

Thermal-Mechanical Analysis of Residual Stress and Deformation in 6082 Aluminum Alloy Welded Joints

Yingqi Cao

*Testing Engineering Department, COMAC Shanghai Aircraft Flight Test Co., Ltd., Shanghai, China
yingqic1128@163.com*

Abstract. Welding residual stress and deformation have a significant impact on the welded structure and service performance of aluminum alloys. The three-dimensional temperature-displacement coupling of 6082 aluminum alloy butt welding joints was established by using ABAQUS software, and the residual stress distribution and deformation behavior during the welding process were studied. First, transient thermal analysis is conducted to obtain the temperature distribution during the welding process. Then, the obtained temperature field is introduced into mechanical analysis to evaluate the residual stress and structural deformation caused by welding. The results show that the temperature in the weld zone is the highest and gradually decreases towards the base material, with a distinct temperature gradient. During the thermal cycling process of welding, due to uneven thermal expansion and contraction, significant residual stress is concentrated in the weld seam and heat-affected zone. In addition, obvious deformation was observed near the welding area. The established numerical model provides a valuable basis for understanding the thermodynamic behavior of welded aluminum alloy structures and can offer references for the optimization of welding processes and structural design.

Keywords: Welding, aluminum alloy, stress, finite element simulation

1. Introduction

Currently, aluminium alloys are widely used in aerospace structures due to their high strength, light weight, corrosion resistance [1], high-temperature resistance and machinability. In order to reduce complex structural designs and to meet the ever-increasing demand for lightweight design [2], the use of welded joints in aluminium alloy structures has become a prevailing trend. However, the complex thermal cycling processes involved in welding, such as heating and cooling, can cause changes in the microstructure of the material and generate residual stresses, which have a significant impact on the structural stability and integrity, particularly in the weld zone and its surrounding areas [3]. Stress build-up often happens in the weld zone. This build-up can start cracks. It can also make cracks grow faster. In the end, this can cause the structure to fail. The start and growth of cracks are very important for the safety of aerospace structures. So a good understanding of how welding stresses affect crack behavior is needed.

To date, many studies have been done on temperature fields and leftover stresses in welding. These studies have been carried out by researchers around the world. But most of these studies look

at just one physical field at a time. There is not enough work on the link between welding stresses and how cracks start and grow. This limits the full understanding of how welded structures fail.

In response to the above issues, this paper looks at residual stresses in aluminum alloy welds. A coupled physical field simulation model is built. This model is used to study how these stresses affect crack start and growth. The findings help explain how welding stresses affect the failure of aluminum alloy structures. They also provide a basis for better design of aluminum alloy parts in the future.

2. Literature review

In recent years, scholars both in China and abroad have conducted extensive research into the temperature distribution during welding.

As for residual stress, areas heat up and cool down very quickly during welding. This fast temperature change makes the material expand and shrink in uneven ways. The uneven movement creates a complex leftover stress field. Luo et al. utilised the finite element method to investigate the formation mechanisms of welding residual stresses [4, 5]; Withers PJ et al. further pointed out that residual stresses not only affect the static strength of structures but also reduce the fatigue life and service reliability of welded structures [6, 7]. At the same time, thermal strain is induced in the structure due to the heat input from welding, phenomena such as deformation, longitudinal contraction and warping typically occur. Kim et al. found that welding deformation is closely related to the magnitude of energy input, the welding path and the structural constraints [8, 9].

With the development of computer technology, finite element software such as ABAQUS and ANSYS has gradually become an important tool for numerical simulation of welding. Among these, ABAQUS, with its powerful thermal-displacement coupling analysis capabilities, has been widely applied in the prediction of welding temperature fields, residual stresses and deformation [10].

In summary, whilst considerable progress has been made in research on welding temperature fields, residual stresses and deformation behaviour, there remains scope for further in-depth systematic research into the thermal-displacement coupling behaviour of 6082 aluminium alloy welded joints.

3. Numerical methodology

3.1. Geometry model

In order to investigate the effect of temperature variations during the welding process on the distribution of post-weld residual stresses in aluminium alloys, a finite element model of aluminium alloy welding was developed in this study. As thermal-mechanical coupled simulations of welding structures involve a significant computational load, the model was appropriately simplified. The model primarily comprises the base metal region, the weld region, and the constrained region.

This geometric finite element model was designed and constructed using the ABAQUS platform. The final design model consists of a plate measuring 100 mm×100 mm with a thickness of 4 mm. A weld was then created along the X-axis, and the model was split along the X-axis at its centre to simulate the weld.

3.2. Material properties

In the thermodynamic analysis, the material parameters defined primarily include density, thermal conductivity and specific heat capacity; whereas in the structural analysis, parameters such as elastic modulus, Poisson's ratio, coefficient of thermal expansion and yield strength must be defined. These parameters vary with temperature. During the welding process, rapid fluctuations in local temperature cause uneven thermal expansion and contraction in different regions of the material, thereby generating thermal stresses. Generally, the relationship between thermal stress and temperature can be expressed as,

$$\sigma = E\alpha \Delta T \quad (1)$$

where σ is the thermal stress, E is the Young's modulus of the material, α is the coefficient of thermal expansion of the material, and T is the change in ambient temperature.

As can be seen from this formula, changes in ambient temperature affect the stress conditions of the material. Therefore, to improve simulation accuracy and more accurately reflect changes in material properties during the welding process, the material parameters used in this paper are defined based on temperature variations as follows.

Table 1. Material parameters based on temperature

Temperature (°C)	Density (kg/m ³)	Thermal Conductivity (W/m·K)	Thermal Expansion (1e-6/K)	Young's Modulus (GPa)	Poisson's Ratio	Yield Strength (MPa)	Specific Heat (J/kg·K)
20	2700	180	23.5	70	0.33	250	900
100	2685	185	24	68	0.33	200	950
200	2670	190	24.8	64	0.33	140	1000
300	2655	195	25.5	58	0.33	80	1050
400	2640	200	26.5	50	0.34	30	1100

3.3. Mesh generation

By appropriately meshing the domain and controlling the element size, the stability and accuracy of the temperature field and residual stress calculation results can be effectively improved.

Therefore, the elements were placed along the edges of the model, with four elements in the thickness direction; the number of elements in the horizontal direction gradually decreases from the weld towards both sides; and the elements were distributed uniformly in the depth direction. Due to the significant temperature gradients and stress concentrations in the weld zone, local mesh refinement was employed to enhance computational accuracy. In regions distant from the weld, a relatively coarse mesh was used to reduce the overall computational load and improve computational efficiency. Following the generation of the mesh, the element type was assigned as temperature-displacement coupled, specifically the C3D8RT element.

To simulate the gradual formation process during welding more realistically, birth-death elements were established at the weld. In actual welding, the weld does not form entirely at the initial moment, but is formed as the surrounding base metal is gradually melted and deposited during heating; therefore, the live-dead element method can more accurately reflect the welding heat process and the step-by-step activation of the material. In the initial state, all live-dead elements are

deactivated; as the welding process progresses and the temperature gradually rises, the corresponding live-dead elements are activated sequentially.

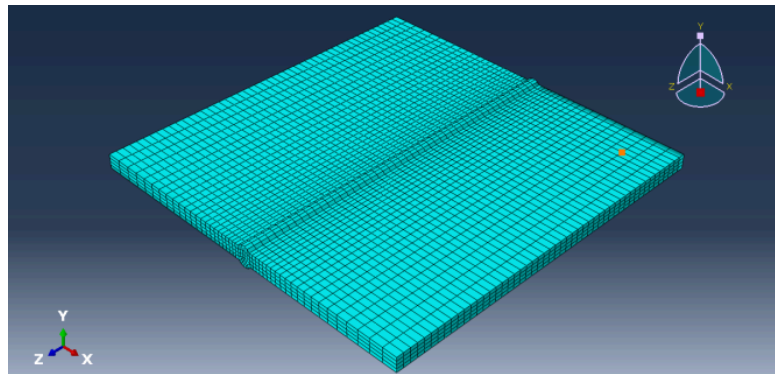


Figure 1. Mesh division illustration

3.4. Thermal analysis

The welding process of aluminium alloys constitutes a transient heat conduction problem. In order to more accurately reflect actual welding conditions, this paper conducts a transient thermal analysis of the temperature field within aluminium alloy plates during the welding process. The heat flux density is expressed as follows:

$$Q = \frac{q}{A} \quad (2)$$

Here, q represents the heat flux density, Q represents the heat input from the heat source, and A represents the area over which the heat input acts.

Analysis of this formula reveals that, during the welding process, the weld zone heats up rapidly and heat is conducted to the surrounding base metal. As the welding heat input is distinctly localised, a significant temperature gradient forms in the weld zone. As welding concludes, the aluminium alloy plate structure gradually cools and stabilises.

In this temperature-displacement coupled simulation, the heat input is applied to the aluminium alloy plate region in the form of surface heat flux to simulate the heating process of the entire structure by the welding heat source. To make the simulation closer to real welding conditions, set the model's initial temperature to 20°C, and the maximum heating temperature is set to 400°C.

3.5. Thermo-mechanical analysis

Heating step is first set up. Then a transient analysis is performed. It uses a temperature-displacement coupled analysis step on the finite element model. The weld was divided into 10 time steps during meshing. Therefore, 10 analysis steps must also be set up. Each time step has a welding time of 2 seconds. After the 10th analysis step, a cooling step is added. This cooling step also uses temperature-displacement coupling.

Due to structural influences, the weld often experiences large tensile stresses. While the base metal zone experiences compressive stresses. These residual stresses in the weld can change the overall behavior of the structure. They can also shorten the fatigue life of the structure. And they can increase the risk of structural failure.

3.6. Interaction

The ambient temperature across the entire model surface is set to 20 °C. The film layer's heat transfer coefficient is set to 10. Radiation conditions are applied to the outer surface, with an ambient temperature of 20 °C and an emissivity of 0.85. These surface heat exchange conditions are included in the simulation. This allows heat to dissipate to the surroundings during the post-weld cooling process. The accuracy of the welding simulation is therefore improved.

3.7. Load

During welding, the heat input changes the temperature and stress fields of the structure. So loads must be added to the model in the simulation.

In real welding, the structure cannot deform freely. To modify this condition, proper boundary limits must be applied to the whole model. These limits also prevent rigid-body movement in the finite element analysis. In this simulation, displacement-rotation limits are set in the initial step. The faces on both sides of the plate are fixed. Displacement limits are applied to both ends of the model. These limits stop overall rigid-body motion. At the same time, they keep the structural freedom for thermal expansion. This setup gives a more accurate picture of how thermal stresses form during welding.

During the structural analysis, the temperature field from the thermal analysis must be brought in. It is used as a predefined field in the structural analysis step. This allows thermal expansion and cooling shrinkage to create welding residual stresses. In this finite element analysis, the surrounding temperature is set to 20 °C.

4. Results and discussion

4.1. Residual stress distribution analysis

In order to better investigate the behaviour of residual stresses during the welding process and improve the quality of subsequent structural designs, this paper analyses the Von Mises stress field (S) following welding; the stress contour plot is shown in the figure 2a.

The contour plot shows clear stress build-up in the weld zone and the heat-affected zone. The highest residual stresses are found at the center of the weld. High temperatures are applied to the weld zone during welding. The local heat input causes the material in the weld zone to expand. During the cooling stage, the weld zone is held back by the surrounding base metal. This constraint creates large residual stresses inside the structure.

The contour plot also shows that the stress generated decreases gradually in regions further away from the weld. The heat source is concentrated mainly in the weld zone during welding. So the temperature drop becomes smaller further away from the weld. Consequently, the effects of thermal expansion and cooling contraction on the material become weaker, this leads to lower stress levels toward the outside areas. The high temperatures in the weld zone also change the material's properties. The material becomes more likely to deform plastically. During cooling, the deformation cannot fully return to its original shape. So large residual stresses are formed.

Analysis of the stress field distribution in aluminium alloy welded structures reveals that the high-temperature input into the weld zone during welding is the primary cause of residual stresses. These stresses can affect the overall performance of the structure. They may also shorten the structure's fatigue life. And they can even lead to structural failure. In real engineering work, these

residual stresses can be reduced. Better welding methods can be chosen. The heat input can be controlled. The cooling conditions can be improved. These steps can also make the structure safer.

4.2. Structural Deformation Analysis

As can be seen from the above analysis, during the welding process, the structure undergoes significant thermal expansion due to the high temperatures acting on the weld zone. In order to understand the effects of welding on material deformation and the structural deformation behaviour, this paper analyses the displacement field (U) following completion of the weld; the displacement contour plot is shown in Figure 2b.

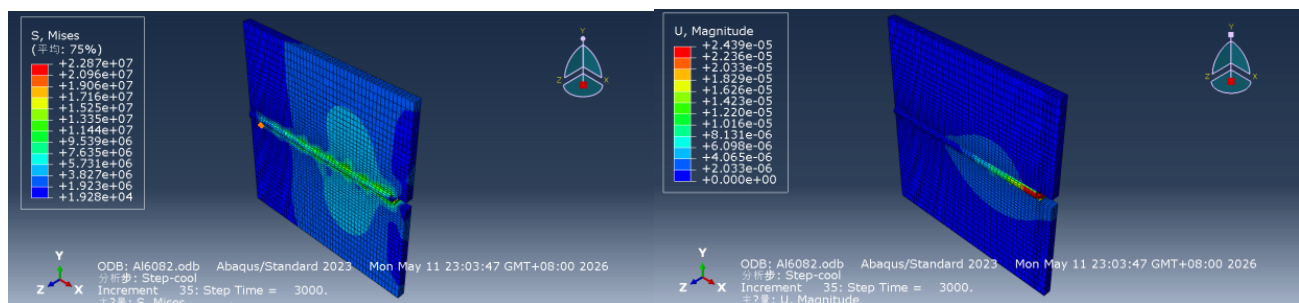


Figure 2. contour plot (a) stress contour plot (b) displacement contour plot

The stress-strain map show that the structure has deformed to some degree. The displacement changes are most noticeable near the weld zone. The weld zone receives the highest heating temperatures. So the material there expands the most. This causes the largest displacement around the weld. Conversely, the further away from the weld zone, the less the material is affected by heat. This leads to less displacement in those areas. During welding, the material in the weld zone undergoes thermal expansion when heated and contraction during the cooling phase. However, different parts of the welded plate are held together by constraints. So the material cannot move freely. This leads to deformation after welding. Furthermore, due to the influence of high temperatures, thermal deformation is more pronounced in the weld zone. Further away from the weld, the temperature change is smaller. So the structure deforms less in those areas.

In real engineering work, welding deformation can be reduced. The heating settings and welding paths can be controlled. Proper process steps can also be used. These measures improve the structure's stability and manufacturing quality.

4.3. Discussion

Analysis of the stress and displacement fields during welding shows that the heat input generated by the welding heat source creates a clear temperature difference in the weld zone. The weld zone heats up and cools down quickly during the process. This rapid temperature change is the main cause of stress formation. So large stress build-up appears in the affected areas of the structure.

Furthermore, high concentrations of residual stress can lead to structural failure. So in real engineering work, steps must be taken to reduce these effects, such as optimising welding heat input parameters.

Furthermore, this study is primarily based on numerical simulation methods and does not account for the effects of factors, such as actual welding defects and changes in the internal microstructure of

the material on the structure. Subsequent experiments can be done to check the results. Further studies can also look at welding crack growth and fatigue failure behavior.

5. Conclusion

This paper uses ABAQUS to perform a heat-displacement coupled finite element analysis of the welding of 6082 aluminium alloy. Based on the simulation results, the following conclusions can be drawn:

(1) The heat input from welding is concentrated in the weld zone. The heat spreads out as the distance from the weld increases. This creates a temperature gradient.

(2) Welding causes pronounced stress concentration within the structure. The stress contour plots show that the highest leftover stresses are near the weld and the heat-affected zone. This happens because the weld zone expands when heated. During the cooling phase, the surrounding structure holds the weld zone back. This results in substantial residual stresses.

(3) Welding causes significant thermal deformation in the structure. The displacement contour plots show that the deformation is largest near the weld zone. The deformation gets smaller as the distance from the weld increases. This trend matches the temperature field pattern.

Due to limitations in experimental conditions, the study of the welded structure has been based solely on finite element simulations; in future, experimental testing may be incorporated to validate the simulation results, and research may be conducted into the effects of factors such as welding defects, microstructural evolution and multi-pass welding processes on weld performance.

References

- [1] Zhao, S., Yu, C., Zhang, W. et al. Galvanic Corrosion Behavior of 6082 Aluminum Alloy Coupled to SUS304 in Simulated Water Circumstance with Various Salt Concentrations. *J. of Materi Eng and Perform* (2026). <https://doi.org/10.1007/s11665-026-14025-4>
- [2] Sun, Q., Zhang, Z., Zheng, J. et al. Study on the Microstructure and Mechanical Properties of 6082 Alloys for Pre-hardened Forging Process. *J. of Materi Eng and Perform* 35, 12928–12937 (2026). <https://doi.org/10.1007/s11665-025-12500-y>
- [3] Khodabakhshi, A., Havia, J., Afkhami, S. et al. The effects of different weld zone characteristics on mechanical properties and fracture behavior of MIG welded 5754, 5083, and 6082 aluminum alloys. *Weld World* (2025). <https://doi.org/10.1007/s40194-025-02180-6>
- [4] Luo, Z., Liu, C., Li, J. et al. Optimization of Finite Element Mesh for Residual Stress Measurement Using Contour Method. *Exp Tech* 50, 481–502 (2026). <https://doi.org/10.1007/s40799-025-00859-5>
- [5] M. S. Fakhri, A. Al-Mukhtar and I. A. Mahmood, Investigating spot weld fatigue failure with experimental and finite element analysis methods, *Jordan Journal of Mechanical & Industrial Engineering*, 81 (2) (2024).
- [6] Withers PJ (2007) Residual stress and its role in failure. *Rep Prog Phys* 70(12): 2211–2264. <https://doi.org/10.1088/0034-4885/70/12/R04>
- [7] James MN (2011) Residual stress influences on structural reliability. *Eng Fail Anal* 18: 1909–1920. <https://doi.org/10.1016/j.engfailanal.2011.06.005>
- [8] Kim, H., Yeon, B., Jeon, J. et al. Influence of heat source model on plasma arc welding thermal deformation analysis. *J Mech Sci Technol* 39, 3371–3379 (2025). <https://doi.org/10.1007/s12206-025-0530-9>
- [9] Jin, C., Xu, C., Li, P. et al. Study on the effect of high energy ultrasonic wave on MIG welding deformation and welding joint performance of LC52 aluminum alloy plate. *Sci Rep* 15, 11466 (2025). <https://doi.org/10.1038/s41598-025-89545-5>
- [10] Sudhir Kumar, Maurya, M., Bansal, S. et al. Assessment and Analysis of Hardness for Bobbin Tool-Friction Stir Welded Joint for AA 6063 Using Taguchi Technique and Abaqus Software. *Phys. Metals Metallogr.* 126, 969–980 (2025). <https://doi.org/10.1134/S0031918X25600782>