Aleksandar Grbović[1], Aleksandar Sedmak,[1,a], Abdulgasem Sghayer[1], Ivana Ivanović[2]

FINITE ELEMENT SIMULATION OF LASER BEAM WELDING

SIMULACIJA LASERSKOG ZAVARIVANJA METODOM KONAČNIH ELEMENATA

Original scientific paper / Originalni naučni rad

The paper was published in its original form in the Proceedings of the 31st Conference with international participation "Welding 2020" held in Kladovo, Serbia from 13 to 16 October 2021.

Paper received / Rad primljen:
Septembar 2021.

Paper accepted / Rad prihvaćen:
May 2022.

Author's address / Adresa autora:

[1]Faculty of Mechanical Engineering, University of Belgrade, Serbia

[2]Innovation Center of the Faculty of Mechanical Engineering, Belgrade, Serbia

[a] **E mail:** *asedmak@mas.bg.ac.rs*

Keywords: Finite Element Method, laser welding, numerical simulation

Ključne reči: Metoda konačnih elemanata, lasersko zavarivanje, numerička simulacija

Abstract

Joining process play a significant role in producing lightweight constructions. Specifically, the use of Aluminum alloys for structural components or body airplane panels is one of the major challenges for joining because AA6xxx are very susceptible to hot cracks forming during fusion welding. As laser beam welding is increasingly used for welding aircraft components, special techniques are required to avoid hot cracks in weld seams. The impact of the process characteristics, e.g., the temperature field, the temperature gradients, or the molten pool geometry, can be determined by using numerical methods, e.g. Finite element method, as used here. Results presented here, demonstrate strong possibilities of numerical simulation of laser beam welding.

Rezime

Proces spajanja igra važnu ulogu u razvoju lakih konstrukcija, posebno u slučaju aluminijumskih legura i njihove primene za panele koji se ugrađuju u avione. Aluminiumska legure AA6xxx su vrlo osetljive na vruće prsline pri zavarivanju topljenjem. Stoga primena laserskog zavarivanja zahveta posebne mere i tehnike rada da bi se spečila pojava topliha prslina. Uticaj karakteristika laserskog zavarivanja, kao što su temperaturna polja i gradijenti, uz geometriju rastopljenog metala šava, mogu da se odrede primenom numeričkih metoda, npr. metoda konačnih elemenata, koja je ovde korišćena. Prikazani rezultati ukazuju na velike mogućnosti numeričke simulacije laserskog zavaraivanja.

1. Introduction

The aircraft production industry has a high demand for lightweight structures. In this context, the joining process plays a significant role in producing the lightweight construction. Aluminum alloys AA6xxx are susceptible to hot cracks during fusion welding. As laser beam welding is increasingly used for welding aircraft components [1], special techniques are required to avoid hot cracks. The impact of process features, such as temperature field and gradients, or the molten pool geometry, can be determined by using numerical model. Here, we demonstrate possibilities of numerical simulation of laser beam welding using plate with one stringer as the case study. All the parts in the model were assigned the same material properties of Al6156-T6 [2]. For simplicity, no temperature dependent properties were considered and material was considered to be linear elastic. Welding process simulation involves two steps: (1) Transient thermal and (2) Thermal Stress analysis. To simulate the thermal field produced by the welding process, it is necessary to model the heat source accurately and for that purpose "Moving Heat Flux" ACT extension – available for download from ANSYS support website – must be used [3]. Also, change of convection coefficient with temperature for aluminum must be defined, as well as velocity of laser beam (5 mm/s in our case) and Gaussian heat flux constant (7.5 W/mm^2). After the thermal analysis, temperatures are used to perform thermal stress analysis. Values of strains and von Mises stress distribution at the end of the welding process simulation are then obtained.

2. Numerical simulation

Simulation of welding process for a plate with one stringer is presented. Dimensions of the stringer are the same as dimensions of four stringers used in fatigue strength assessment. Geometry and mesh is presented in Figure 1, while boundary conditions are presented in Figure 2. It is important to emphasize that all initial connections between stringer, base metal and weld lines were modeled as frictional (coefficient of friction 0.2) because these elements are not connected at all before welding.

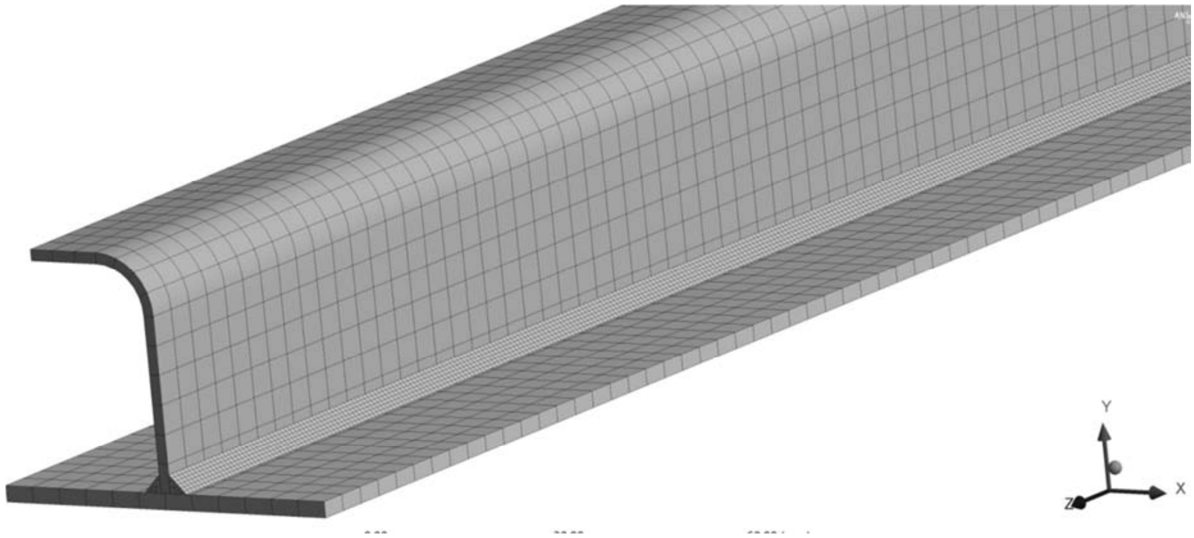

Figure 1. *Mesh of stringer, weld lines and base metal*

Slika 1. *Mreža od ukrućenja, linija zavara i osnovnog metala*

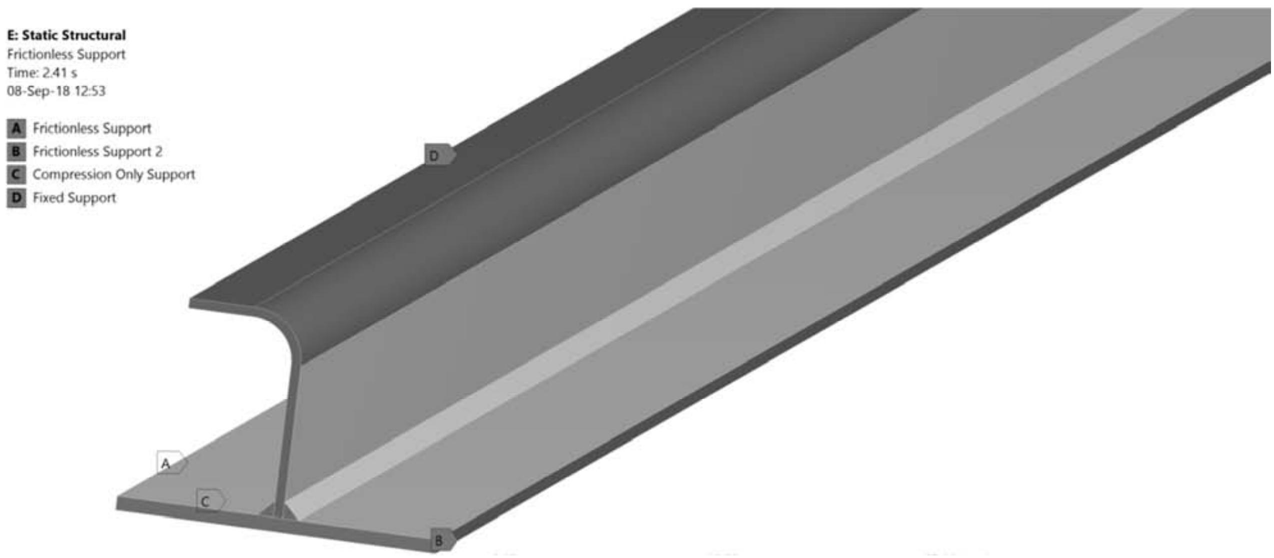

Figure 2. *Boundary conditions used in laser beam welding simulation*

Slika 2. *Granični uslovi koji se koriste u simulaciji zavarivanja laserskim snopom*

To simulate contact change, software command shown in Figure 3 was used to change type of contact from frictional to bonded (i.e. welded) when temperature reaches 150^{0}C. The contact status will remain bonded for the rest of the analysis, even if the temperature subsequently decreases.

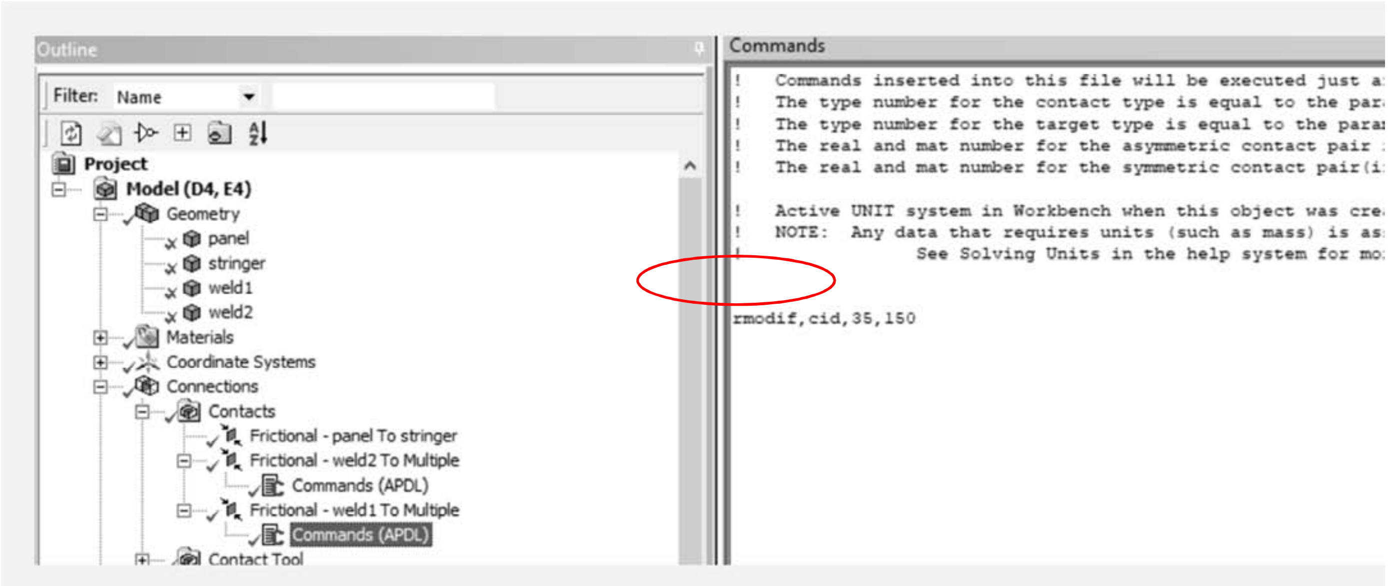

Figure 3. *Command for defining critical bonding temperature of 150 ^{0}C*

Slika 3. *Komanda za definisanje kritične temperature spajanja od 150 ^{0}C*

The same linear elastic material properties were assigned for the whole model. To simulate the temperature field due to welding process "Moving Heat Flux" ACT extension, available for download from ANSYS support website, was used. Also, change of convection coefficient with temperature for aluminum was defined (Figure 4), as well as velocity of laser beam (5 mm/s) and Gaussian heat flux constant (7.5 W/mm^2).

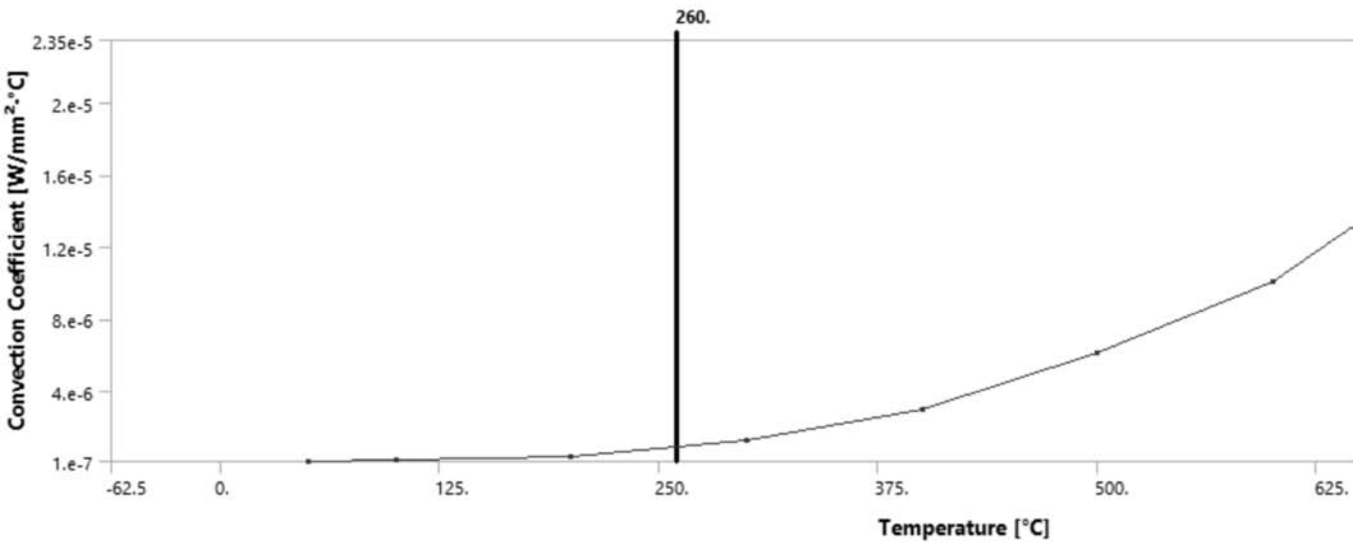

Figure 4. *Convection coefficient vs. temperature for aluminum*

Slika 4. *Koeficijent konvekcije u zavisnosti od temperature za aluminijum*

3. Results and discussion

Figure 5 shows temperature distribution during laser beam welding simulation, while Figure 6 presents heat affected zone formed during the welding process. After the thermal analysis, temperatures are used as input for static structural analysis for all time instants (Figure 7) to perform thermal stress analysis. Values of strains and von Mises stress distribution at the end of the welding process are shown in Figures 8 and 9.

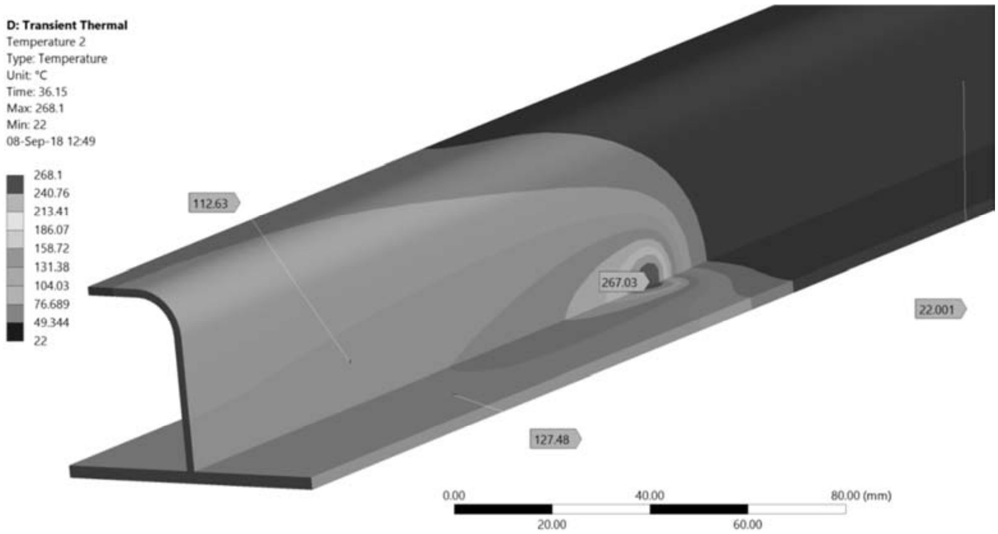

Figure 5. *Temperature distribution 36.15s after welding started*

Slika 5. *Raspodela temperature 36.15s nakon početka zavarivanja*

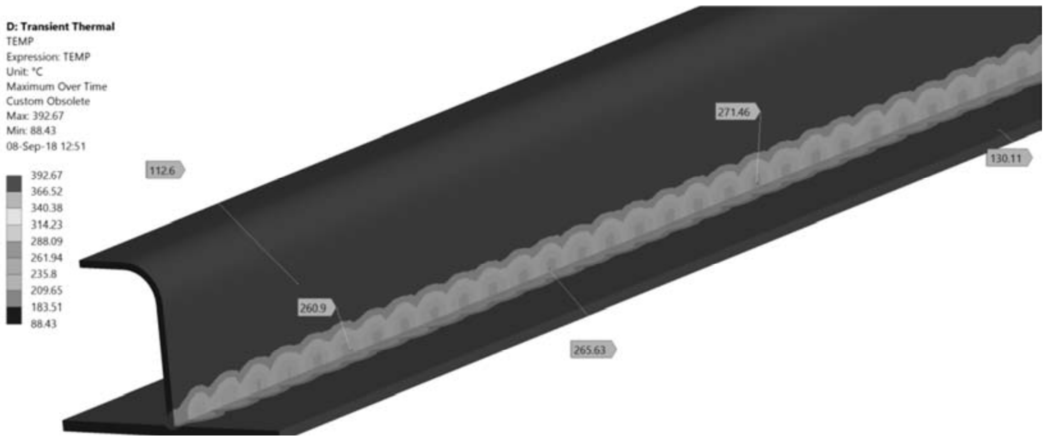

Figure 6. *Heat effected zone – HAZ*

Slika 6. *Zona uticaja toplote - ZUT*

Figure 7. *Imported temperature for static structural analysis at initial time (t=0s)*

Slika 7. *Temperature korišćene za statičku analizu konstrukcija za početno vreme (t=0s)*

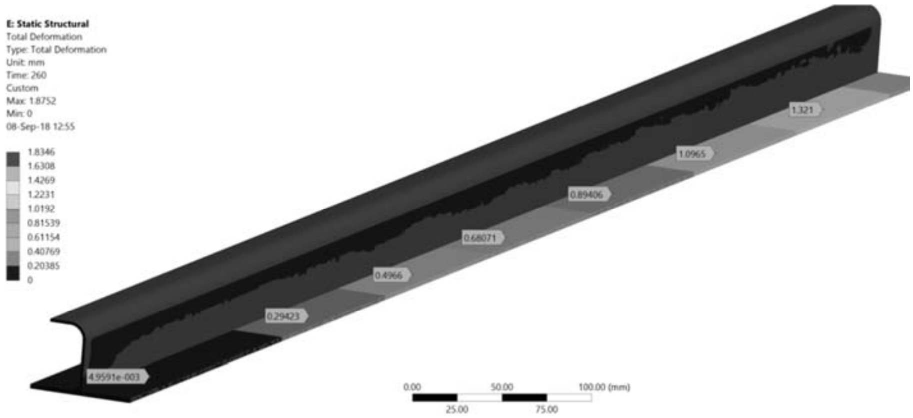

Figure 8. *Strains of base plate after welding*

Slika 8. *Deformacije osnovne ploče nakon zavarivanja*

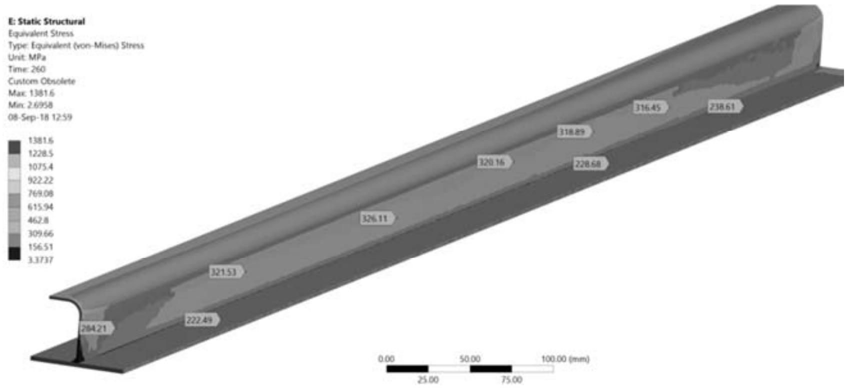

Figure 9. *Equivalent von Mises stress 20 s after welding ended*

Slika 9. *Ekvivalentni von Mizesov napon 20 s nakon završetka zavarivanja*

As can be seen in Figure 9, thermal stresses obtained in numerical simulation of laser welding were higher than 290 MPa (value representing the yield stress of Al6156-T6), indicating possibility of plastic deformation of stringer. To prevent this, heat source should have power less than 7.5 W/mm^2. In any case, it is shown that residual stresses after welding can affect fatigue strength of welded joint. This is why thermal and/or mechanical treatment should be applied for stress relief of skin-stringer plate.

4. Conclusions

Based on the presented results one can conclude that numerical analysis provides strong possibility of numerical simulation of laser beam welding, including thermal and static structural analysis.

One can also conclude that high level of residual stresses require post weld heat treatment in the analysed case.

4. Zaključci

Na osnovu prikazanih rezultata može se zaključiti da numerička analiza pruža veliku mogućnost numeričke simulacije zavarivanja laserskim snopom, uključujući termičku i statičku analizu.

Takođe se može zaključiti da visoki nivoi zaostalih napona zahtevaju termičku obradu posle zavarivanja, u analiziranom slučaju.

References / Literatura

[1] Munroe J., Wilkins K., and Gruber M., Integral Airframe Structures (IAS) - Validated Feasibility Study of Integrally Stiffened Metallic Fuselage Panels for Reducing Manufacturing Costs, NASA/CR-2000-209337, May 2000.

[2] Sghayer, A. Fatigue life assessment of damaged integral skin–stringer panels, Ph. D. thesis, Faculty of Mechanical Engineering, University of Belgrade, Serbia, 2018.

[3] http://www.cenaero.be/Page_Generale.asp?DocID=27331