International Journal of Advanced Engineering and Management Research
Vol. 2 Issue 5, 2017

http://ijaemr.com/ ISSN: 2456-3676

STUDY ON TEMPERATURE FIELD DISTRIBUTION OF HOT MELT WELDING OF PE100

ZHANG Ao-peng MA Ting-xia LV Ya-feng
（School of Mechatronic Engineering，Southwest Petroleum University，Chengdu, Sichuan, China, 610500）

ABSTRACT

In this paper , ANSYS software was used to simulate the hot melt welding process of PE100, and the distribution of temperature field was obtained. The results of ANSYS simulation were compared with the experimental results in the literature to verify the correctness of the finite element welding process. By comparing the above two aspects, it is verified that the finite element of PE pipe welding temperature field simulates the accuracy of the whole process and obtains the distribution law of temperature field during welding. It has important guiding significance for practical engineering.

Key Words: PE pipe; numerical simulation; finite element method; hot melt welding

0. Introduction

HDPE pipe has the advantages of good corrosion resistance, tightness, long life and large circulation capacity[1]. It has become the world's best choice for the gas pipeline. Welding technology is the main connection method of engineering plastic pipes, and it is the key link that directly affects the safety application of high density engineering plastic pipes[2]. However, the pipeline structure in the welding process, due to uneven heating, cooling, material change, constraint conditions and field operations and other reasons, and will produce the deformation force and residual stress in welding joint transient internal to the production, life and social security will bring huge losses[3].

Through consulting and retrieving, it is reported that the simulation analysis of welding temperature simulation of finite element plastic viscoelasticity is less. Wu Jian-jun et al. Have applied one dimensional thermo viscoelastic finite element method ; The residual stress and bending deformation of flat injection molded parts with simple geometry are obtained and satisfactory results are obtained[4] ; Li Hai-mei et al. Used the two-dimensional thermo viscoelastic finite element method to model the residual thermal stress of plate injection molded parts The result is reasonable[5-6]

Under this background, this thesis proposes HDPE hot melt welding method around the tube, according to the principle and effect of external force field on the polymer melt mechanism of hot melt welding, hot melt welding temperature field is simulated, and the analysis of welding parameters on the temperature field and stress field. This study will provide important engineering value for improving the reliability and life of HDPE gas pipe hot melt welding joint and reducing the occurrence of gas pipe network accidents.

1 The establishment of finite element model

1.1 basic equation of welding temperature field

Based on the consideration of pipe shape and model establishment in thermal fusion welding, this paper is a three-dimensional heat conduction problem[7]:

$$c\rho\frac{\partial T}{\partial t}=\frac{\partial}{\partial x}(\lambda\frac{\partial T}{\partial x})+\frac{\partial}{\partial y}(\lambda\frac{\partial T}{\partial y})+\frac{\partial}{\partial z}(\lambda\frac{\partial T}{\partial z})+Q$$

In the equation, $Q(x,y,x)$—The internal heat source in the solution domain ; c—The specific heat capacity of materials ; ρ—Density of material ; T—The distribution function of welding temperature field ; t—Heat transfer time。Among these parameters λ , c , ρ Both vary with temperature.

1.2 Establishment and assumption of geometric model

The simulated pipe for high density polyethylene pipe PE100, pipe specifications for 160mm, SDR11 standard dimension ratio (the ratio of diameter and thickness), which is widely used in domestic PE pipe specifications for gas delivery system. Considering the symmetry, began to take the side as the research object from the weld center, the shape of welding and welding characteristics determine the finite element model of joint circular, so take the outside diameter 160mm, diameter 130.8mm, thickness 14.6mm, length 100mm cylinder to simulate. As shown in figure 1.

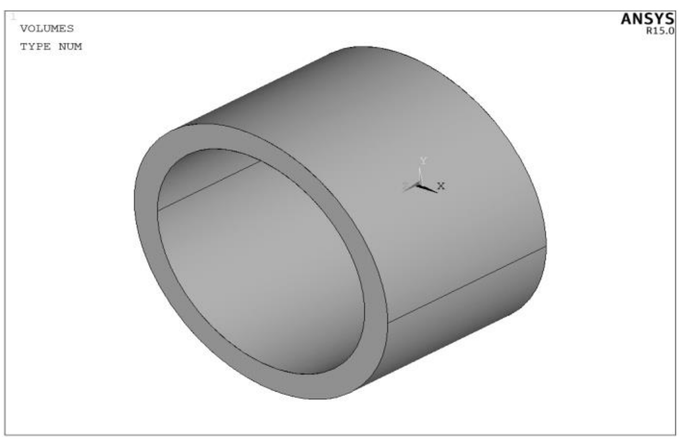


Fig1. The geometry model

Hot melt welding can be summarized into 3 stages: heating, switching and welding cooling. In the process of welding, the model can make the following hypothesis:

(1) the tube is subjected to isotropic heat transfer;

(2) the thermo physical properties of materials are only a function of temperature;

(3) the temperature distribution of the heating plate is even;

(4) ignore the heat radiation of the heating plate.

1.3Element type and mesh

This paper uses the SOLID70 three-dimensional thermal solid element, which has 8 nodes, each node with a temperature degree of freedom, which can be used for three-dimensional steady-state or transient thermal analysis problems. If the model containing the thermal entity element needs to be analyzed, it can be replaced by an equivalent structural unit.

In the finite element analysis, in general, increasing the density of the mesh can improve the accuracy of the calculation results, but the increase of the mesh density means that the computational complexity will increase and the computational cost will increase. At the same time, the density of the grid can not be unlimited rise, generally in order to ensure the accuracy of the calculation results in the scope of user control. The accuracy and the calculation results considering the time cost, the mesh refinement in the welding end, unit grid control in 1mm, the unit grid control in 5mm, a total of 257868 nodes, 236152 units. Finite element model after mesh generationAs shown in figure 2.

Fig2. meshing the geometric model

1.4The solution of temperature field

After entering the ANSYS solver, you should first define the type of analysis and analysis options that cannot be changed after the first load step (that is, after the first SOLVE command is issued). The analysis of the welding temperature field is a typical nonlinear transient heat conduction problem. If the analysis option is not set properly, it usually leads to the difficult convergence of calculation. To do this, the following settings need to be made:

(1) the analysis type of the three-dimensional temperature field is set up as transient analysis, that is, ANTYPE, 4. Activation time integration effect in nonlinear transient analysis;

(2) by Full Newton-Raphson (Newton Raphson) method, for every iteration, is a revision of the stiffness matrix and activation of adaptive function decline;

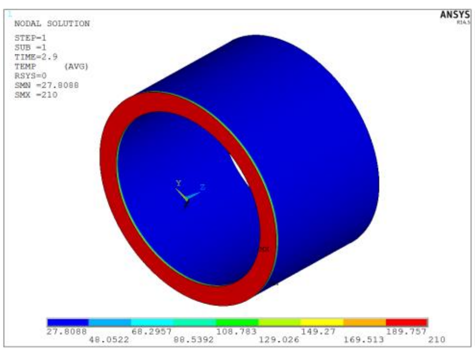
(3) turn on the automatic time step;

(4) open time step prediction;

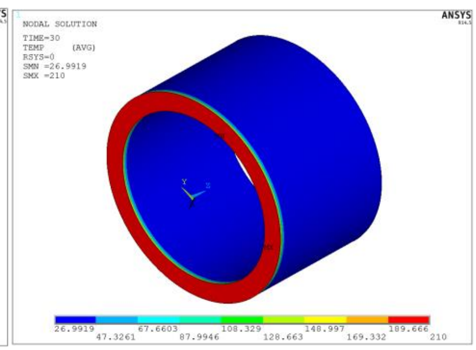
In addition, it is worth noting that due to the thermal analysis of the subject, it is necessary to set the ambient temperature. In this project, the ambient temperature is set at room temperature, or 28 degrees centigrade.

2 The result of temperature field analysis of PE pipe

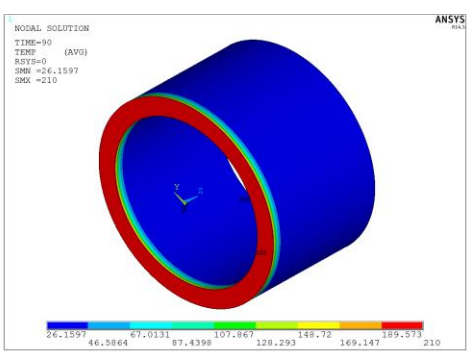
In this paper, the program is written in APDL language of ANSYS software. After the calculation is completed, the temperature field distribution of each load step is obtained by reading the load step of each time through the POST1 general postprocessor. Can also use ANSYS software to generate the animation display function, changes in the whole process of welding temperature field, can be clearly found along with the welding process forward, welding points on the temperature varies with time. The transient temperature distributions of 2.9s, 30s, 90s, 145s and X are respectively shown in the heating stages.



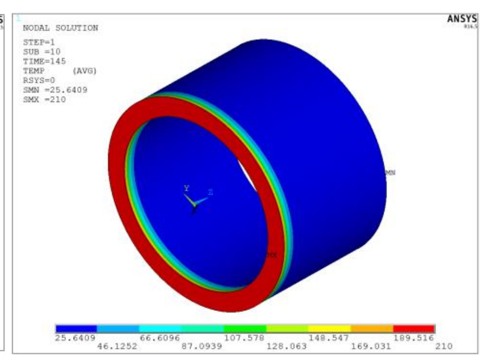
（a） At 2.9s moment



（b） At 30s moment



（ c） At 90s moment



（d） At 145s moment

Fig3. transient temperature distribution nephogram at heating stage

From the above figure 3, we can clearly see the dynamic process of the whole temperature field during the welding heating stage, and the shape of the isotherm profile changes slowly, that is, under the influence of heat conduction, it moves slowly to the non welding surface. In the heating process, the end face of the pipe is in contact with the heating plate, and the temperature of the heating plate is always stable at 210, so the highest temperature of the weldment is always 210 on the welding surface. The temperature of each point on the weldment varies with time, and the temperature is transmitted to the non welded surface, where the temperature has begun to appear on the weldment. With the development of welding process, the temperature distribution of weldment is more and more obvious.

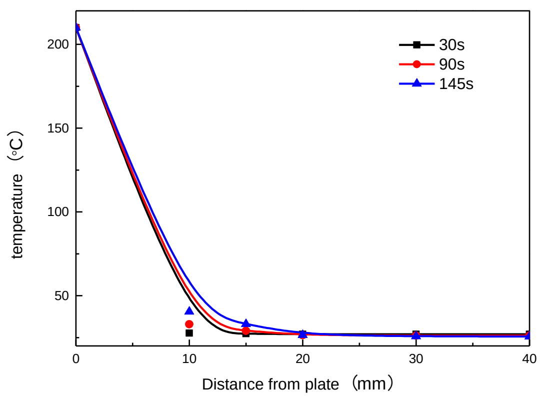


Fig4.axial temperature distribution of outer wall of pipe at different heating times

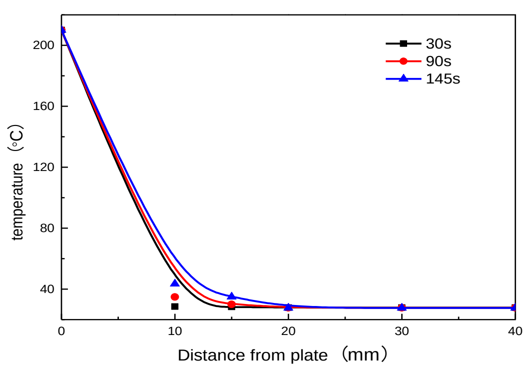


Fig5.axial temperature distribution of pipe wall at different heating times

Figure 4 and 5 respectively. The axial temperature distribution of different heating time of pipe wall, can be seen from the figure, the heating stage of heat affected zone is about 20mm along the axial direction within the range along the axial direction is greater than the temperature change at 20mm is slow, and close to ambient temperature. The thickness of the molten layer increases with the heating process, and the thickness of the melting layer is about 3mm-4mm (the temperature is greater than the axial distance of melting point 135) at 145s, which is close to the value of the analytic solution of the 4.3.2 section.

3 Comparison of analytical solutions and finite element results of temperature fields

At this stage, the temperature of the heating plate is over 210 and the melting point of the super material is very common. The object system not only has more than two kinds of media, but also has the boundary movement problem, that is, the interface between the two media is not static, but moving, which makes the problem more complicated. Although the finite element method and the finite difference method can be used to solve the complex boundary conditions, it is advantageous to establish an appropriate mathematical analytic model for qualitative analysis[8].

The analysis method is simple and can predict the temperature variation at the welded joint and the thickness of the molten layer at different heating temperatures. Therefore, to simplify the problem and establish an appropriate mathematical model is of great significance to optimize the welding process parameters and guide the practice of production.

3.1 heating stage physical model

Simplify the problem and make the following basic assumptions:

(a) the problem is simplified as one-dimensional heat transfer, which is only carried out along the axial direction of the pipe;

(b) assume that the melting process is an ideal process, that is, melting occurs at the point of melting rather than the melting;

(c) it is assumed that the average value of the phase density of the solid phase and the molten phase is the density of the phase transition interface;

(d) thermal convection caused by the expansion of the phase of the melt irrespective of the decrease in density;

(E) ignore the thermal effects of the outside surface and the environment;

(f) uniform temperature of the initial pipe.

3.2 The mathematical model of the heating phase

Based on above assumption, the problem can be simplified as: half infinite objects (the melting point is constant, the initial temperature ($135\,°C$), a constant, this is $28\,°C$), from the time =0 began applying constant temperature at =0 (surface is constant, this is $210\,°C$), for body temperature response.

Considering the influence of fusion heat, the melting phase (pin representation) and solid phase (foot marker) are described by two governing equations, and coupled at the solid-liquid interface.

The melt phase：

$$\begin{cases} \dfrac{\partial^2 T_l}{\partial {x_l}^2} = \dfrac{1}{\alpha_l}\dfrac{\partial T_l}{\partial t} \\ x = 0 : T_l = T_g \end{cases}$$

The solid phase：

$$\begin{cases} \dfrac{\partial^2 T_s}{\partial {x_s}^2} = \dfrac{1}{\alpha_s}\dfrac{\partial T_s}{\partial t} \\ x \to \infty : T_s = T_0;\ t = 0 : T_s = T_0 \end{cases}$$

The coupling conditions of solid-liquid interface：

$$\begin{cases} x = s(t) : T_s(X_s, t) = T_l(X_{l,} t) = T_m \\ x = s(t) : k_l(\dfrac{\partial T_l}{\partial x_l})\Big|_{x_l = X_l} - k_s(\dfrac{\partial T_s}{\partial x_s})\Big|_{x_s = X_s} = L\rho \dfrac{ds(t)}{dt} \\ \rho = \dfrac{\rho_l + \rho_s}{2} \\ \dfrac{X_l}{X_s} = \dfrac{\rho_s}{\rho_l} = \beta \end{cases}$$

Set the error function for the solution of the equation：

$$T_l(x,t) = T_g + Aerf(\frac{x_l}{2\sqrt{\alpha_l t}})$$

$$T_s(x,t) = T_0 + Berfc(\frac{x_s}{2\sqrt{\alpha_s t}})$$

Among them, A 、B is constant.The above two formula can be proved to be the solution of the equation. In the model k_s - Solid thermal conductivity, k_l - Thermal conductivity of molten phase, L - Latent heat of fusion, c_l - Melt phase specific heat at constant pressure, α_l - Thermal diffusivity of molten phase, α_s - Solid phase thermal diffusivity, ρ_l - Melting phase density, ρ_s - Solid density.

3.3 Solving the model

High density polyethylene pipe (HDPE) Φ 160SDR11at ambient temperature 28, the use of 210 heating plate heating 145s, to find the pipe along the axial temperature distribution：

Using Matlab software programming solution: $\lambda = 0.4575$， The temperature distribution along the axial direction of the molten phase and the solid phase is as follows:

$$\frac{T_l(x,t)-210}{135-210} = \frac{erf\left[x_l/(2\sqrt{1.22\times10^{-7}t})\right]}{erf(0.4575)}$$

$$\frac{T_s(x,t)-28}{135-28} = \frac{erfc\left[x_s/(2\sqrt{2.6\times10^{-7}t})\right]}{erfc(0.4575\sqrt{1.22\times10^{-7}/(2.6\times10^{-7})})}$$

That is :

$$T_l(x,t) = 210-155.4824erf(1431.5\frac{x}{\sqrt{t}})$$

$$T_s(x,t) = 28+70.3655erfc(990.5807\frac{x}{\sqrt{t}})$$

3.3 Comparison of analytical solutions and finite element results

These values are brought into the formula t=90s， 120s， 145s， x=0.001m， 0.002m， 0.003m, Calculated

The temperature distribution along the axial direction under heating time 90s, 120s and 145s is expressed in solid line in Fig.8. Then extract the finite element simulation results 90s, 120s, 145s and the corresponding position of the pipe node temperature, in figure 6 expressed in dotted lines.

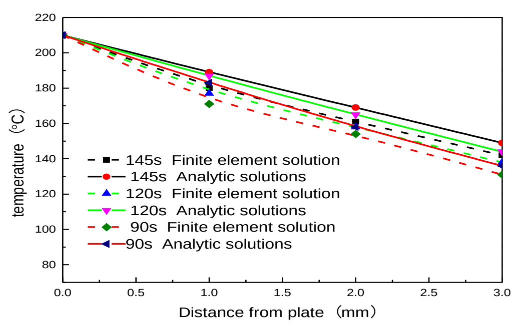


Fig6. Comparison of temperature distribution between finite element solution and analytic solution at different heating time

The results of the heating stage are compared with those of the finite element results, as shown in table 1.

Table 1 Comparison between the analytical results and the finite element results

heating time （s）	The axial distance from the center of the weld（mm）	Mathematical model for calculating temperature（°C）	Numerical simulation temperature（°C）	The error
90	0	210	210	0%
	1	183	171	6.6%
	2	158	154	2.5%
	3	136	131	3.7%
120	0	210	210	0%
	1	187	177	5.3%
	2	165	158	4.2%
	3	144	138	4.1%
145	0	210	210	0%
	1	189	180	4.8%
	2	169	161	4.7%
	3	149	142	4.7%

See from figure 6 and table 1, the welding temperature distribution near the joint finite element calculated with the analytical model calculated values are broadly consistent, but the finite element calculation value is always lower than the analytical solution, the error basically at 10 degrees or less. The main reason is the error on the assumption that the analytical solution was ignored in the thermal effect of the tube surface and the environment, and in fact in the process of welding pipe will heat and air convection, precisely the finite element calculation is considered in this, so the finite element calculation value is lower than the analytical solution is reasonable.

Conclusion

From the above figure 8 and table 1 shows that both the welding or welding temperature field distribution of the overall critical position temperature, hot melt welding simulation experiment of temperature field distribution and temperature field distribution is consistent, which verifies the correctness of the analysis of welding heating stage finite element simulation of hot melt polyethylene pipe.

The reason of individual position deviation is: (1) due to the actual welding environment, changes of convection tube wall heat transfer coefficient with position and time change, while simulated convective heat transfer coefficient by using the same settings; (2) calculating the latent heat need to determine and maintain a temperature range for the phase transformation the temperature range is very small, it is difficult to accurately calculate the latent heat.

Reference

[1] Sun Zengyin. Characteristics of polyethylene pipes and their applications in the field of water supply [J]. File, 2013 (9): 174 - 175.

[2] Song Hongmei. Technical measures for construction of polyethylene gas pipeline in municipal engineering [J]. Theoretical research on urban construction, 2015 (3): 2625 - 2626.

[3] Zhang Jun. Risk assessment and management of urban natural gas transmission and distribution network and application system [D]. Shanghai: Tongji University, 2007.

[4] Wu Jianjun, Ma Zeen. Plastic injection of CAD/CAE/CAM (IV) calculation and analysis of bending deformation of residual stress. Mechanical science and technology.1996, 15 (3): 468~472

[5] Li Haimei, GuYuanxian, ShenChangyu. Numerical calculation of thermal residual stress of injection molded parts. Journal of chemical.2000, 51 (3), 308 ~ 312

[6] Li Haimei, ShenChangyu, GuYuanxian. Uneven cooling on residual stress. China plastic.1999, 13 (6): 106 ~ 111

[7] Yu Ning. Fluid mechanics and thermal engineering [M]. Beijing: China Architecture and Building Press, 2005:118-124

[8] Li Wenping. HDPE tube hot plate welding and its failure mode [J]. plastics science and technology, 1989,6:8-12

Author:*ZHANG Ao-peng*, male, born in 1994, Geermu City Qinghai Prov. China, assistant engineer, master's degree, research direction: Safety Management of Oil and Gas Pipeline Transportation.