Numerical Simulation Study on T-tape Welding of Mild Steel

Liangbo Ji*, Xuejiao Sun
Yingkou Institute of Technology, Liaoning, China

*Corresponding author e-mail: 94193000@qq.com

Abstract. Welding technology as an important technology of industrial production is widely used in production practice, T-type welding joint, as the main form of welding space joint, is widely used in daily production. The article mainly uses software ANSYS, the T-type welding of Q345 low carbon steel was analyzed by numerical simulation. Calculating the changes of temperature field and stress field, the results can effectively predict the problems related to welding quality, improve production efficiency and save production cost.

1. Basic thinking of Finite element analysis of welding
The welding process includes many physical processes, which are very complicated. In the reaction process, metal temperature, thermal stress, phase transition and other properties are coupled with each other. These coupling results directly determine the results of welding temperature field and stress field. The main idea is the high temperature heat conduction of the internal element welding heat source of the metal, generate heat conduction and convection, The internal units are heated to produce thermal deformation, and generate thermal deformation, That's where the deformation comes from, and the deformation leads to stress.

![Multi-field Coupling Effect](image)

**Figure 1.** Multi-field Coupling Effect [1]
The basic thinking of welding finite element analysis, Firstly, is to assign the parameters of low carbon steel, Building the finite element model and input the relevant internal parameters of welding thermal analysis (such as enthalpy), Building element of live-death to analyze welding temperature field, Finally, the temperature field is converted into stress field to improve the whole working condition and optimize the welding process parameters.

2. Data Processing of Welding model

2.1. Building the welding model and selection of heat source

In view of this article, ANSYS software is used to build the welding numerical simulation model, the welding model is shown in FIG. (2), the welding direction is different, use MAG welding method, the welding efficiency is 0.75, the welding current is 220A, the welding voltage is 28V, and the welding speed is 10cm/min.

There are various heat source models for welding, such as gaussian heat source, double-ellipsoidal heat source, body heat generation rate heat source, etc. [2] Both the calculation efficiency and effect, in this article, the heat source of homogeneous body heat generation rate is used as the heat source of this numerical simulation. It’s characterized is high computational efficiency, which can well represent the distribution level of welding heat source and high computational accuracy. The formula of homogeneous body heat generation rate is:

\[ Q = 0.4 \times \frac{KUI}{V} + 0.6 \times \frac{KUI}{A} \]  

A is the surface area of welding seam, to ensure the accuracy of welding simulation, the volume of welding heat source needs to be adjusted and corrected.

2.2. Mesh division and parameter setting

The mesh division principle of welding finite element model is welding seam and the area close to the welding seam. Because of its great influence on the results, the mesh density is closely divided, and away from the welding seam area, it has little influence on the results and the mesh generation is relatively sparse. This can effectively improve the welding efficiency without affecting the welding results; In the processing mode of material thermal performance parameters, the corresponding values of different parameters at key temperature nodes need to be input. use ANSYS software to linear insertion or extrapolation, obtain the continuous parameters, As the parameters at high temperature cannot be consulted and obtained, we need to modify the parameters appropriately through experience to meet the actual requirements. The mainly defined temperature field parameters include thermal conductivity, temperature, density and convective heat transfer coefficient, the stress field is mainly
elastic modulus, Poisson's ratio and linear expansion coefficient. Because welding is a dynamic process, it must reflect the metal cladding and heat source movement. Therefore, this article adopts the form of APDL command stream to realize simulation, and the command stream of temperature field parameters is as follows:

```
mpdata,dens,1,1,7.86,7.86,7.86,7.86,7.86,7.86      ! Define the density (mg/mm3)
......
mpdata,kxx,1,1,52,50,44,35,32.5                   ! Coefficient of heat conduction (mW/(mm*C))
*dim,conve,table,8,1,1,temp                     ! Considering the radiation and total convective heat transfer coefficient
    conve(0,1,1)=1
    conve(1,0,1)=20
    conve(1,1,1)=2.5
    conve(2,0,1)=150
    conve(2,1,1)=5.4
    conve(3,0,1)=350
    conve(3,1,1)=7.0
```

For the setting of welding time step, considering the calculation accuracy and actual situation of welding, we can use the trial method, according to the results produced each time, determines the numerical that increases or decreases between sub-steps. The article involves the cooling of welding seams, so in the choice of time steps, using the way of the welding time step is short, and the cooling time step is gradually enlarged, the time of the heat source is calculated to be 6s each time, in the welding process, the two-step iteration has increased or decreased the calculation accuracy, what calls for special attention is that when the time step changes, the original time needs to be deleted, go to the next step make calculate, if not handled properly, the results can be severely distorted.

```
/solu
time,j
deltim,6,6,6                         ! First cooling, time step is 6S
alls
solve
j=j+6
......
j=j-6
j=j+100                               ! Conversion time step
......
time,j
deltim,100,100,100                     ! Second cooling, time step is 100s
alls
solve
j=j+100
......
```

2.3. **Welding Numerical Simulation Calculation Process**

In this article, welding numerical simulation adopts indirect coupling method, First, calculate the temperature field of welding and then calculate the stress field of welding, this method can effectively improve the computational efficiency, and its basic steps are shown in FIG. (3).
3. **T-tape welding numerical simulating analysis**

First of all, Steady-state analysis of the entire work-piece, set the room temperature at 20°C, the weld length at 200mm, and the welding speed at 10cm/min, the time of the first welding seam is 120s, and then it is cooled 4120s. Keep the work-piece close to room temperature, start the welding of the second weld at 4240s, the welding direction of the second weld is opposite to the first weld, after 120s’ welding, cool 4120s again. The whole simulation is mainly divided into four parts, the first weld -cooling-the second weld (in the opposite direction) -cooling.

---

**Figure 3.** ANSYS Welding Numerical Simulation Flow Chart [3]
By analyzing the temperature field of welding temperature, we can clearly see the distribution rule of heat source of the welding seam moving the whole work-piece, it’s characteristic is near the welding seam temperature is high, far away from the welding seam temperature is low, the welding work-piece temperature with time transfer is clearly visible, heating and cooling curves through specific points calculate the cooling rate of the metal. The distribution rule and welding temperature of welding temperature field are roughly the same as the actual situation, It can be used as an accurate basis for welding stress field analysis.

It is clear from the simulation analysis that the maximum position of welding residual stress mainly occurs in weld edge and weld toe, the main reason is that the high temperature of welding causes the metal to expand and then cool down quickly, it forms a thermal tension, In view of this phenomenon, summarize two ways to reduce welding residual stress, preheat and keep warm. When the welding constraint is transformed, constraints should be selected to be far away from the welding seam, to avoid concentration of residual stress caused by too close proximity; The welding direction was compared in the same direction and in the different direction, the same direction and different direction have little influence on the residual stress of welding, but it has a big effect on strain, the strain in the same direction welding is much smaller than that in the different direction welding, this conclusion can provide a theoretical basis for guiding the operation method of T-type welding to reduce strain.
4. Conclusion
The heat source of homogeneous body heat generation rate can well show the characteristics of thermal gradient of welding seam, adjust the position of welding constraints can affect the numerical of welding residual stress, when the constraint position is close to the weld, the maximum value of residual stress increases significantly, when the restraint is far away from the weld, the residual stress changes little. The residual stress of welding is mainly distributed in the welding seam area and the position of welding toe, the residual stress decreases significantly away from the weld seam. The residual welding stress at the welding seam mainly comes from the tensile stress along the direction of the welding seam. The strain of welding in the same direction was significantly reduced compared with that in the different direction.

References
[1] Goldak J A, Akhlaghi M. Computational welding mechanics[M]. New York: Springer Science Business Media, 2005.
[2] Deng D, Liang W, Murakawa H. Determination of welding deformation in fillet-welded joint by means of numerical simulation and comparison with experimental measurements[J]. Journal of Materials Processing Technology, 2007, 183(2-3):219-225.
[3] Zhang J X, Liu C. Finite Element Calculation and Engineering Application of Welding Stress and Distortion[M]. Beijing:Sinence Press, 2015.