Journal of Engineering Research and Reports

Volume 27, Issue 1, Page 219-224, 2025; Article no.JERR.129851
ISSN: 2582-2926

Numerical Analysis of Welding Heat Source Temperature Field

Baichuan Peng [a*], Linbing Li [a] and Zhaoyang Cao [a]

[a] *School of Civil and Transportation, North China University of Water Resources and Electric Power, Zhengzhou 450045, Henan, China.*

Authors' contributions

This work was carried out in collaboration among all authors. All authors read and approved the final manuscript.

Article Information

DOI: https://doi.org/10.9734/jerr/2025/v27i11381

Open Peer Review History:
This journal follows the Advanced Open Peer Review policy. Identity of the Reviewers, Editor(s) and additional Reviewers, peer review comments, different versions of the manuscript, comments of the editors, etc are available here: https://www.sdiarticle5.com/review-history/129851

Short Communication

Received: 10/11/2024
Accepted: 13/01/2025
Published: 16/01/2025

ABSTRACT

This article studies the numerical simulation analysis of the temperature field of welding heat sources. During the welding process, the rapid input of local heat and subsequent rapid cooling can cause permanent plastic deformation of the weld seam and its surrounding area, resulting in residual stress and deformation. In order to accurately grasp the variation law of the welding temperature field, this article uses ANSYS software for numerical simulation, and uses birth and death element technology and volume heating rate to simulate the welding temperature field. In the simulation, convection and heat conduction are mainly considered, and the heat source is simulated by applying the heat generation rate at the nodes of the welded components. Taking Q235 thick plate T-joint as the research object, a double-sided weld model is established, and thermal analysis is carried out using ANSYS software. The results indicate that during the welding process, the temperature at the weld seam rises sharply and then diffuses to the surrounding area,

**Corresponding author: Email: 2538791996@qq.com;*

Cite as: *Peng, Baichuan, Linbing Li, and Zhaoyang Cao. 2025. "Numerical Analysis of Welding Heat Source Temperature Field". Journal of Engineering Research and Reports 27 (1):219-24. https://doi.org/10.9734/jerr/2025/v27i11381.*

forming a trend of diffusion from the centerline of the weld seam to the surrounding area. After a period of welding, due to the influence of welding sequence, the welded part will form a quasi steady state temperature field with a steeper temperature gradient behind the heat source.

Keywords: Welding heat source; temperature field; numerical simulation; ANSYS; finite element analysis.

1. INTRODUCTION

Welding is a complex physical and chemical process. In industry, the distortion, residual stress and tensile strength of weld and heat affected zone are directly related to the thermal cycle of welding heat melting process, which makes the simulation of welding temperature field become a hot topic for welding workers at home and abroad.

In the process of welding, a large amount of heat is input in a short time, which leads to uneven heat transfer and heat dissipation. The temperature in some parts of the workpiece rises rapidly, resulting in different thermal expansion and strain and deformation, resulting in serious residual stress and welding deformation after the assembly of a large number of welded components (Cui et al., 2018), which leads to the reduction of the ultimate bearing capacity of the component and the fatigue strength of the structure. Therefore, it is extremely important to accurately grasp the numerical simulation and variation law of the welding temperature field to ensure the welding quality (Zhao & Wu, 2012).

In the welding process, due to the short-term application of a large amount of local heat and subsequent rapid cooling, permanent plastic deformation occurs in the weld and its surrounding areas, resulting in residual stress and distortion, and additional time is required to correct the deformation during the structural assembly process (Numerical simulation, 2024). From the 1960s to the 1970s, finite difference method (FDM) and finite element method (FEM) were successively introduced into welding numerical calculation, and two new methods were used to solve various complex problems of welding. At present, a lot of research has been carried out on the numerical simulation of welding process at home and abroad. In recent years, many scholars have carried out a large number of numerical simulation studies on welded components. Ronda et al. calculated and analyzed the temperature field change law of multi-pass butt welding of thick plates with TMM model (Deng, 2009). Wang Jianhua et al. of Shanghai Jiao Tong University and Osaka University of Japan jointly studied the three-dimensional welding temperature field and explored its characteristics and methods to improve the accuracy (Ronda & Oliver, 2000; Wang et al., 1999; Wang et al., 1996).

During welding, the local heat and subsequent rapid cooling can cause permanent plastic deformation of the weld seam and its surrounding area, resulting in residual stress and deformation. In order to further understand the change law of temperature field in the welding process, ANSYS software is used to simulate the welding process of the connecting weld. In this paper, the welding temperature field is simulated by the birth and death unit and the body heat generation rate (Perret et al., 2010).

2. BASIC FINITE ELEMENT THEORY OF WELDING TEMPERATURE FIELD

During welding, the heat source heats the local part of the weldment with a very high energy density, so that the temperature in the region rises rapidly, and then as the heat source moves away, the weldment cools rapidly. This process of heating and cooling is not a simple linear change (Ai et al., 2023), but involves complex heat transfer, material properties and dynamic evolution of temperature field with time. It belongs to a typical nonlinear transient heat conduction problem. The governing equation of thermal analysis is:

$$\frac{\partial}{\partial_x}\left(\lambda_{xx}\frac{\partial T}{\partial_x}\right)+\frac{\partial}{\partial_y}\left(\lambda_{yy}\frac{\partial T}{\partial_y}\right)+\frac{\partial}{\partial_z}\left(\lambda_{zz}\frac{\partial T}{\partial_z}\right)+q=\rho c\frac{dT}{dt}$$

$$\frac{dT}{dt}=\frac{\partial V}{\partial t}+V_x\frac{\partial T}{\partial_x}+V_y\frac{\partial T}{\partial_y}+V_z\frac{\partial T}{\partial_z}$$

In the formula : material density ; c is the specific heat capacity of the material ; t is the temperature field distribution function ; t is time ; the thermal conductivity of the material ; q is the intensity of internal heat source ; c,, change with the change of temperature(Wang et al., 2020 & Hu & Wu, 2007).

In the actual simulation process, there are three main boundary conditions commonly used for welding heat conduction analysis: the initial temperature of the known component, the heat flux density distribution on the boundary of the known component during the welding process, and the heat exchange between the known component and surrounding objects.

In the analysis process, the focus is on two factors: convection and heat conduction. As for heat radiation, the appropriate approach is to increase the convection coefficient and heat conduction coefficient to roughly simulate the effects of radiation. The simulation work for welding heat sources is achieved by applying heat generation rates at the corresponding unit nodes of the weld seam.

3. FINITE ELEMENT MODEL OF STRUCTURE

The establishment of geometric model is an important step in the process of numerical analysis. The quality of the model is not only reflected in the speed of calculation, but also affects the results of calculation (She et al., 2021). Taking the T-joint of Q235 thick plate as the research object, the double-sided welding seam model is established. It is assumed that the physical performance parameters of wood and welding material are the same. As shown in Fig. 1, the two thick plates are both 20mm.In order to improve the calculation efficiency and accuracy, the mesh size is 5mm, and the mesh is adaptively divided. The thermal analysis function of the large general finite element software ANSYS is used to simulate the welding process. Thermal analysis of the structure is performed using the Solid70 unit.

During the implementation of welding operations, precise control of the entire welding process is achieved through the loop control function of APDL language. At the same time, according to relevant professional standards, carefully review the data and accurately input the corresponding key parameters of the welding heat source, which cover multiple aspects such as welding voltage, welding current, welding heat power, and welding speed. The detailed welding heat source parameters are listed in Table 1.

Using the birth and death element technology to simulate the welding heat input process by simulating the weld filling method, the temperature load is applied in the form of heat generation rate. The calculation formula for heat generation rate is shown as follows:

$$HGEN = Q/(A_{weld\times}v) = \eta UI/(A_{weld\times}v)$$

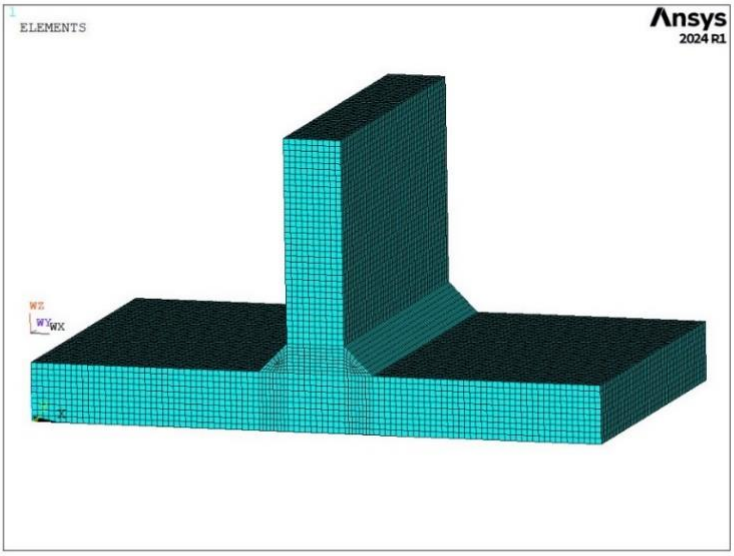

Fig. 1. T-shaped finite element model

Table 1. Heat source parameter table

weld seam	Welding consumables	Voltage u(v)	electric current I(A)	Speed V(mm/s)	Welding thermal efficiency
fillet weld	Q235	28	220	10	0.75

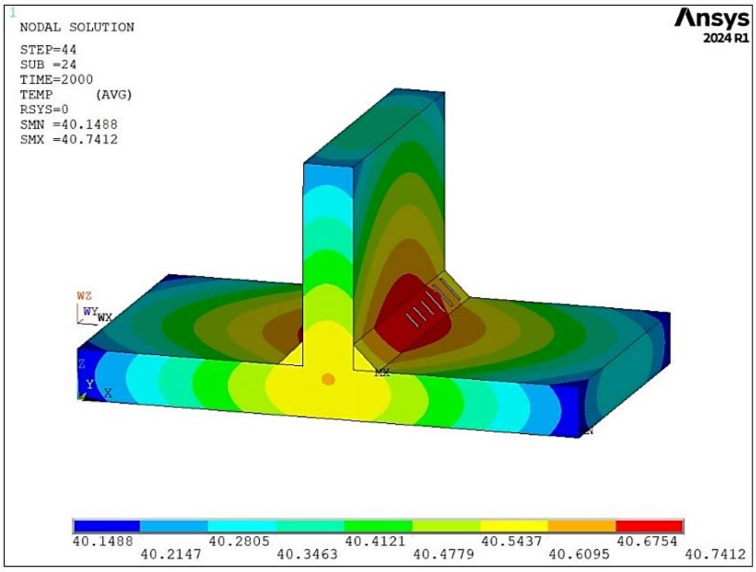

Fig. 2. Finite element model analysis results

In this simulation process, the birth death unit technique was used to carry out relevant operations. In the initial calculation stage, all units within the weld seam are first set to a "dead" state. As the calculation progresses, these units are gradually activated in a predetermined order to accurately simulate the filling process of weld metal (Gao et al., 2022; Sun & Yang, 2022; Zhang et al., 2018). Among them, each time step is set to 1 second, and the first weld seam undergoes a total of 12 time steps in the heating process, completing the welding process smoothly in 12 seconds. After the first weld seam cools to a specific temperature, the welding process for the second weld seam is immediately started, which also takes 12 seconds to complete the welding operation. Finally, the welded component enters the comprehensive cooling stage. Until the welding is completely completed, the welded component is cooled to the preset temperature, and the resulting temperature stress cloud map is shown in Fig. 2. As can be clearly seen from the figure, the difference between the highest and lowest temperatures is controlled within 1℃, which fully indicates that the weldment has been sufficiently cooled and fully meets the requirements for subsequent analysis.

4. DISCUSSION AND ANALYSIS

During the continuous progress of welding, the temperature field in which the weldment is located is in a dynamically changing state and will continue to change over time. Due to the constant movement of the welding heat source, there is a close and highly correlated temperature field and time in various parts of the welded component. In order to accurately and intuitively grasp this changing trend, we can observe the evolution of the temperature field of the welded joint over time in detail by selecting the method of generating temperature time curves.

The previous display shows the temperature field of the welded component at the final moment, in order to clearly show the changes in temperature at different positions around the weld seam over time. Select the nodes at 1/2 section of the template weldment to observe the temperature changes around the weld seam. The temperature curves of each point in Fig. 3 correspond to the temperature changes over time, as shown in Fig. 3. From Fig. 3, it can be seen that the temperature changes at each point during the entire welding process are very uneven. At the measuring point inside the weld seam, when the heat source acts on this point, the temperature rises sharply, producing the first peak. As time cools, the temperature decreases. Then, the second weld seam is welded, and the temperature rises sharply, producing the second peak. As the heat source leaves, the temperature drops sharply, then gradually becomes flat, and finally all drop to room temperature. The higher the temperature peak, the more intense the change. At locations farther away from the heat source, when the heat source acts on the section, the peak value is lower and the change is relatively gentle.

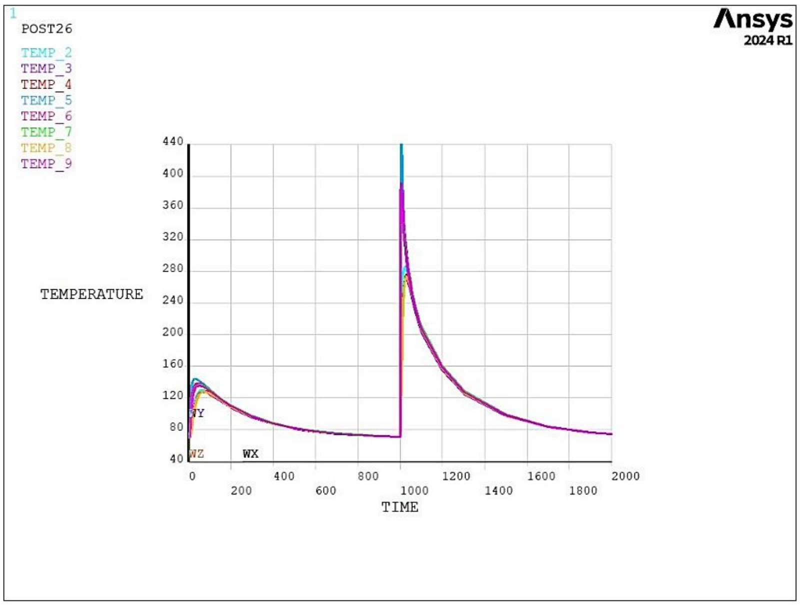

Fig. 3. Temperature time curve during welding process

5. CONCLUSION

Based on this study, it can be concluded as follows

When the welding is carried out, the temperature at the weld seam rises sharply, and then passes to the surrounding area, forming a trend of diffusion from the weld center line to the surrounding area.

During the welding process, due to the influence of the welding sequence, a quasi-steady-state temperature field with a steeper temperature gradient behind the heat source than in front will be formed on the weldment.

The welding process of the structure is simulated by ANSYS, and the welding is realized by APDL cycle, which can simulate the welding process more accurately.

DISCLAIMER (ARTIFICIAL INTELLIGENCE)

Author(s) hereby declare that NO generative AI technologies such as Large Language Models (ChatGPT, COPILOT, etc) and text-to-image generators have been used during writing or editing of this manuscript.

COMPETING INTERESTS

Authors have declared that no competing interests exist.

REFERENCES

Ai, M. J., Du, D. Z., Bai, Y. P., et al. (2023). Simulation analysis of temperature and stress fields in dissimilar high-strength steel welding based on ANSYS. *Metal Processing (Hot Processing), (05)*, 86-92.

Cui, C., Bu, Y., Li, J., et al. (2018). Distribution characteristics of welding residual stress between steel box girder panel and U-rib. *Journal of Southwest Jiaotong University, 53*(02), 260-265.

Deng, D. (2009). FEM prediction of welding residual stress and distortion in carbon steel considering phase transformation effects. *Materials and Design, 30*(2), 359–366.

Gao, X., Wu, C. W., Zhang, C. P., et al. (2022). Numerical simulation of temperature and stress fields in low carbon steel cylindrical ring welding based on ABAQUS. *Equipment Manufacturing Technology, (04)*, 80-83.

Hu, M. Y., & Wu, Z. S. (2007). Research on temperature field and stress field simulation based on unit birth death welding. *Mechanical Engineering and Automation, (06)*, 58-60.

Numerical simulation of multi-layer and multi-pass welding of Q420 steel plate butt joint. (2024). *Metal Processing (Hot Processing), (09)*, 25-31 + 35.

Perret, W., Schwenk, C., & Rethmeier, M. (2010). Comparison of analytical and numerical welding temperature field calculation. *Computational Materials Science, 47*(4), 1005-1015.

Ronda, J., & Oliver, G. J. (2000). Consistent thermo-mechano-metallurgical model of welded steel with unified approach to derivation of phase evolution laws and transformation-induced plasticity. *Computer Methods in Applied Mechanics and Engineering, 189*(2), 361–417.

She, C. L., Su, K. L., Zhang, C. X., et al. (2021). Finite element numerical simulation of temperature field and stress-strain field in steel plate welding. *Welding Technology, 50*(04), 16-20. https://doi.org/10.13846/j.cnki.cn12-1070/tg.2021.04.005

Sun, H. J., & Yang, S. H. (2022). Research on the heat source model of welding temperature field for aluminum alloy T-joints. *Journal of Naval Electronic Engineering, 42*(05), 164-169.

Wang, J., et al. (1996). Improvement in numerical accuracy and stability of 3-D FEM analysis in welding. *Welding Journal, 75*(4), 129-134.

Wang, J., et al. (1999). An FEM model of buckling distortion during welding of thin plate. *Journal of Shanghai Jiao Tong University, E-4*(2), 69-72.

Wang, S. J., Sun, Z. G., Zhou, Y. X., et al. (2020). Simulation analysis of welding temperature field based on ANSYS. *Hot Processing Technology, 49*(09), 120-121 + 129. https://doi.org/10.14158/j.cnki.1001-3814.20190914

Zhang, S. L., Guo, Z., Liu, D., et al. (2018). Analysis of temperature field of L-type welding parts based on ANSYS. *Automotive Practical Technology, (14)*, 120-122. https://doi.org/10.16638/j.cnki.1671-7988.2018.14.046.

Zhao, Q., & Wu, C. (2012). Numerical simulation analysis of welding residual stress of U-rib stiffened plate. *Engineering Mechanics, 29*(08), 262-268.

Peer-review history:
The peer review history for this paper can be accessed here:
https://www.sdiarticle5.com/review-history/129851