Residual Stress Analysis of Laser Welding Between Output Needle and Helix

Wei Su\textsuperscript{1,2}, Wenhao Luo\textsuperscript{1,2,3,*}, Xianshan Dong\textsuperscript{1,2} and Renhuai Liu\textsuperscript{1,2,3}

\textsuperscript{1}MOE Key lab of Disaster Forecast and Control in Engineering Guangzhou 510632, China
\textsuperscript{2}Science and Technology on Reliability Physics and Application Technology of Electronic Component Laboratory, China Electronic Product Reliability and Environmental Testing Research Institute, Guangzhou 510610, China
\textsuperscript{3}School of Mechanics and Construction Engineering, Jinan University, Guangzhou 510632, China

*Corresponding author: lwhcuber@stu2019.jnu.edu.cn

Abstract. In order to obtain the welding stress of the output needle and helix, a finite element model is established and the mesh at the weld is refined to evaluate the welding residual stress. ANSYS parameter design language is used to simulate the motion load of welding heat source. Based on the exact calculation of the temperature field, the corresponding thermal units are converted into structural units to calculate the welding residual stress. Finally, the thermal stress distribution curve and residual stress distribution trend are obtained after post-treatment.

Keywords: Finite Element Analysis, Residual Stress, Welding.

1. Introduction
The residual stress of laser welding between the output needle and the helix in TWT is the key factor affecting its structural performance. Laser welding is a complex process involving heat transfer, metallurgy and mechanics, which is directly related to the quality of engineering and the safety of structure [1]. Therefore, the calculation and analysis of welding residual stress have very important practical significance [2]. This paper combines the numerical calculation and finite element analysis software ANSYS to carry on the three-dimensional real-time dynamic simulation research of welding temperature field and welding residual stress field [3-4], and puts forward the simulation analysis method of welding temperature field and stress based on ANSYS platform [5].

2. Finite element model and numerical calculation

2.1. Finite element model
The finite element solid model is shown in Figure 1. The output needle is welded with the helix through platinum solder. In the welding process, the temperature gradient of the welding seam and the welding heat affected area change greatly, and the dense mesh should be used in this part. In the area far away from the welding seam, the temperature gradient changes relatively little, so relatively sparse
grids can be used. The mesh division of the finite element model is shown in Figure 2, in which all hexahedral grids are used for welding.

Figure 1. Welding finite element calculation solid model.

Figure 2. Finite element calculation mesh division model.

2.2. Calculation of temperature field of defect-free solder joint
The simulation of welding temperature field belongs to transient thermal analysis, so this step needs to set the load step option [6]. The initial temperature field and the final temperature field during the welding heating process are shown in Figure 3 and Figure 4 respectively.

Figure 3. Initial temperature field at solder joint (22°C).
Figure 4. Temperature field of solder joint at 1s (final peak state).

2.3. Simulation calculation of the welding residual stress field of the defect-free solder joint

After the calculation of welding transient temperature field is completed and the test meets the requirements, the next step is to simulate the stress field. Then we go back to PREP7, read the temperature field model, and convert the thermal unit into structural unit. The finite element calculation results are shown in Figure 5-8. Because the stress in the Z direction is much smaller than that in the other two directions, it can be ignored and unconsidered.

Figure 5. Initial residual stress at solder joint (X-direction stress).

Figure 6. Initial residual stress at solder joint (Y-direction stress).
Figure 7. Final residual stress at solder joint (X-direction stress).

Figure 8. Final residual stress at solder joint (Y-direction stress).

The initial extreme value of residual stress and the final extreme value of residual stress at the welding spot calculated by finite element method are shown in Table 1 below:

|                          | Initial residual stress extreme (MPa) | The ultimate residual stress (MPa) |
|--------------------------|---------------------------------------|-----------------------------------|
|                          | X-direction | Y-direction | X-direction | Y-direction |
| Tensile stress           | 1.46        | 0.292       | 62.3        | 58          |
| Compressive stress       | -0.064      | -0.156      | -140        | -61.1       |

3. Simulation calculation and experimental analysis of welding residual stress field of defective solder joint

The solder joints between the output needle and the helix usually contain defects. Common defects are shown in Figure 9. There is a hollow spherical structure inside the solder joint, and the sphere is a hollow structure, and the volume accounts for about 30% of the whole solder joint. Finite element mesh division is shown in Figure 10.
Figure 9. Schematic diagram of solder joints with defects.

Figure 10. Schematic diagram of mesh division of solder joints with defects.

The finite element calculation results are shown in Figure 11 and Figure 12, which contain the initial residual stress in X direction and Y direction. In the same way, the final residual stress in X direction and Y direction of the solder joint after 60s cooling is given, as shown in Figure 13 and Figure 14:

Figure 11. Initial stress at defective solder joints (X-direction stress).
Figure 12. Initial residual stress at solder joints with defects (Y-direction stress).

Figure 13. Final residual stress at defective solder joints (X-direction stress).

Figure 14. Final residual stress at defective solder joints (Y-direction stress).

The initial extreme value of residual stress and the final extreme value of residual stress at the welding spot calculated by finite element method are shown in Table 2:
Table 2. Extreme value of residual stress.

|                        | Initial residual stress extreme (MPa) | The ultimate residual stress (MPa) |
|------------------------|--------------------------------------|-----------------------------------|
|                        | X-direction | Y-direction | X-direction | Y-direction |
| Tensile stress         | 2.78        | 0.725       | 95.7        | 88.9        |
| Compressive stress     | -0.624      | -0.847      | -255        | -123        |

4. Conclusion

The welding residual stress analysis shows that there is a large residual stress between the output needle and the helix. The finite element calculation results of X and Y directions with large stress are extracted by simulation, as shown in Table 1 and Table 2 respectively. It can be seen from the comparison that the residual stress increases obviously after the cooling of spot welding. Among them, the residual stress of the solder joints with deep-buried spherical defects in the same direction is greater than that without defects. Therefore, in the process of reducing solder joint defects can significantly reduce the welding residual stress value. The distribution of residual stress is different as well. After cooling, both X and Y directions of solder joints without defects have residual stress of tension and compression, and the value of compressive stress (absolute value) is greater than the value of tensile stress. The extreme values of residual stress in the X and Y directions appear at the edge of the V-shaped groove section. Residual stresses in X and Y directions also exist in welding spots with defects after cooling, and the value of compressive stress (absolute value) is greater than that of tensile stress. The extreme value of residual stress appears at the peripheral edge of the deep buried spherical defect.

Acknowledgments

This work was financially supported by the MOE Key lab of Disaster Forecast and Control in Engineering (20180930008), Jinan University.

References

[1] Yukio Ueda, Hidekazu Murakawa, Ninshu Ma. The Finite Element Method - Welding Deformation and Residual Stress Prevention - Chapter 4 [J]. Welding Deformation & Residual Stress Prevention, 2012: 99–129.
[2] Perret W, Thater R, Alber U, et al. Approach to assess a fast welding simulation in an industrial environment — Application for an automotive welded part [J]. International Journal of Automotive Technology, 2011, 12 (6): 895-901.
[3] Jiang J H, Wang Q, Lv W. Analysis on the Welding Thermal Field and Residual Stress of Thick Plate [J]. Advanced Materials Research, 2015, 1095: 693-697.
[4] Elofsson, Johan. Welding simulation with Finite Element Analysis [J]. Harvard Educational Review, 2004.
[5] Yuan Z. Simulation of Welding Residual Stress Based on ANSYS [J]. Journal of Wuhan University of Technology, 2004.
[6] Perret W, Schwenk C, Ethmeier M R, et al. Case Study for Welding Simulation in the Automotive Industry [J]. Welding in The World, 2011, 55 (11): 89-98.