TELKOMNIKA, Vol. 11, No. 6, June 2013, pp. 3149 ~ 3158
e-ISSN: 2087-278X

The Research of Welding Residual Stress based Finite Element Method

Qinghua Bai
College of Mechanical and Electrical Engineering, Agriculture University of HeBei, Baoding, 071000, China.
Corresponding author, e-mail: 82469269@qq.com

Abstract

Welding residual stress was caused by local heating during the welding process, tensile residual stress reduce fatigue strength and corrosion resistance, Compressive residual stress decreases stability limit. So it will produce brittle fracture, reduce working life and strength of workpiece; Based on the simulation of welding process with finite element method, calculate the welding temperature field and residual stress, and then measure residual stress in experiments, So as to get the best welding technology and welding parameters, to reduce welding residual stress effective, it has very important significance.

Keywords: *residual stress, fatigue strength, working life, finite element method, parameters*

Copyright © 2013 Universitas Ahmad Dahlan. All rights reserved.

1. Introduction

We simulate residual stress field of welding process with finite element method on computer, and then weld workpiece as the same welding parameters and measure residual stress with hole-drilling method, last we compare the result between them, so we can know some way to improve welding parameters and welding quality.

The rest of the paper is organized as follows. Section II focuses on temperature field distribution of the flat with multi-welding seams. Section III is the description of the stress field distribution of the flat with multi-welding seams. Section C focuses on experiment. Finally, we end this paper with conclusion.

2. Temperature Field Distribution of the Model

2.1. The Reason of Producing Welding Residual Stress

It is quitely complex of the situation during producing welding residual stress. The value of residual stress is different with size of parts and welding methods and so on. The reason of welding residual stress often produce as three situations like this [1-4]:

During welding process, the metal parts were nonuniform heated and cooled in partly, so there are temperature gradient in it. In the heating process nonuniform temperature will prevent thermal expansion of metal and form local compressive plastic deformation area in heated region; While in the cool process, the metals in compressive plastic deformation area were prevent shinking. So it is mean reason which produce welding residual stress.

Constraint stress was produced by processing conditions before welding. With the different of conditions, nonuniform force, concentration gradient inside parts and displacement error in direction of grain, some area in parts produces different yield stress to nonuniform deformation and have some constraint stress.

Transformation stress was caused that metal parts were heated and cooled nonuniformed in some region and have some change in volume. The transformed length is in connection with value of residual stess.

In the article, we use ANSYS software to simulate multi-welding seams of a flat of 60CrMnMo material with limited elements method. With the time transmission in welding process, we can find temperature field variation, relif residual stress field, and can use proper welding parameters, so we can improve residual stress in maximum degree.

Received January 24, 2013; Revised April 1, 2013; Accepted April 17, 2013

2.2. Analysis Process of Temperature Field

With the development of computer technology, FEM (Finite Element Method) is using widely in engineer field as modern calculation method[5-6]. It can slove discrete domin as a assembly which limited elements linked each other in a certain way. Not omly FEM is using in structure analysis, but using to slove engineer field. In this, we use ANSYS software to simulate temperature field and stress field of the flat with multi-welding seams, the process we can see Figure 1.

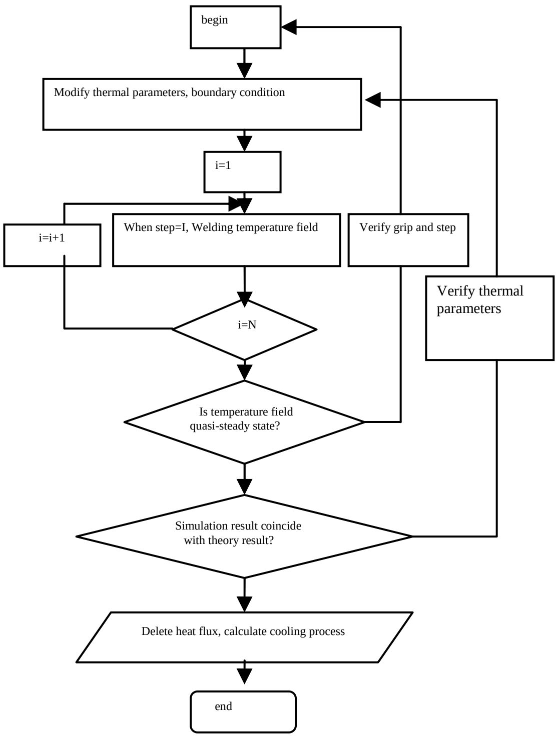

Figure 1. Flow Sheet of Welding Temperature Field

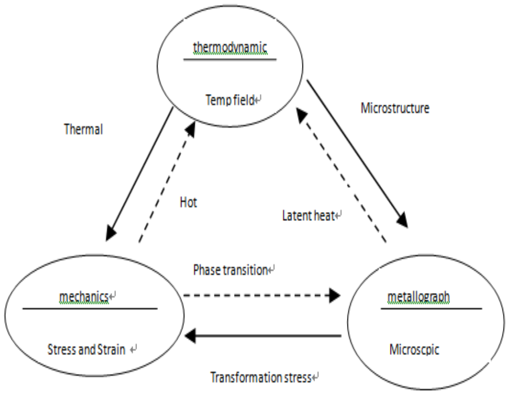

Figure 2. The Couple of Temperature, Phase Change and Thermal Stress

Welding is quite complacated process including arc physics, heat conduction, metallurgy, mechanics. Welding program includes welding of electromagnetic, heat transfer process, metal melting and solidification, cooling transformation, the welding stress and strain and so on [7]. We can see relations between them in Figure 2.

2.3. The Pre-post of Temperature Field Simulated by Ansys Software

It is a process that welding seam was heated quickly to high temperature, and then cooled suddenly. With the move of heat source, temperature and thermophysical properties of metal parts have rapid changes following temperature change, melting and phase transition happened at the same time. Welding temperature field is a typical non-linear transient problem, the equation [8]:

$$\frac{\partial}{\partial x}\left(k_{xx}\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(k_{yy}\frac{\partial T}{\partial y}\right)+\frac{\partial}{\partial z}\left(k_{zz}\frac{\partial T}{\partial z}\right)+q=\rho c\cdot\frac{\partial T}{\partial t} \qquad (1)$$

ρ— material density
c — specific heat
k — material conductor
q — welding resource intensity

The material of part is 60CrMnMo, its physics parameters see Table 1. Dimension is 120mm x120mm x8mm, welding seam is in the middle of it. Being a axial symmetry, we just need analysis half of part.

Table 1. Thermo Physical Performance Parameter

Temp °C	200	400	600	800	1000	1200	1400	1600
Conductivity coefficient(W/mm. °C)	0.0419	0.0394	0.0323	0.0291	0.0265	0.0294	0.0294	0.0294
Density(g/mm3)	7.85	7.85	7.85	7.85	7.85	7.85	7.85	7.85
Specific heat c(J/kg. °C)	502	536	586	695	674	670	670	670
Exchange coefficient (W/mm. °C)	18.85	33.44	55.95	88.92	134.6	195.4	273.4	370.9

2.4. The Boundary Condition of the Model

We look part's bottom surface, up surface, and handle the boundary condition as exchange heat boundary condition.

$$k \cdot \frac{\partial T}{\partial n} + \alpha \left(T - T_0\right) = 0 \quad (2)$$

n — normal direct of boundary surface
α— surface exchange coefficient
T0 — surrounding medium temperature (20°C)

The typical heat sources are point source, surface source, solid surface. We use Guass surface source here, the mathematic model is in Figure 3 [9]:

$$q = q_m \exp\left(-3\frac{r^2}{R^2}\right) \quad (3)$$

qm— maximum heat flux
R — heat radius of electric arc
r — the distance to center of heat source

The problem of enthalpy happened from liquid to solid, it absorb heat in this process. It may define different enthalpy with variable temperature to solve in ANSYS software [10].

$$H = \int \rho c(T)\, dT \quad (4)$$

C(T)— specific heat

In the model, we need to mesh the model using finite element method and to increase analytical accuracy. We use solid45 element type to mesh dense grid on the place near welding seams, so can get more accurate data. We make a model suah as Figure 4.

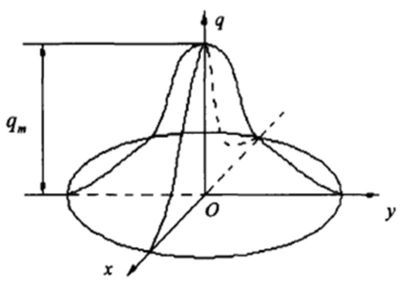

Figure 3. Heat Flux of Gauss Distribution

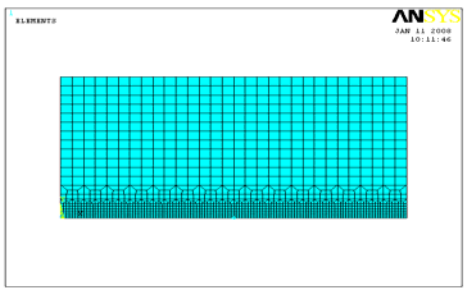

Figure 4. Finite Element Illustration of Model

3. Analysis Theory of Temperature Field

3.1. Purpose of Analysis Theory of the Model

Because the welding process is highly centralized process of instantaneous heat input, after the welding process and welding will have a lot of welding stress and deformation. Welding stress and deformation calculation is based on the analysis of the welding temperature field, microstructure transformation considering the welding area influence on the stress and strain. At present, the research of welding stress and strain theory mainly include thermal elastic-plastic analysis, analysis of inherent strain, viscoelastic plasticity, considering the phase transition and thermal stress coupled effects, etc.

Welding stress and strain in the material nonlinearity, geometrical nonlinearity of nonlinear problems. Considering the welding thermal stress of the complexity of the process, in order to the accuracy of the calculation, the welding thermal stress field as material nonlinear transient problems. Selection of elastic-plastic mechanics model, calculated by the increment theory.

Thermal elastic-plastic problem is a problem of thermodynamics, a thermodynamic system of welding materials, the free energy density is not only related with the strain, but also related to the temperature. Look from the energy, the heat energy input in the welding material temperature rise at the same time, also due to the expansion of the structure deformation and consumption part of doing work. At this time, to increase in the heat balance equation of stress related. Strictly speaking, the welding temperature field and stress field is of mutual coupling, but the coupling effect in addition to the individual circumstances, usually is very small, and near the weld temperature change is very big, material of all kinds of physical properties also changed accordingly, this effect compared with the coupling effect is much larger, therefore, on the basis of welding thermal elastic-plastic analysis, do the following assumptions [11]:

1. Material yield to Von Mises yield criterion
2. The behavior of the plastic zone to plastic flow standard and strengthen the standards
3. Elastic strain and plastic strain and temperature strain are inseparable
4. Temperature on the mechanical properties, stress and strain in the small incremental changes in linear time.

3.2. The Types of Nonlinear Problems

Because of the complexity of the nonlinear problems, using the analytical method can get the answer is very limited. With limited linear analysis method in the successful application, its application in nonlinear analysis also have made great progress, and have gained a lot of different types of practical problems solving scheme, has a group of large general nonlinear analysis program into practical application.

There are three nonlinear problems which include material nonlinear problem, the geometric nonlinear problem and boundary nonlinear problem.

Material nonlinear problem: In this problem, physical equation of stress and strain relation is not linear. Such as discontinuous changes the shape of the structure of the parts there is stress concentration, when the load reaches a certain value, the region first into plastic, at this moment although most other areas of the remain flexible structure, but in the region of linear elastic stress and strain relationship is no longer applicable. As under high temperature condition for a long time the structure of the creep change will happen, namely under the condition of the load or stress remains unchanged, deformation or strain is still with the progress of the time and continue to grow, it is not linear elastic properties of equation can describe. Both plastic and creep is recovery of elastic deformation, and stress the relationship between them is nonlinear.

The geometric nonlinear problem: This problem is the feature of structure in the loading process of large displacement and rotation. Such as the large deflection of plate and shell structure, buckling and buckling problem. Materials may remain for linear elastic state at this time, but the structure of the balance equation must be established after the deformation condition, in order to consider the effect of deformation on the balance. At the same time due to the big displacement and rotation of actual, simplify the geometric equations can no longer for the linear form, namely strain displacement must be included in the expression of the second item.

Boundary nonlinear problem: This problem the most typical example is the contact of two objects and collision problem. They contact boundary and scope, and the position of contact surface force distribution and taken first cannot be given, depend on the solution of the problem to determine. Another example is the size of the force on the boundary of the external force and direction depended on the deformation, the nonlinear effect on pressure distribution on the surface of the thin-walled structure, for example, when the structure deformation, its effect will be the size and direction of change, and the geometric nonlinear problem in need at the same time and the load changes.

Material nonlinear problems can be divided into two categories. A class is not dependent on the elastic-plastic problem of time, its characteristic is, after loading, material deformation and no longer over time changes immediately. Another kind is dependent on the time of the adhesive (elastic and plastic), its characteristic is that after loading, the material not only occur immediately the corresponding elastic deformation (plastic), and the deformation and continue to change over time. Under the condition of load remain unchanged, as a result of material viscosity and continue to grow as creep deformation. Under the condition of deformation, on the other hand, remain unchanged, because the material viscosity and attenuation is called the stress relaxation.

3.3. The Solution of Nonlinear Problems

Solving the nonlinear equations has many methods, such as direct iterative method, Newton raphson method and incremental method, minimization method and variable step extrapolation, etc.

It is the basic idea of using piecewise linear instead of nonlinear.

$$\begin{cases} \left\{ \Delta T_{r+1} \right\} = -\left[\frac{\partial}{\partial \{T\}} \{\psi(T)\}_r \right]^{-1} \{\psi(T_r)\} \\ \Delta T_{r+1} = T_{r+1} - T_r \end{cases} \tag{5}$$

Nonlinear equations near the Tr into an approximate linear equations

If we know r value approximation, then calculate the unknown quantity r+1 time value, multiple iterative until convergence.

Direct iteration method: For the nonlinear problem, if the calculation of direct iteration method, first need to assume that an initial test solution, it usually can be obtained from the solution of the linear elastic problem first. Followed by direct iteration method of need each iteration calculation and form new is essentially a secant stiffness matrix of the coefficient matrix, and inverse calculation, it is only applicable to has nothing to do with the deformation history of the nonlinear problem, such as nonlinear elastic problem can use the analysis of the deformation theory of the elastoplastic problems and steady-state creep problem, for this kind of problem, stress can be determined by the strain, can also be determined by the displacement. For loading path changing or unload and repeated loading elastoplastic problems such as cannot use direct iteration method, must take advantage of incremental theory are analyzed.

Newton-Raphson methed [12]: The method of solving process can be expressed as follows. Under normal circumstances, it has good convergence, each iteration process to form a new tangent and inverse matrix. In order to overcome N - R method for each iteration needs to be formed and inverse new tangent matrix of trouble, can use the modified scheme as Modified Newton Raphson method, we can see as Figure 5.

Incremental method: For elastic-plastic, creep, etc material nonlinear problems, usually caused by stress as deformation, deformation theory will not be used at this time, must be incremental theory are analyzed.

For this three solving methods, due to the Newton-Raphson method has better convergence and higher rates of convergence, make it become an important approximation method for solving all kinds of nonlinear problems. In ANSYS in solving nonlinear problems are also choose this way.

In the process of simulation, we divide welding seam area into elements alone, use the method of dead and live element, kill the second welding seam firstly, then put Guass heat source which writes with APDL language on first welding seam, live the second welding seam and put Guass heat source on it. So, we can get temperature distribution and have a good base for stress analysis, easily to decide welding paramenters.

3.4. Temperature Field Distribution of Physics Model

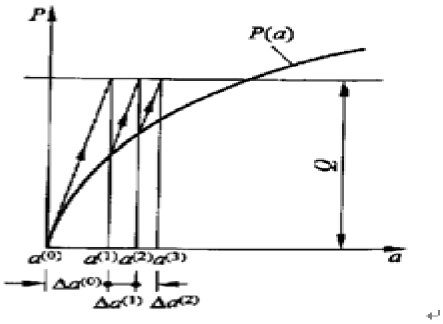

Figure 5. Newton-Raphson Method

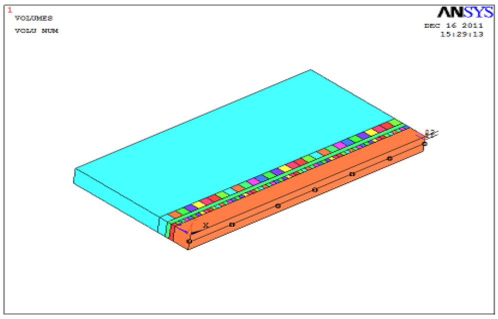

Figure 6. Points A-G in the Model

In the analysis, the welding current is 250A, welding voltage is 25V, welding time is 6S, and welding efficiency is 0.9. When the time is from 0S to 8440S, We chose seven points which are in the middle of part including A(0, 0, -0.004), B(0.02, 0, -0.004), C(0.04, 0, -0.004), D(0.06, 0, -0.004), E(0.08, 0, -0.004), F(0.1, 0, -0.004), G(0.12, 0, -0.004), See Figure 6:

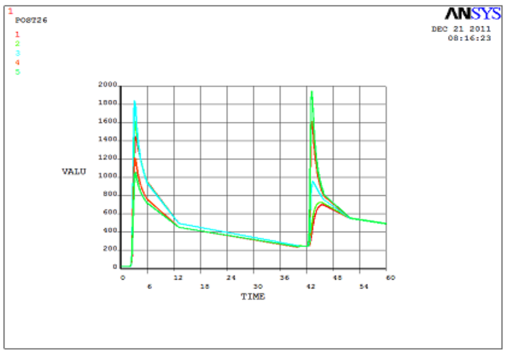

Figure 7. 0~8500s Points A-G Distribution of Temperature

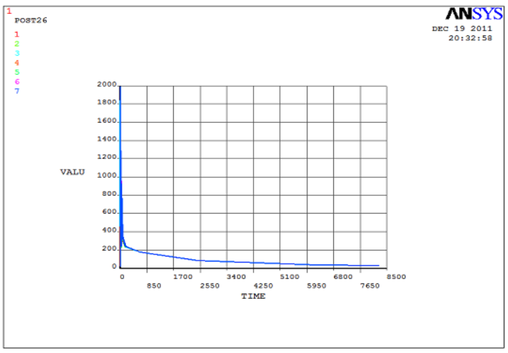

Figure 8. 0~40s Points A-G Distribution of Temperature

From Figure 7 and Figure 8, the highest temperature is 2102°C. In this figure we can know, the temperature in the welding seam is highest than other points. More far away welding seam, the lower temperature is. We also can know, the temperature of second welding seam is 2102°C which is high almost 100°C than the first one.

4. Stress Field Distribution of Model

We simulate the field of weld stress with indirect method. After simulated transient temperature field and satisfied with classical theory, we can begin to simulate the field of weld stress.

4.1. Limited Elements Equation of Stress Field Simulation

For some structure element, there a balance equation as following:

$$\{dF\}^e + \{dR\}^e = [K]^e\{d\delta\}^e \quad (6)$$

In this equation:

$\{dF\}^e$ - The increment of nodal force in element

$\{dR\}^e$ - Equivalent nodal increment of element initial strain caused by temperature

$\{d\delta\}^e$ - Increment of nodal displacement

$[K]^e$ - Element stiffness matrix

4.2. Mathematic Model of Stress Field in the Flat with Multi-welding Seams

It needn't build a new model again and use the mesh model which used in temperature field, first come in preprocessor PREP7 and read output of temperature field, secondly transform heat elements (SOLID70, PLANE55) into structure elements(SOLID45, PLANE42), then define the mechanical properties of materials parameters such as Table 2. Considering the welding deformation is great strain, it choices the option of plastic analysis as biaxial isotropic (BISO), assigns yield stress and shear modulus which change with temperature.

Table 2. Mechanical Performance Parameter of 60CrMnMo

Temp(□)	200	400	600	800	1000	1200	1400	1600
Poisson ratio μ	0.3	0.3	0.3	0.3	0.3	0.3	0.3	0.3
Linear expansion coefficient ×10-6 α (1/□)	13.9	16.1	16.7	16.7	16.7	16.7	16.7	16.7
elastic modulus×105E(Mpa)	2.00	1.29	0.678	0.497	0.459			

4.3. The Determination of Boundary Conditions

Welding process belongs to the large deformation problems, set analysis options, the large deformation and large strain options, Full Newton-Raphson method of equilibrium iteration and activation of the adaptive function decline, open the automatic time step and time step prediction to speed up the convergence. Time step should be the same as the setting of the temperature field. Loading, read the node temperature of the thermal analysis and specify the corresponding time point or load step can, reference temperature and thermal analysis in the setting of initial temperature.

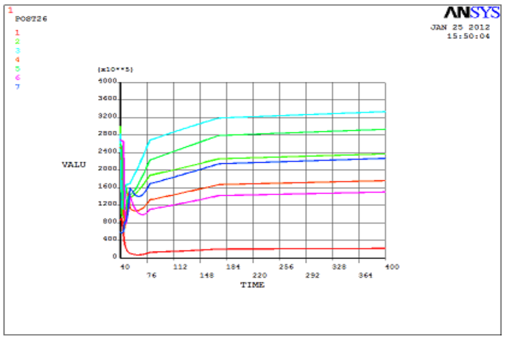

Figure 9. Points A-G Distribution of Stress in 40s

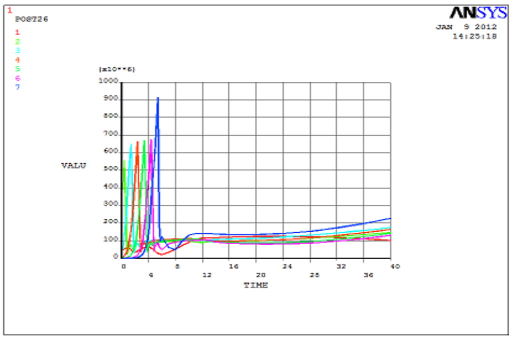

Figure 10. Points A-G Distribution of Stress in 400s

The live and dead element method [13] was used in this paper. Live and dead element method in welding the weld units "kill" at the beginning, and each step in the calculation of thermal stress, the corresponding calculation results of temperature field of choice, melting point than melting unit to the "dead", and below the melting point of the unit, and more than melting point unit will not melt the "activation", the process and change unit properties basically do not have too big difference, this is "live and dead" unit is used to simulate the welding problem. When using this function, should pay attention to the details of the process, under normal circumstances, the program "kill" and "active" element was carried out one by one, in the "kill" and "activate" unit, if you want to put all of the selected unit "kill" and "activate", should choose

Full Newton-Raphson algorithm calculated transient problems, and pay attention to when using node temperature to select units, if there is no choice, should choose the unit of this program to ignore the selected unit of orders, and carry out a command, it will "kill" or "activation" other should not be "kill" or "activation" of the unit.

Define the boundary conditions are mainly constrained freedom of welded components, according to the specific situation. Loading displacement boundary conditions should be to avoid the rigid displacement in the process of finite element calculation and cannot produce serious obstacles in the process of welding stress and deformation. Defined reference temperature, if there is no preheating before welding, it is at room temperature, conversely for preheating temperature, this paper does not consider preheating, the temperature is 20 degrees Celsius.

Calculating stress when the applied load is the results of the temperature field calculation, so just read temperature of each node can, here will be the time step and the temperature field calculation, so as to read and improve the precision of temperature load.

4.4. Stress Field Distribution of Physics Model

After analyzing temperature field distribution of the model, we use the software to analyze the stress field distribution. We can have several simulating results of stress in Figure 9 and Figure 10. In Figure 9, we can see stress distribution of those points which are in the middle of part (A, B, C, D, E, F, G). When welding the first time, the point F have a biggest stress about 91.56Mpa, the stress will increase when welding it second time after cooled 40sec.

From Figure 10, in welding process of first 6 sec(40sec~46sec), we can see the stress of points which first increasing suddenly to 30Mpa, and after a little second descending, then increasing to 32Mpa. We also can know the second seam stress less than the first one, and the stress will increase with the process of cooling.

5. Experiments on Temperature and Residual Stress

After simulated to the temperature field and stress field on the computer, we do some experiment to verify the result.

5.1. Experiments on Temperature Field

We use contactless temperature sensor to measure temperature field. Limited by some problem, we chose NH series contactless temperature sensor which made in China, and may measure temperature from 400°C to 1200°C. Because the welding temperature is higher than its measuring range, so we measure the place which is 40mm behind welding torch and the temperature is between 400°C to 1200°C. After measured temperature, we can have the Figure 11. From it we can see, the experiment result conform to the simulation of welding temperature field.

5.2. Experiments on Residual Stress Field

We use hole-drilling method to measure stress field on the welding parts. It has some advantages such as having very small destructive, easy operating, so widely used in these fields.

We can calculate main stress on some point using next formula.

$$\sigma_1 = \frac{\varepsilon_1(\mathrm{A} + \mathrm{B}\sin\gamma) - \varepsilon_2(\mathrm{A} - \mathrm{B}\cos\gamma)}{2\mathrm{AB}(\sin\gamma + \cos\gamma)}$$
$$\sigma_2 = \frac{\varepsilon_2(\mathrm{A} + \mathrm{B}\cos\gamma) - \varepsilon_1(\mathrm{A} - \mathrm{B}\sin\gamma)}{2\mathrm{AB}(\sin\gamma + \cos\gamma)} \tag{7}$$
$$\gamma = 2\psi = \mathrm{tg}^{-1}\frac{\varepsilon_1 - 2\varepsilon_2 + \varepsilon_3}{\varepsilon_1 - \varepsilon_3}$$

A, B are strain relief parameters, we can have as next equation:

$$A = -\frac{(1+\mu)R^2}{2Er_1r_2}$$
$$B = -\frac{2R^2}{Er_1r_2}\left[1 - \frac{(1+\mu)(r_1^2 + r_1r_2 + r_2^2)R^2}{4r_1^2r_2^2}\right] \quad (8)$$

E- elastic modulus, 206GPa
μ- poisson ratio, 0.3
ρ- density, 7.8x103 kg/m3

We drill five holes on the parts, the position is A(28,5,0), B(38,3,0), C(68,5,0), D(105,10,0), E(120,5,0), then calculate stress of these points in Table 3.

Table 3. The Stress of Points

	A	B	C	D	E
σ_x (MPa)	-25.402	-29.55	-45.729	-37.64	-104.28
σ_y (MPa)	-27.062	-74.931	-84.587	-104.27	-160.82

In order to compare the results, we draw the stress which attain to experiment on the result which attain to simulation on the computer, such as Figure 12 and Figure 13.

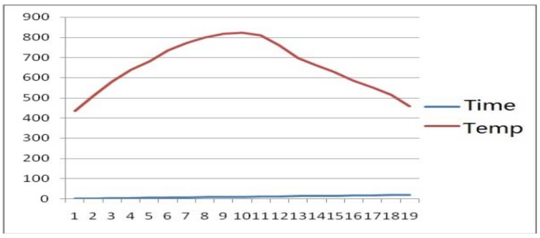

Figure 11. The Graphs of Welding Temperature Field

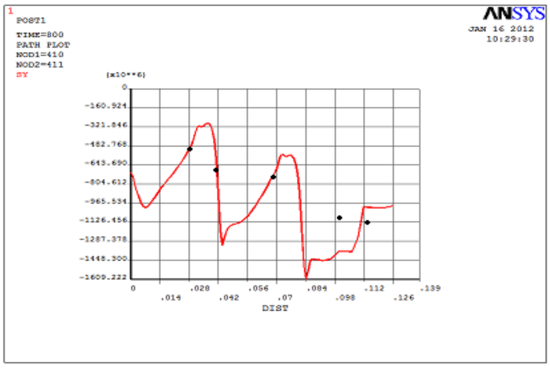

Figure 12. Transverse Stress of the Points

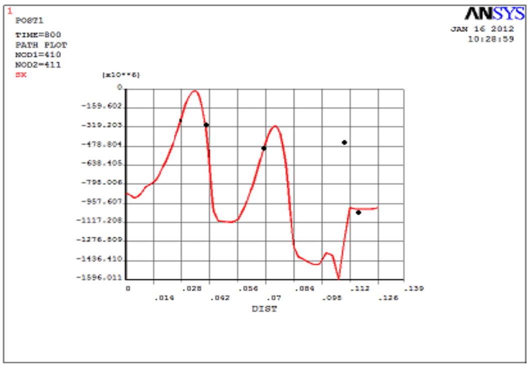

Figure 13. Longitudinal Stress of the Points

From Figure 12 and Figure 13, we can see the stress of points including A, B, C, E is reasonable, the stress of point D has a big tolerance.

6. Conclusion

We proposed a feasible dynamic simulation method on 3D welding temperature field, stress field has been established, which provides theory foundation and instruction on optimizing the welding technology and parameters, and provides theory foundation for the weld trailing peening technology at the same time. The distribution of welding temperature field we had is coinciding to measured result.

References

[1] Wang Jianhua, Lu Hao. *Welding residual stress formation mechanism and eliminate the principle of some issues to discuss*. Welding. 2002; 23(3).
[2] Tian Xitang. *Welding structure.* Harbin Engineering University, China Machine PRESS. 1996.10.
[3] Yonetani Shigeru. *Residual stress and the formation of the countermeasures*. Japan.
[4] Wang Kuanfu. *Pressure vessel welding structure engineering analysis*. Chemical industry press. 1998: 20-50.
[5] José Luis Manzanares Japón, Ignacio Hinojosa Sánchez-Barbudo. Non-linear plastic analysis of steel arches under lateral buckling. *Journal of Constructional Steel Research.* 2011; 67(12): 1850-1863.
[6] Zhu ZQ, Wang P, Tuo SF, Liu ZA. Structured/unstructured grid generation method and its application. *Acta Mechanica.* 2004; 167(3-4): 197-211.
[7] WANG Xuewen, YANG Zhaojian, Liu Hunju. Finite element analysis on double-telescopic prop of hydraulic support. *Telkomnika.* 2013; 11(1).
[8] WeiBing Zhu, HeShun Wang, Lin Dong. Influencing Factors of Thermal Deformation on Hydrostatic Pressure Mechanical Seal and Optimization of Rotating and Stationary Rings. *Telkomnika.* 2013; 11(2).
[9] Fenhua Li, Jian Xing, Yuan Liu. *Thermal Analysis and Stress Analysis of the Heat-Exchange Pipe Based on ANSYS*. In Proceeding of 2011 Fourth International Conference on Information and Computing. 2011; 283-285.
[10] Zhang Xiao-ning, Yang Peng. Finite Element Analysis of High Modulus Asphalt Pavement. *International Journal of Advancements in Computing Technology, Advanced Institute of Convergence Information Technology.* 2012; 4(2): 72-79.
[11] Fu Jianbao, Luan Maotian, Yang Qing, Nian Tingkai. Three-Dimensional Finite Element Analysis of Wall Pressure on Large Diameter Silos. *Journal of Convergence Information Technology, Advanced Institute of Convergence Information Technology.* 2010; 5(7): 120-128.
[12] Xianzhang FENG. Analysis of Field of Stress and Displacement in Process of Meshing Gears. *International Journal of Digital Content Technology and its Applications, Advanced Institute of Convergence Information Technology.* 2011; 5(6): 345-357.
[13] Yu-Jie Sun, Yong Zang, Qing-Yu Shi. *Sensitivity Analysis of Some High Efficiency Computational Methods for Welding Process Numerical Simulation.* In Proceeding of 2009 Second International Conference on Information and Computing Science. 2009; 4: 92-95.