SIMULACIJA ZAVARIVANJA KORIŠTENJEM „ROĐENJA I SMRTI“ ELEMENATA U PROGRAMU ANSYS I USPOREDBA ZAOSTALIH NAPREZANJA

WELDING SIMULATION USING ELEMENT BIRTH AND DEATH TECHNIQUE IN ANSYS SOFTWARE AND RESIDUAL STRESS COMPARISON

Zvonimir Kovačev, Pejo Konjatić, Darko Damjanović, Dorian Turk
Strojarski fakultet u Slavonskom Brodu, Sveučilište J. J. Strossmayera u Osijeku

Ključne riječi: simulacija zavarivanja, rođenje i smrt elemenata, zaostala naprezanja
Key words: welding simulation, element birth and death, residual stress

Sažetak:
U ovome radu prikazana je simulacija zavarivanja korištenjem postupka rođenja i smrti elemenata u programskom paketu ANSYS Workbench. Na osnovu odabranih parametara zavarivanja, u diskretiziranom 3D modelu cijevi i ploče postavljen je unos topline za svaki dio kružnog zavara u vremenskim intervalima. Zavareni spoj je postupno formiran kroz vremenske intervale rađanjem elemenata koji opisuju postupno nanošenje dodatnog materijala. Nakon simulacije zavarivanja, dodatnom termalnom simulacijom analizirano je hlađenje spoja na zraku. Rezultati temperaturnih polja tijekom vremena hlađenja u zavarenom spoju korišteni su za analizu zaostalih naprezanja. Za usporedbu zaostalih naprezanja korišten je zavareni model s identičnim postavkama, ali bez primjene rađanja i umiranja elemenata kako bi se utvrdio njen utjecaj.

Abstract:
In this paper, the welding process has been simulated using element birth and death method in program ANSYS Workbench. Starting welding parameters were chosen and heat input was implemented in discretized 3D model of pipe and plate for every part of circular weld in defined time intervals. Circular weld was gradually formed for every time interval using element birth option to simulate filler material deposition. After welding simulation, extra thermal simulation was used to analyze cooling effects at room temperature on stagnant air. Temperature field results during cooling time of the model were used to analyze residual stresses in the welded part. For comparison of residual

stresses, an identical numerical model was used, only without using element birth and death to see the difference.

1 INTRODUCTION

Manufacturing and assembly processes require accuracy and precision when it comes to welded parts. During weld formation, different problems may arise in form of inclusions, sticking, geometric and dimension errors and cracks. Some of those problems depend on welding method and operator competence. But some of the problems, especially cracks, occur due to the residual stresses induced by the heat input. For best welded joint, numerical simulations are widely used to get the insight into the welding parameters and joint behavior during and after the welding process. To perform a welding simulation, in this case a simple tube and plate are chosen to be welded together using circular weld. Figure 1 shows considered model with predefined parts of the welded joint that will be used in the analysis.

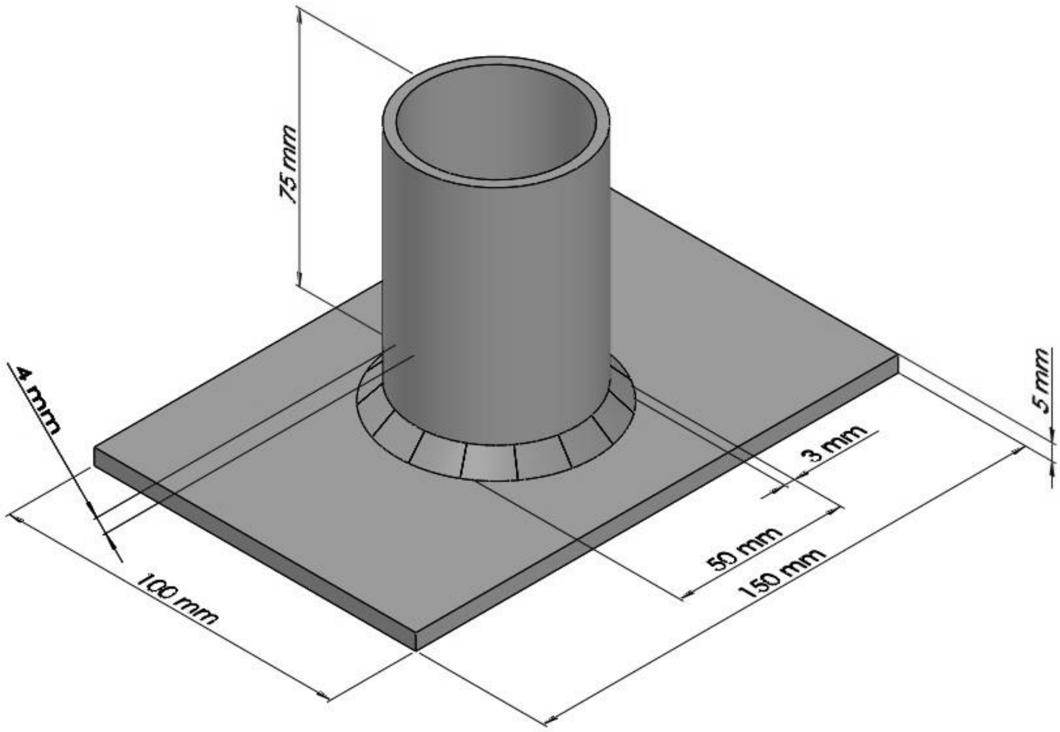

Figure 1. Tube and plate model with corresponding dimensions

2 ANALYSIS FORMULATION AND SETTINGS

To perform a welding simulation, parameters for the heat input are needed and transient thermal analysis is used [1]. In this case, it was assumed the welding method is manual arc welding [2]. For that purpose, needed current is 90 A and voltage 20 V. Figure 1 shows the weld joint which consists of 15 subparts (filler material depositions) that will gradually form during simulation using element birth and death option. As each subpart forms, an internal heat generation boundary condition is applied to it over a 4 seconds period in every load step to approximate welding process with moving heat source. By calculating volume for each subpart (180 mm^3), internal heat generated during every second is 10 W/mm^3. The weld size is chosen to be 4 mm. Also, a free convection is defined on model surfaces, with convection coefficient of 5 W/mm^{2}°C of stagnant air. Bonded contact is defined between all subparts of the weld joint, and at the start of the simulation, all elements have been deactivated (dead) by using element death option [2]. Welding begins as the first subpart forms by using element birth option, and simulation lasts 60 seconds until all subpart are activated (alive). The total duration of welding is 60 seconds, after which cooling on stagnant air continues.

The material for this simulation is not specifically considered, so the standard structural steel linear elastic material model from ANSYS engineering database is selected [1]. It's thermal conductivity is 60,5 W/m°C and specific heat 434 J/kg°C with the yield strength of 250 MPa. Liquid phase of the material and it's effects are not considered [3].

Figure 2 shows discretized 3D FE model using hexahedral linear elements. Linear elements are used due to interpolation deviations giving unrealistic temperature results [2]. The mesh consists of 17211 elements and 25577 nodes.

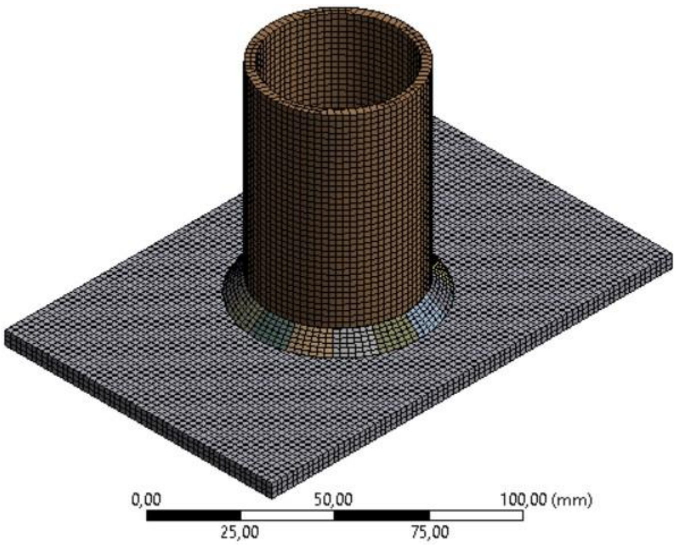

Figure 2. Simulation model with hexahedral mesh

3 RESULTS

Figure 3 shows maximal and minimal temperatures on the model during the welding simulation. After welding, the cooling effect on stagnant air is simulated during next 300 seconds. Temperature fields given in Figure 4 are from the cooling period over 300 seconds.

Figure 3 shows that the temperature rises with constant heat input during the welding. Temperatures reach over the melting point in every subpart that is being welded, thus creating a weld. Soon after the welding stops, the temperature rapidly drops and then slowly decreases to over 300°C after 300 seconds as seen in Figure 4. Also, the minimum temperature in the rest of the model starts rising due to steel conductivity. It takes 30 seconds for temperature to drop from 800°C to 500°C which is temperature parameter $t_{8/5}$.

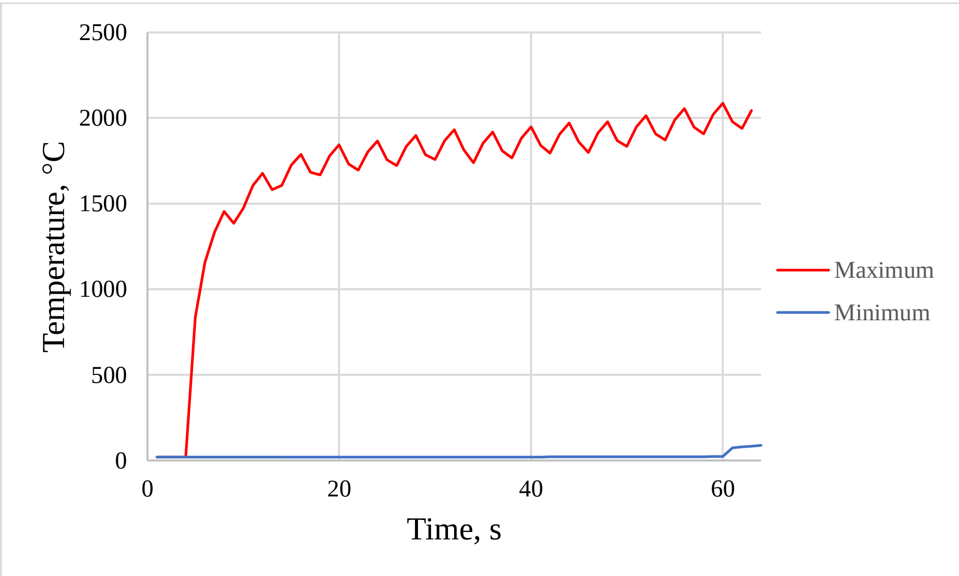

Figure 3. Temperature over time during welding

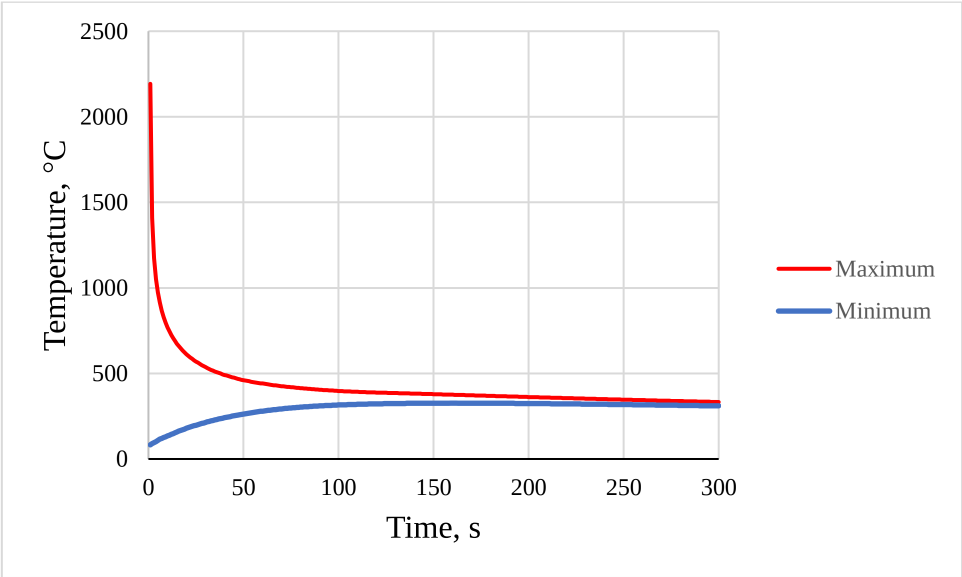

Figure 4. Temperature over time during cooling

Figures 5, 6 and 7 show temperature fields of the model for t = 1 s, t = 120 s and t = 300 s of the analysis. Over time, the temperature spreads evenly through the entire model. These temperature fields are used as starting data for transient structural analysis for obtaining residual stresses.

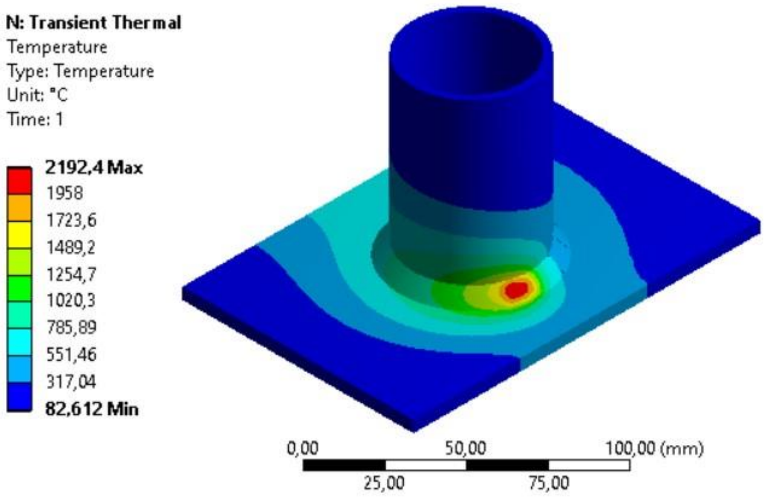

Figure 5. Temperature field of the model for t = 1 s

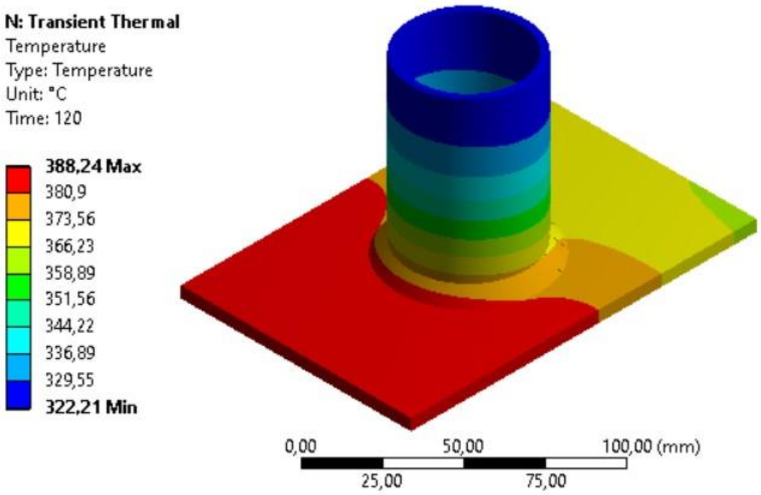

Figure 6. Temperature field of the model for t = 120 s

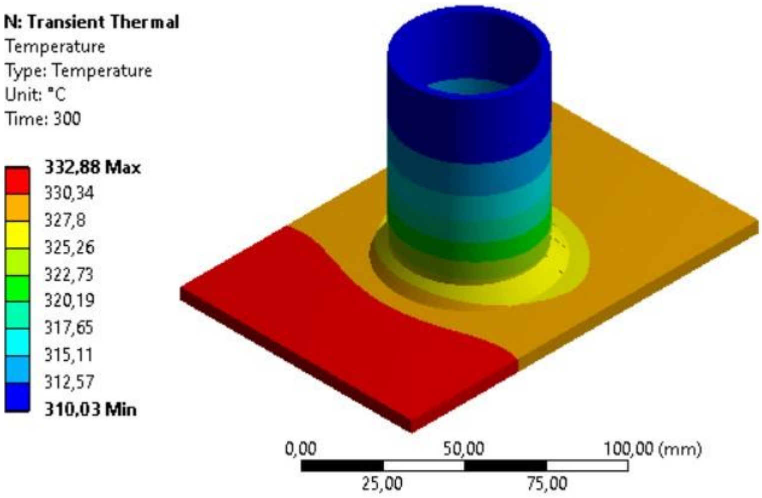

Figure 7. Temperature field of the model for t = 300 s

Similar to first model, the second model for comparison was made but without using element birth and death method. Comparison of the maximum and minimum temperatures during cooling was made, as shown in Figure 8. It can be observed that in the second model, it takes longer to cool from the same maximal temperature than in the first model and the minimum temperatures are identical. According to those results, transient structural analysis was used to show residual stresses (equivalent von Mises stress) in the both models (Figure 9) [4].

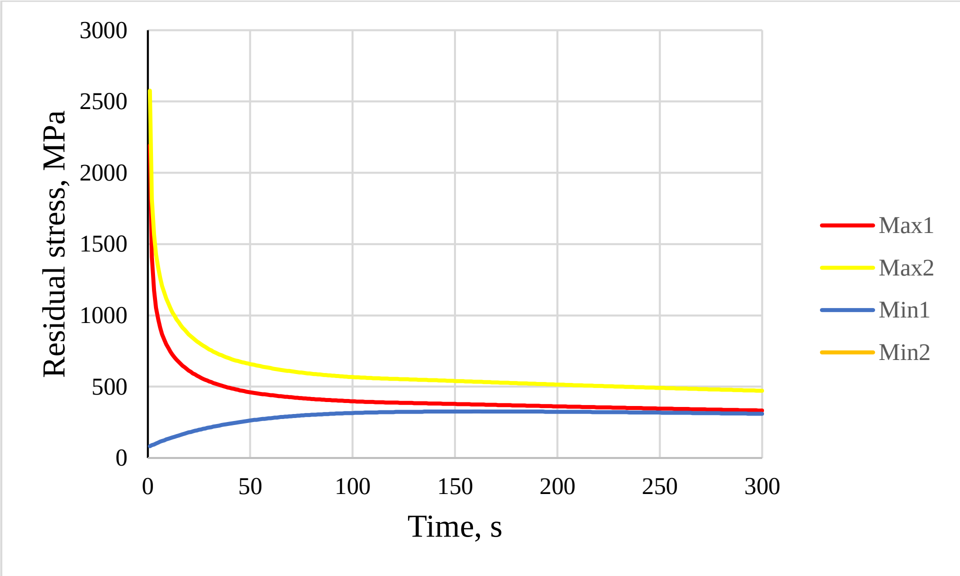

Figure 8. Maximal and minimal temperatures over time for both models

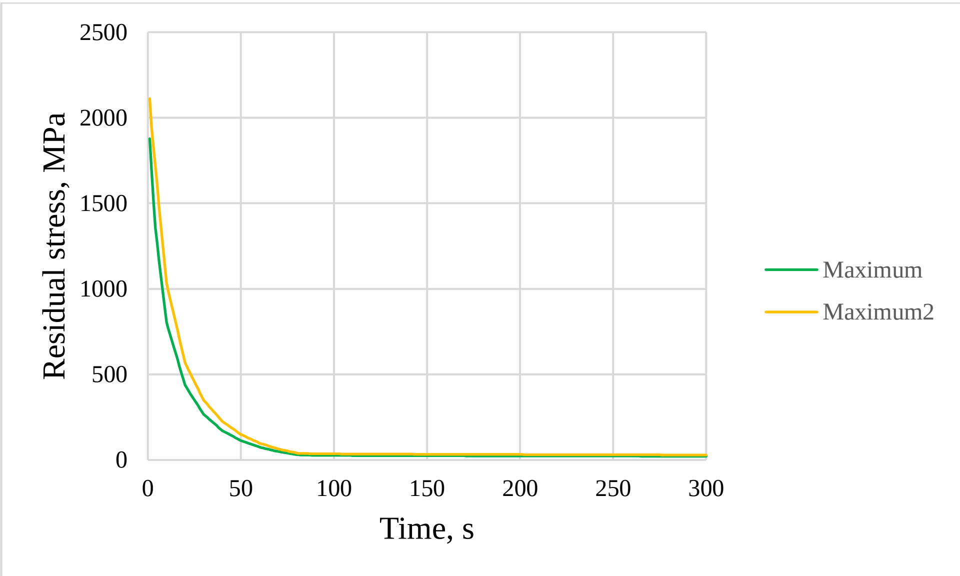

Figure 9. Maximum residual stress over time for both models

Since temperatures decrease over time, the residual stress is high at the beginning but after 50 seconds of cooling it rapidly drops and continues to decrease in both models. Figure 10 shows a model with residual stress field for t = 300 s of the simulation. For considering residual stresses in the weld joint only, remote displacement was used as the boundary condition [1]. Various geometrical constraints and material nonlinearities can affect residual stress but are not considered in this paper. [2]

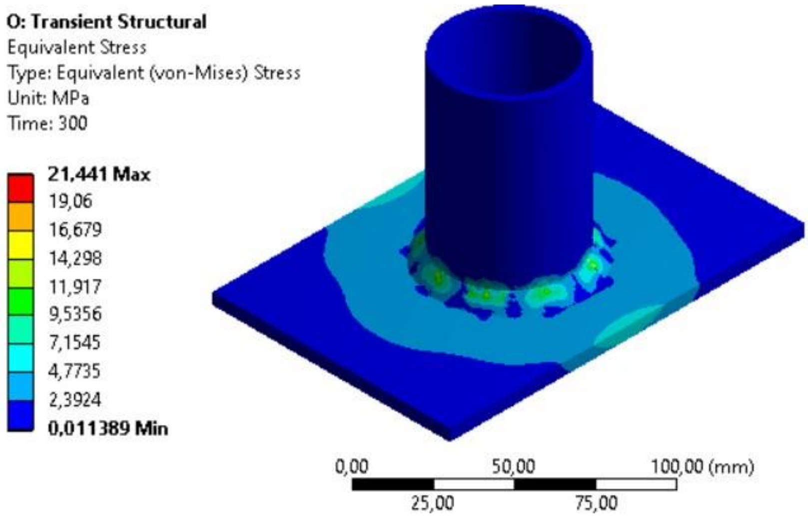

Figure 10. Residual stress field for t = 300 s

4 CONCLUSION

This paper presents FE model for numerical simulation of welding of tube and plate. Simulation was carried out as coupled thermal and mechanical analysis with element birth and death technique for simulation of filler material deposition. Results of the first simulation showed that by using element birth and death temperature drops quicker than in the other model due to less heat input as element birth and death slowly add elements during the welding simulation. After reviewing residual stresses, it shows that eventually both models reach lower temperature, where material relaxes, and residual stresses decrease. In this paper, microstructure changes during thermal exposure were not considered, which may have significant influence on the results. Residual stress also depends on the geometry of the weld. Further research and experiment considering residual stresses may be conducted to collect data on real specimen.

5 REFERENCES

[1] ANSYS Release 17.0, Ansys Workbench Help: *Ansys Documentation*. Canonsburg , ANSYS Inc., 2015.

[2] Stamenković, D.; Vasović, I.: *Finite Element Analysis of Residual Stress in Butt Welding Two Similar Plates, Scientific Technical Review,* Vol. 59, 2009, pp 57-60

[3] Capriccioli A.; Frosi P.: *Multipurpose ANSYS FE procedure for welding processes simulation*, Fusion Engineering and Design, Vol. 84, 2009, pp 546-553

[4] Lee, Huei-Huang: *Finite Element Simulations with ANSYS Workbench 17 – Theory, Applications, Case Studies.* Mission; SAD, SDC Publications, 2017., 596 str.