOI: 10.1016/S0029-5493(00)00414-3. Available at: http://www.sciencedirect.com/science/article/pii/S0029549300004143 (visited on 09/11/2017).

[3] *Data sheet material: 1.4541*. Lucefin. Available at: http://www.lucefin.com/wp-content/files_mf/1.4541en.pdf (visited on 06/31/2017).