Finite Element Analysis of Welding Residual Stress in Long-distance Gas Pipeline

Yeye Cheng, Hehui Wang*

School of Mechanical and Power Engineering, East China University of Science and Technology, Shanghai, 200237, China

*Corresponding author e-mail: Y30180329@mail.ecust.edu.cn

Abstract. This paper takes L415 pipeline steel as the research object. The temperature field and stress field of the gas pipeline welding process are numerically simulated by using ANSYS finite element analysis software. Gradient meshing is adopted by the finite element model. The simulation of the welding heat source is realized by the band heat source model, and ANSYS is used to load the heat source in a APDL command program. In the simulation process, the element birth and death technology is used to simulate the generation of weld material, and the material constitutive relationship is a bilinear follow-up strengthening elastoplastic model that changes with temperature. The simulation results show that applying the heat source in the form of heat generation rate with a belt-shaped heat source can realize effective calculation of the residual stress field of multi-layer and multi-pass welding. The calculation results in this paper provide a mechanical reference for pipeline installation and operation.

1. Introduction

Pipeline welding is widely used in gas pipeline laying due to a series of advantages such as strong and durable welded joints, excellent sealing performance, high joint stiffness, and low welding cost. However, due to the inhomogeneity of the temperature field and the nonlinearity of the material during the welding process, the welding structure has a large welding stress and deformation. The complex residual stress state may directly or indirectly reduce the bearing capacity of the welded structure. This is an inherent feature of welding thermal processing, which is caused by permanent shrinkage strain of the weld metal and its nearby base metal due to heating and cooling cycles. Due to the long distance, large diameter of the gas pipeline and laying at the bottom of the river, the actual working conditions are more severe, it is necessary to conduct an in-depth study on the residual stress distribution of such welded components.

The input of the heat source is locally concentrated, which makes the weldment form an unstable non-uniform temperature field. In addition, the temperature field can indirectly affect the welding residual stress. Therefore, the analysis of the temperature field is the premise of the welding stress and strain analysis [2].

Welding is a professional technology that requires professionals to ensure welding quality. Therefore, the use of experimental methods to obtain temperature field data during the welding process is not only time-consuming but also costly [3]. However, the ANSYS calculation can make up for this defect. We use indirect coupling to calculate the stress field of the pipeline, which is cost-effective and efficient. The calculation results in this paper provide a mechanical reference for pipeline installation and operation.
2. Welding structure and welding parameters
The pipe material simulated in this paper is L415M (X60), the pipe diameter is $\phi 813 \times 15.9\text{mm}$, the strength grade is 510~560MPa, and the V-shaped groove processing form is selected. The bevel angle is $30^\circ \pm 2^\circ$, the thickness of the blunt edge is 1~1.5mm, and the root gap is 1.5~2.5mm, as shown in the figure [1], butt welding is adopted. The ambient temperature is 20°C. The wall of the pipe is thick, so multiple layers and multiple passes are needed. The weld bead layout is shown in Figure 2.

![Figure 1. Groove shape in the weld area](image1)

![Figure 2. Welding bead layout](image2)

According to the literature [1], since the pipeline needs to be transported to the tunnel for welding and the thickness of the pipeline is thick, reaching 15.9mm, the submerged arc welding process is adopted for welding. The submerged arc welding process parameters selected in the calculation are as follows:

| Weld name | material grade | diameter (mm) | polarity | $I$ (A) | $V$ (V) | $V$ (cm/min) |
|-----------|----------------|---------------|----------|---------|--------|--------------|
| Root      | E6010          | $\phi 4.0$    | DC-      | 80-100  | 25-35  | 12-14        |
| Heat      | E8010          | $\phi 4.0$    | DC+      | 115-130 | 25-30  | 13-15        |
| Filler 1  | E71T8NiIJ      | $\phi 2.0$    | DC-      | 190-210 | 19-21  | 20-22        |
| Filler 2  | E71T8NiIJ      | $\phi 2.0$    | DC-      | 200-240 | 20-22  | 19-21        |
| Filler 3  | E71T8NiIJ      | $\phi 2.0$    | DC-      | 200-240 | 20-22  | 19-21        |
| Cover     | E71T8NiIJ      | $\phi 2.0$    | DC-      | 200-220 | 19-21  | 18-20        |

3. Calculation process and finite element model

3.1. Finite element model

![Figure 3. 3-D finite element model](image3)

A 1/4 model is established and meshed in order to facilitate the calculation, as shown in Figure 3. The gradient mesh is adopted because the mechanical problems in the weld are more complex. The mesh division in the weld zone is denser, and the mesh division far away from the weld zone is relatively sparse. In this way, under the premise of ensuring accuracy, calculation time is reduced and calculation efficiency is improved. The finite element model has 74,620 nodes and 68,400 elements.

3.2. Material properties
Accurate temperature is very important for simulating because stress and strain are related to temperature during welding process. The constitutive relationship in this paper adopts a bilinear follow-up strengthening elastoplastic model to simulate the elasto-plastic stress-strain relationship of welded parts. It assumes that the strengthening modulus of the material after yielding is 1/10 of that before yielding. The material of the pipeline is L415 (X60) pipeline steel, and its thermophysical and mechanical properties are related to temperature. The main thermophysical properties of L415 (X60) material [5] are shown in Table 2.

| T/℃ | E/Pa   | υ    | λ/[W/(m·℃)²] | ρ/(kg·m²) | C/[J/(Kg·℃)] |
|-----|--------|------|--------------|-----------|--------------|
| 20  | 2.1×10¹¹ | 0.3  | 1.22×10⁻⁵    | 7800      | 460          |
| 500 | 1.5×10⁵    | 0.35 | 1.37×10⁻⁵    | 7800      | 690          |
| 1000| 0.35×10⁵   | 0.385| 1.4×10⁻⁵     | 7800      | 880          |
| 1500| 0.065×10⁵  | 0.49 | 1.46×10⁻⁵    | 7800      | 920          |

3.3. Band heat source model

The heat source model in this paper is simplified to moving band heat source. The method is as follows: divide the welding seam into 45 parts along the central angle, that is, the length of the heat source applied each time is 2°. The back point of each section is used as the center of the heat source and the heat generation rate is applied. After heating for a period time, the heat source moves to the next heating area. The program needs to delete the heat source applied in the previous time when calculating the next time. The temperature value calculated in the previous time period as the initial value of the next time period, the *DO-ENDDO loop statement of APDL statement can simulate the movement of heat source load and realize the calculation of welding transient temperature field.

The wall of the pipe exceeds 6mm so the Gaussian heat source cannot be penetrated [6], and the weld of the weldment adopts a V-shaped groove, so the heat source should be used as the internal heat treatment of the weld element. This paper intends to apply the load in the form of heat generation rate, the formula is:

\[ H = \frac{\eta \times U \times I}{A \times V \times DT} \]  

In the formula, \( \eta \) - effective coefficient of welding power, \( U \) - welding voltage, \( I \) - welding current, \( A \) - cross-sectional area, \( V \) - welding speed, \( DT \) - load step time.

3.4. Thermal-Solid Coupling and element birth and death technology

This article uses the thermal-structure indirect coupling method of ANSYS. The indirect coupling method is a one-way coupling. The temperature field can be calculated first, and when the temperature difference is calculated accurately, the stress field is calculated. The method is flexible and efficient, and it can greatly shorten the calculation cycle. The direct coupling method is a strong coupling. Due to the large scale of transient thermal analysis and elastoplastic nonlinear iterative calculation, if the temperature field is wrong, it needs to be recalculated again, which increases the calculation cost. Therefore, the indirect coupling method is generally selected.

The formation of the weld is due to the continuous melting of the welding wire. So, the welding seam does not exist before the welding starts. When the welding starts, the welding seam begins to be continuously produced until the end of the welding. ANSYS "Element Life and Death" technology is used to simulate this process in finite element. The "birth and death" of element refers to adding or deleting part of the material in a certain period of time in the model, and the corresponding element in the model is "born" or "death". In ANSYS, element life and death is defined as a non-linear change problem of contact type.

It takes 180s for each simulation to complete a weld and 120s to cool the weld in the actual working conditions. When all the welding is completed, the finite model needs to wait for 1200s to cool to the ambient temperature. In the calculation process, each load is divided into 45, the sub-steps are divided into 10, and the unloading is divided into 40 steps. There are about 10000 iterative sub-steps in total,
which requires a large amount of calculation.

3.5. Boundary conditions of temperature field and stress field

In the temperature field simulation, the heat transfer boundary conditions of the pipe are mainly convective heat transfer and radiative heat transfer between the outer surface of the pipe and the air. To simplify the calculation, the outer surface in contact with the air is set as the convective surface load and the heat transfer coefficient can be set to 30W/(m²·℃). Suppose the air temperature is 25°C, and the symmetry plane is set as the adiabatic surface. According to the literature [4], it is known that the pipeline needs to be preheated. The preheating temperature is set to 100°C and the interlayer temperature is controlled above 100°C. When calculating the stress field, symmetric boundary conditions are applied to the symmetry plane and displacement constraints are applied to the two ends of the pipe to ensure that the entire model does not undergo rigid movement or rotation during the calculation process. The final component stress state is the welding residual stress without any external load.

4. Analysis of welding results

4.1. Simulation results of welding temperature field

Select the interlayer temperature of the initial welding position for analysis. The distribution of observation points is shown in Figure 4.

Figure 4. The location of the observation point

Figure 5 shows the thermal cycle curve of each point, where the abscissa represents the welding time and the ordinate represents the welding temperature. Location1 has 11 wave crests, and the time interval between each wave crest is 300s. Location2 is the midpoint of the cover weld, which is the outer surface of the pipe. It can be seen from Figure 5 that the interlayer temperature of the welding is higher than 100°C, indicating that the simulation of the temperature field meets the preheating conditions.
4.2 Simulation results of welding stress field
This article has six layers of welds and four welding methods. Select six representative moments for observation and analysis. As shown in Figure 6, it can be seen from the following stress diagram that the stress at the groove has increased to 407 MPa. This is caused by the thermal expansion and contraction of the weldment. The stress distribution in the figure shows a high degree of symmetry, and the heat-affected zone is narrow in the middle and wide at the ends.
5. Conclusion

(1) Analyzing the thermal cycle curve of multi-layer welding, it can be seen that the subsequent weld bead has a thermal effect on the observation point of the root weld layer, causing 11 wave peaks in the observation point of the root weld. Since the subsequent weld bead has a preheating effect on the welded bead, the position of the node with the highest temperature is constantly changing.

(2) The welding residual stress is concentrated in the weld and the heat-affected zone. The maximum equivalent residual stress is 407MPa, which is less than the yield strength of the material.

References

[1] Chen Haijun(2014). Application of Downward Welding in the Construction of City Gas Long-distance Pipeline[J]. Technical Discussion, (24):1374-1376.
[2] Chu Yajie, Liu Peng, Ma Qunshuang(2019). Welding Finite Element Technology [M]. Chemical Industry Press, Beijing.
[3] Wang Shijun, Sun Zengguang, Zhou Yongxin(2020), etc. Simulation analysis of welding temperature field based on ANSYS [J]. Thermal processing technology, 49(09):120-121+129.
[4] Zhao Hongyun, etc. Numerical simulation of temperature field in laser cladding welding based on life and death unit[J].Transactions of the China Welding Institution, 2010,31(5):81-84.
[5] Diao Luyang(2017), etc. Finite element simulation of the coupling of welding temperature and residual stress of X60 steel pipe[J]. Industrial Technology Innovation, 04(02):15-18+26.
[6] Ji Liangbo(2016). Finite element analysis of heat source for heat generation rate of double-sided flat welded body[J]. New Industrial Technology, 03(01): 33-37.