Finite Element Analysis of Friction Stir Welded 6082-T6 Aluminum Alloy’s Residual Stress

Bowen Liang1,2, Ronghua Li1, Xiujuan Zhang1, Yanjun Chen
1 Huanghe Road 794, Dalian Jiaotong University, Dalian, China
2 liangboowen@163.com

Abstract. The existing finite element simulations of friction stir welding are mostly static analysis or simulation of a certain machining process. A new dynamic model is proposed, in order to simulate the whole process of friction stir welding, it is based on the thermoelastic-plastic method, Johnson-Cook constitutive equation and penalty contact algorithm. The residual stress of aluminum plate after friction stir welding is obtained by using this model. The results obtained by this model are close to the results gained by experimental verification. This model can simplify the simulation of friction stir welding process, and the results gained can be greater.

1. Introduction
Friction stir welding is suitable for aluminum alloy welding. A numerical model of friction stir welding temperature field for 6061-T6 aluminum alloy plate was established by Xiao Yihua[1]. A numerical simulation of friction stir welding of 5083 aluminum alloy sheet using three-dimensional finite element method was carried out by Chen Xinhua[2]. Li Chengjin[3] simulated the welding process by presetting holes on the welding plate. Du Yanfeng[4] used a solid coupled finite element method to analyze the thermal process of friction stir welded 2219 aluminum alloy. Smriti Choudhury[5] established a finite element model to analyze the welding process of FSW. Ashu Garg[6] established a finite element model to analyze stress changes during FSW. Kareem N[7] used finite element method to conduct a fully coupled thermomechanical simulation of aluminum 6061-T6 alloy in the friction stir welding process.

A new dynamic thermo-mechanical coupling finite element simulation model is proposed for the complete welding process. It is based on the thermoelastic-plastic method, Johnson-Cook constitutive equation and penalty contact algorithm. The analysis results are compared with the experimental results.

2. Theoretical research foundation of finite element model
2.1. Johnson-Cook constitutive equation
The most widely used material constitutive equation is the Johnson-Cook constitutive equation[8]. Its mathematical form can be written as

\[ \sigma(\varepsilon_r, \dot{\varepsilon}, T) = (A + B\varepsilon_r^n)(1 + C\ln\dot{\varepsilon}^*)\left[1 - T^*\right] \]  

(1)

Where \( \sigma \) is the flow stress; \( \varepsilon_r \) is the equivalent strain; \( \dot{\varepsilon}^* \) is the equivalent strain rate, ratio of the strain rate \( \dot{\varepsilon} \) to the reference strain rate \( \dot{\varepsilon}_0 \) , i.e. \( \dot{\varepsilon}^* = \frac{\dot{\varepsilon}}{\dot{\varepsilon}_0} \); \( T \) is the temperature; \( T^* \) is equivalent
temperature, there is $T^* = \frac{T - T_s}{T_m - T_s}$; A is the yield stress of the material; B is the strain hardening factor; C is the strain rate sensitivity index; m is the temperature softening index; n is the work hardening index.

2.2. Penalty contact algorithm

The penalty contact algorithm is adding contact constraints to achieve contact constraint conditions. It is judged by whether the node passes through the main surface. If it passes through, an interface contact force is introduced, the force is proportional to the penetration depth and the rigidity of the main surface. In the research, classical Coulomb contact is often used. Its mathematical form can be described as

$$\tau = \mu p$$  \hspace{1cm} (2)

Where $\tau$ is the shear stress; $\mu$ is the coefficient of friction, and $\mu$ takes 0.3

3. Finite Element Analysis

3.1. Establishment of finite element model

Based on the above theory, the finite element model is established. As shown in Fig. 1(a), the welding plate is a 170mm × 170mm × 6mm cuboid 6082-T6 aluminum alloy plate. As shown in Fig. 1(b), a finite element model is established. Fig. 1(c) is the distribution diagram of the measuring points of the welding plate. Table 1 and Table 2 are the thermal physical parameters of the welding plate and the Johnson-Cook constitutive equation parameters, respectively.

(a) Welding plate appearance (b) Finite element model of welding plate

(c) Distribution of analytical points

Fig. 1 Welding plate model

| Temperature (℃) | Young's modulus (GPa) | Linear expansion coefficient (10^-6/K) | Thermal conductivity (W/mK) | Specific heat capacity (J/Kg°C) |
|----------------|-----------------------|---------------------------------------|----------------------------|-------------------------------|
| 25             | 75                    | 22.8                                  | 201                        | 955                           |
| 100            | 69                    | 23.2                                  | 212                        | 1001                          |
| 200            | 56                    | 24.7                                  | 213                        | 1064                          |
| 300            | 40                    | 25.5                                  | 223                        | 1103                          |
| 400            | 26                    | 26.5                                  | 240                        | 1150                          |
Table 2 Parameters of Johnson-Cook constitutive equation for 6082-T6 aluminium alloys

| A (MPa) | B (MPa) | C   | n   | m   | T_t (°C) | T_m (°C) |
|---------|---------|-----|-----|-----|----------|----------|
| 285     | 94      | 0.002 | 0.41 | 1.34 | 25       | 588      |

3.2. Finite element analysis results

After the friction stir welding process, the residual stress in the weld seam and its surrounding area is mainly plastic stress, and the residual stress in the base material area of the welding plate is mainly elastic stress. In actual production, the quality of the weld seam is based on the residual stress. Therefore, the analysis of the plastic residual stress in the weld seam area and its surrounding area is necessary.

3.2.1. Longitudinal residual stress

The finite element simulation of longitudinal residual stress is shown in the Table 3. It can be seen that: (1) The maximum residual stress appeared at the welding toe of the advance side is 128.55 MPa, and it is tensile stress; (2) The residual stress near the advance area is larger than the residual stress near the reverse area, the material at the advance area flow faster and the temperature changes more are caused by the rotation speed of the mixing head; (3) Near the welding area, tensile and compressive stresses appear simultaneously, due to the boundary of the welding area, temperature difference is large.

| No | Stress (MPa) | No | Stress (MPa) | No | Stress (MPa) | No | Stress (MPa) | No | Stress (MPa) | No | Stress (MPa) |
|----|--------------|----|--------------|----|--------------|----|--------------|----|--------------|----|--------------|
| 1  | -17.98       | 7  | -4.75        | 13 | 106.3        | 19 | 52.23        | 25 | 21.31        | 31 | -6.07        |
| 2  | -27.18       | 8  | -9.27        | 14 | 96.72        | 20 | 51.86        | 26 | -1.21        |    |              |
| 3  | -26.34       | 9  | 5.35         | 15 | 93.02        | 21 | -25.47       | 27 | -15.96       |    |              |
| 4  | -19.82       | 10 | 95.00        | 16 | 84.23        | 22 | -8.56        | 28 | -12.55       |    |              |
| 5  | 0.96         | 11 | 128.55       | 17 | 76.48        | 23 | -16.96       | 29 | -8.51        |    |              |
| 6  | 5.04         | 12 | 124.08       | 18 | 64.20        | 24 | -4.50        | 30 | -5.44        |    |              |

3.2.2. Transverse residual stress

The finite element simulation of the transverse residual stress is shown in the Table 4. It can be seen that: (1) The maximum residual stress appeared at the receding side is 42.40 MPa ; (2) The residual stress in the welding area is small, because the transverse temperature changes are small; (3) The welding area is compressive stress, the low temperature outside the welding area causes the welding area squeeze;

| No | Stress (MPa) | No | Stress (MPa) | No | Stress (MPa) | No | Stress (MPa) | No | Stress (MPa) | No | Stress (MPa) |
|----|--------------|----|--------------|----|--------------|----|--------------|----|--------------|----|--------------|
| 1  | 9.73         | 7  | 7.96         | 13 | -9.44        | 19 | -2.61        | 25 | 12.26        | 31 | 4.18         |
| 2  | 11.20        | 8  | 7.90         | 14 | -15.17       | 20 | 1.98         | 26 | 25.31        |    |              |
| 3  | 14.59        | 9  | 7.57         | 15 | -18.44       | 21 | -6.88        | 27 | 39.19        |    |              |
| 4  | 20.31        | 10 | 4.84         | 16 | -17.28       | 22 | -0.44        | 28 | 42.40        |    |              |
| 5  | 18.21        | 11 | -6.36        | 17 | -13.21       | 23 | -3.06        | 29 | 21.96        |    |              |
| 6  | 11.91        | 12 | -6.34        | 18 | -8.99        | 24 | -0.52        | 30 | 6.70         |    |              |

4. Experimental verification

Welding process parameters of this experiment, stirring head speed is 2000r / min; pressure is 6kN;
welding speed is 0.6m / min; inclination angle is 2 °. The experimental instrument is an iXRD laboratory X-ray stress detector produced by Proto of Canada.

4.1. Experimental measured values
Table 5 and Table 6 are the experimental measures values of longitudinal residual stress and experimental measured value of transverse residual stress, respectively. It can be seen that: (1) The maximum longitudinal residual stress appeared at the advancing side is 96.62MPa, and the maximum value of the lateral residual stress appeared at the receding side is 46.43MP; (2) Longitudinal residual stress is mostly tensile stress, and transverse residual stress is mostly compressive stress; And(3) the residual stress in advancing side changes larger than the residual stress in receding side.

| No. | Stress (MPa) | No. | Stress (MPa) | No. | Stress (MPa) | No. | Stress (MPa) | No. | Stress (MPa) |
|-----|--------------|-----|--------------|-----|--------------|-----|--------------|-----|--------------|
| 1   | -13.83       | 7   | 47.62        | 13  | 77.76        | 19  | 55.63        | 25  | 32.33        |
| 2   | -18.31       | 8   | 12.39        | 14  | 73.31        | 20  | 40.61        | 26  | -12.06       |
| 3   | -17.02       | 9   | 13.03        | 15  | 63.63        | 21  | 26.66        | 27  | -26.64       |
| 4   | -15.32       | 10  | 14.69        | 16  | 69.12        | 22  | 19.03        | 28  | -43.61       |
| 5   | 24.04        | 11  | 96.62        | 17  | 53.91        | 23  | -11.24       | 29  | -61.53       |
| 6   | 37.63        | 12  | 80.73        | 18  | 59.04        | 24  | -22.15       | 30  | -63.43       |

| No. | Stress (MPa) | No. | Stress (MPa) | No. | Stress (MPa) | No. | Stress (MPa) | No. | Stress (MPa) |
|-----|--------------|-----|--------------|-----|--------------|-----|--------------|-----|--------------|
| 1   | 13.43        | 7   | 8.96         | 13  | -11.02       | 19  | -22.51       | 25  | 15.33        |
| 2   | 8.19         | 8   | 4.85         | 14  | -17.94       | 20  | -22.43       | 26  | 27.76        |
| 3   | 18.53        | 9   | 4.57         | 15  | -25.22       | 21  | 8.40         | 27  | 33.19        |
| 4   | 18.62        | 10  | 4.44         | 16  | -24.33       | 22  | -7.44        | 28  | 46.43        |
| 5   | 15.23        | 11  | -4.78        | 17  | -27.36       | 23  | -6.06        | 29  | 25.02        |
| 6   | 9.92         | 12  | -8.99        | 18  | -20.02       | 24  | -7.52        | 30  | 9.70         |

4.2. Comparison of experimental measure and finite element simulation
Comparing the finite element simulation residual stress values with the experimentally measured residual stress values, it can be seen that: (1) The change trend of residual stress is basically consistent, and the new finite element simulation model is more suitable than the traditional finite element simulation model; (2) There are still errors in the finite element simulation data; And(3) the numerical peaks of the finite element simulation and the experimentally measured numerical peaks appear at the same position.
Table 7 Longitudinal residual stress error values of Finite element simulation and experimental measure

| No | Stress (MPa) | No | Stress (MPa) | No | Stress (MPa) | No | Stress (MPa) | No | Stress (MPa) |
|----|--------------|----|--------------|----|--------------|----|--------------|----|--------------|
| 1  | -4.15        | 7  | -52.37       | 13 | 28.54        | 19 | -3.4         | 25 | -11.02       |
| 2  | -8.87        | 8  | -21.66       | 14 | 23.41        | 20 | 11.25        | 26 | 10.85        |
| 3  | -9.32        | 9  | -7.68        | 15 | 29.39        | 21 | -52.13       | 27 | 10.68        |
| 4  | -4.5         | 10 | 80.31        | 16 | 15.11        | 22 | -27.59       | 28 | 31.06        |
| 5  | -23.08       | 11 | 31.93        | 17 | 22.57        | 23 | -5.72        | 29 | 53.02        |
| 6  | -32.59       | 12 | 43.35        | 18 | 5.16         | 24 | 17.65        | 30 | 57.99        |

Table 8 Transverse residual stress error values of Finite element simulation and experimental measure

| No | Stress (MPa) | No | Stress (MPa) | No | Stress (MPa) | No | Stress (MPa) | No | Stress (MPa) |
|----|--------------|----|--------------|----|--------------|----|--------------|----|--------------|
| 1  | -3.70        | 7  | -1.00        | 13 | 1.58         | 19 | 19.90        | 25 | -3.07        |
| 2  | 3.01         | 8  | 3.05         | 14 | 2.77         | 20 | 24.41        | 26 | -2.45        |
| 3  | -3.94        | 9  | 3.00         | 15 | 6.78         | 21 | -15.28       | 27 | 6.00         |
| 4  | 1.69         | 10 | 0.40         | 16 | 7.05         | 22 | 7.00         | 28 | -4.03        |
| 5  | 2.98         | 11 | -1.58        | 17 | 14.15        | 23 | 3.00         | 29 | -3.06        |
| 6  | 1.99         | 12 | 2.65         | 18 | 11.03        | 24 | 7.00         | 30 | -3.00        |

5. Conclusions
Based on the analysis and experimental verification of the 6082-T6 FSW model, the following conclusions are obtained: (1) The residual stress distribution obtained using Johnson-Cook constitutive equation, thermo-elastoplastic method and penalty contact algorithm are basically same as the experimentally measured residual stress distribution; (2) The maximum of longitudinal residual stress appeared at the advancing side is 128.55MPa, and the maximum of transverse residual stress appeared at the receding side is 42.40MPa; And(3) the longitudinal residual stress distribution of the welded plate shows a bimodal trend. And the transverse residual stress shows a valley trend.

Acknowledgments
This work has been supported by the National Key Research and Development Project of China(2018YFB1107805)

References
[1] Xiao Yihua, Zhang Hao-functional. Numerical model of 6061-T6 aluminum alloy friction stir welding temperature field and influence analysis of parameters [J]. Mechanical Science and Technology, 2017, 36 (01): 119-126.
[2] Chen Xinhua, Qiu Yanchao, Zhang Jun. Finite element study of 5083 aluminum alloy friction stir welding [J]. Welding Technology, 2019, 48 (01): 10-14.
[3] Li Chengjin, Wang Luzhao, Liu Qipeng, Yang Xinhua. Numerical analysis of the influence of geometric parameters and tilt angle of the stirring head on the quality of friction stir welding [J]. Journal of Dalian Jiaotong University, 2017, 38 (05): 70-74.
[4] Smrity Choudhury, Tamrory Medhi, Durjoydhan Sethi, Sanjeev Kumar, Barnik Saha Roy, S.C. Saha. Temperature distribution and residual stress in Friction Stir Welding process [C]. Materials Today: Proceedings, India, 2020.
[5] Ashu Garg, Madhav Raturi, Anirban Bhattacharya. Experimental and finite element analysis of progressive failure in friction stir welded AA6061-AA7075 joints [J]. Procedia Structural Integrity, 2019, 17(02):456-463.
[6] Kareem N. Salloomi. Fully coupled thermomechanical simulation of friction stir welding of aluminum 6061-T6 alloy T-joint [J]. Journal of Manufacturing Processes, 2019, 45(01): 746-754.

[7] Du Yanfeng, Bai Jingbin, Tian Zhijie, Li Jinsong, Zhang Yanhua. Three-dimensional solid coupling numerical simulation of temperature field of 2219 aluminum alloy friction stir welding [J]. Journal of Welding, 2014, 35 (08): 57-60 + 70 + 115-116.

[8] AD Belegundu. A General Optimization Methodology for Ballistic Panel Design [C]. Engineering Optimization 2008 Conference, Proceedings, Brazil, 2008.