Study and application of numerical simulation method for welding process based on Marc

Qihan Gao*
Dalian Jiaotong University, China,116028

*Corresponding author e-mail: 306405608@qq.com

Abstract. The bridge and shipbuilding industry have been troubled by the issues of increased production costs and delays due to the welding residual stress and deformation. It is of great practical significance to study welding residual stress and deformation. In this paper, the FEM software Marc is used to carry out numerical simulation of plate pass welding, and the welding temperature field and residual stress distribution after welding are obtained. The simulation calculation results are qualitatively analyzed.

Keywords: butt welding, numerical simulation, Marc

1. Introduction
The rapid development of computer technology has brought profound effects to various fields. In combination with the continuous improvement of numerical calculation methods and technology, more and more problems in engineering and science can be studied by computer numerical simulation methods [1]. In the field of welding, compared with traditional empirical methods and experimental methods, the numerical simulation method of welding based on the finite element method has the following advantages: it can deeply understand the nature of the welding phenomenon, clarify the mutual effect of heat transfer, metallurgy and mechanics in the welding process and Function [2]; it can optimize structural design and workmanship design, thereby reducing experimental workload, shortening production cycle, improving welding quality, and reducing process costs [3].

During the welding process, the physical and mechanical properties of the material change highly nonlinearly with temperature, especially near the molten pool. The elastic modulus and yield strength of the material are reduced to very small values, which makes the singularity of the coefficient matrix of the numerical calculation equation increase and leads to difficulty in converging the solution [4]. Marc is a finite element program based on the displacement method, which has powerful functions in nonlinear aspects. The solver Marc uses a full range of functions. The analysis uses a highly nonlinear problem-solving technology with high numerical stability, high accuracy, and fast convergence [5].

In this paper, the FEM software Marc is used to carry out numerical simulation of plate pass welding, and the welding temperature field and residual stress distribution after welding are obtained.
The simulation calculation results are qualitatively analyzed.

2. Numerical model
The cut-out form of the plate butt multi-pass welding is shown in Figure 1. The size of the two plates is 120mm × 100mm × 8mm, and the welding material is 16Mn steel. The welding specifications are: the first welding current is 65A, the voltage is 20V, and the welding speed is 2.5mm / s; the second welding current is 135A, the voltage is 22V, and the welding speed is 1.6mm / s. The first and second welds are done at 180s intervals.

![Figure 1. Plate butt welding cutout shape](image)

Among them, the weld width is shown in formula (1):

\[ C = AB + CD + b + 2e = 2(\delta - P)\tan(\alpha/2) + b + 2e \]  

(1)

In the formula, \( e \) —the width of the welds on both sides of the groove, generally \( e = 1.5 \sim 2 \) mm, the groove angle is \( \alpha \), the gap is \( b \), and the blunt side is \( P \).

Welding is a thermo-mechanical coupling process, but because the effect of welding stress and strain on temperature is very weak compared with the temperature of the welding process, only one-way coupling of temperature to stress and strain is generally considered. There are two methods to calculate the welding temperature and stress field with Marc. One is to conduct a heat conduction analysis to obtain the temperature field distribution at the weldment and the weld during the welding process, and then apply the obtained node temperature as a load to the stress. The same geometric model is analyzed; the other is direct coupling analysis. At this time, the element used has two degrees of freedom of temperature and displacement. After the analysis and calculation are completed, the temperature field and the stress-strain field distribution are obtained simultaneously. The example in this paper uses the latter method.

2.1. Calculation model and material properties
The physical model of plate butt welding has symmetry. In order to reduce the calculation amount and shorten the calculation time, the finite element model can take only half of the symmetrical structure for calculation. The solid element is used to divide the grid, and the temperature gradient near the weld gradually increases, and the mesh is correspondingly encrypted. The grid uses 8-node hexahedral full-integral isoparametric elements, and the element number is 7. Element 7 can be used for both stress analysis and thermal-mechanical coupling analysis. In order to avoid the volume lockup caused by the incompressibility of the full integration unit, the constant dialatation parameter is selected to improve the integration method. At the same time, in order to improve the accuracy of thermal stress in low-order elements, constant temperature parameters were selected.

Some physical and mechanical parameters of 16Mn steel as a function of temperature are shown in Figure 2. The high-temperature data that cannot be obtained from the existing data can be manually set by comprehensively considering the calculation accuracy and calculation efficiency according to related literature. The traditional methods to consider the effect of phase change latent heat are modified specific heat method and enthalpy method. In Marc2005, the solid-liquid phase change
temperature and latent heat value can be directly input through the mental panel to calculate the effect of solid-liquid phase change latent heat. The solidus temperature is 1480 °C, the liquidus temperature is 1520 °C, and the latent heat of phase transition is 277 000J / Kg. Therefore, the effect of latent heat of solid phase change is relatively small, and it is not considered here.

![Thermal conductivity curve](image)

**Figure 2.** Curve of thermal conductivity with temperature

### 2.2. Welding heat source model

One of the default heat source models in the Marc2005 welding module is the Goldark circular surface heat source, which can be used to define two-dimensional line heat sources and three-dimensional surface heat sources. The other is the Goldark double ellipsoid heat source, which is used to define two-dimensional or three-dimensional body heat sources. The round surface heat source is suitable for flat plate surfacing and butt welding with small penetration depth, while the double ellipsoidal heat source is suitable for open cuts or large penetration depth welds.

The expression of the first half of the ellipsoidal heat source is shown in equation (2):

\[
(q, x, y, z, t) = \frac{6\sqrt{3}Qf_1}{abc\sqrt{\pi^3}} \exp\left\{-3\left[\frac{x^2}{a^2} + \frac{y^2}{b^2} + \frac{(z-vt)^2}{c_1^2}\right]\right\}
\]  

(2)

The expression of the ellipsoidal heat source in the second half is shown in equation (3):

\[
(q, x, y, z, t) = \frac{6\sqrt{3}Qf_2}{abc\sqrt{\pi^3}} \exp\left\{-3\left[\frac{x^2}{a^2} + \frac{y^2}{b^2} + \frac{(z-vt)^2}{c_2^2}\right]\right\}
\]  

(3)

In the formula, \(f_1\) and \(f_2\) are the heat flux density distribution coefficients, \(Q\) is the input heat source power, \(v\) is the welding speed, and \(a\), \(b\), \(c_1\), and \(c_2\) are the parameters defining the ellipsoid shape.

### 2.3. Setting of bead and weld metal filling

Marc2005 provides two methods for the progressive filling of welded metal. One is the static element method. Before welding, small values are assigned to the material parameters of the filled unit so that it hardly transfers heat or force. With the movement of the heat source, the stress and strain of the activated filled element are reset to 0, and at the same time, the normal material properties are restored. The static element method allows the element to move with the model and is particularly suitable for large deformation situations. However, the use of such elements is prone to ill-conditioned matrices and is not conducive to convergence. Another method is the unit life and death method, which is to change the life and death of the unit to process the filling process of the weld metal.
During the welding process, the weld elements that are not activated do not participate in the analysis and calculation, and do not appear in the post-processing file. Units using life-and-death techniques cannot move with other parts of the model. In large deformation problems, the elements may be distorted. Considering that the model in this example will not undergo a large displacement, in order to obtain a better convergence effect, the author uses the element life and death technology to simulate the progressive filling of the weld metal.

Since welding is a typical transient high-temperature process, in order to avoid thermal oscillation when a concentrated heat source acts on the system suddenly, a concentrated mass matrix and a concentrated heat source matrix are used in the calculation. Also activate the Large Displacement and Large Strain Plasticity Analysis option.

3. Calculation results

During the welding process, as the heat source moves, the weld unit gradually appears in the model and the temperature rises sharply. It can be seen from the temperature cloud during the welding of the second weld that the isothermal ribbon in front of the heat source changes more densely, and the isothermal ribbon change behind the heat source gradually decreases with distance, which indicates that the temperature gradient in front of the heat source is higher than the rear Temperature gradient. After the first weld was cooled for 180s, the temperature of the weld was reduced to about 100 °C, and then the welding simulation of the second weld was performed. During the second cooling process, the maximum temperature of the system dropped sharply, and the isothermal ribbon gradually dispersed. After welding, the model was finally cooled to room temperature.

Figure 3 is the temperature history curve of a node near the weld on the upper surface of the plate. When the heat source approaches, the temperature of the node rapidly rises, and after the heat source leaves, the temperature slowly decreases; during the second welding process, when the heat source approaches again, the node temperature quickly reaches another peak again, and this peak temperature is more above the previous peak temperature. During the second cooling process, the temperature drop rate slowed down.

4. Conclusion

In this paper, the non-linear FEM software Marc is used to perform three-dimensional real-time dynamic numerical simulation of plate butt multi-pass welding, and the key technologies involved in the simulation are introduced. The typical calculation results of the temperature field, stress field and deformation of the welding are given. And curve, and analysis analysis shows that in the numerical simulation of welding, the heat source model, heat dissipation coefficient and material high temperature characteristic coefficient all have certain deviations from the actual welding situation,
these factors will affect the accuracy of the simulation calculation results. Nevertheless, the calculation results can provide a qualitative reference for the analysis of welded structures.

References
[1] Yuewei Ai, Jianzhuang Wang, Ping Jiang. Parameters optimization and objective trend analysis for fiber laser keyhole welding based on Taguchi-FEA[J]. International Journal of Advanced Manufacturing Technology, 2017, 90(5):1-14.
[2] Hu R, Pang S, Chen X, et al. An octree-based adaptive mesh refinement method for three-dimensional modeling of keyhole mode laser welding[J]. International Journal of Heat and Mass Transfer, 2017, 115(7):258-263.
[3] Xiaohong Zhan, Qi Zhang, Qibing Wang. Numerical simulation of flow field in the Invar alloy laser-MIG hybrid welding pool based on different heat source models[J]. International Journal of Numerical Methods for Heat & Fluid Flow, 2018, 28(4):1-12.
[4] Hossein Talebi, Martin Frönd, Jorge F. dos Santos. Thermomechanical Simulation of Friction Stir Welding of Aluminum Using an Adaptive Element - Free Galerkin Method[J]. Pamm, 2017, 17(1):473-474.
[5] Diogo F. Almeida, Rui F. Martins, João B. Cardoso. Numerical simulation of residual stresses induced by TIG butt-welding of thin plates made of AISI 316L stainless steel[J]. Procedia Structural Integrity, 2017, 5(3):633-639.