A study of residual stresses in the surface hardening of a blade mould by high frequency induction heating

W.-B. Kim and S.-J. Na*
Department of Precision Engineering and Mechatronics, School of Mechanical Engineering, Korea Advanced Institute of Science and Technology, 373-1 Kusong-dong, Yusong-gu, Taejon, 305-701 (South Korea)

(Received January 22, 1993; accepted in final form February 24, 1993)

Abstract

High frequency induction heating is a method of surface treatment which restricts the hardening area using the skin effect. Since the penetration depth of the magnetic field in the workpiece is dependent on the frequency, the required size of hardening area can be obtained by selecting an appropriate frequency. High frequency induction heating is able to harden a large area at once, in which the shapes of the coil and workpiece and the distance between them are important factors for the hardening area shape and the distribution of residual stresses. In this study, the transient heat flow and thermal stress were analysed for the high frequency induction surface hardening of a blade mould by using the modified two-dimensional finite element method. Besides the volume change in the phase transformation, the effect of transformation plasticity was also considered as an additional strain in the numerical analysis of the high frequency induction hardening process.

The hardening area was fairly uniform in the mould surface except around the corner where the distance between the coil and workpiece was slightly larger than on the other parts. The thermal stress was induced mainly by the temperature gradient and martensitic phase transformation, while the latter was found to have a greater influence on the residual stress than the former. Simulation results revealed that compressive residual stresses occur in the hardening area, while the maximum tensile residual stress occurs near the boundary of the hardened zone.

1. Introduction

High frequency induction heating is widely used for hard facing and local heating because of its high workability, simplicity and economic advantages. This technique for surface hardening is easily adaptable to the precise control of temperature of the workpiece surface and process automation, because the hardening depth is simply adjustable through the selection of the appropriate frequency. When compared with the competitive laser beam hardening process, the potential benefit of this process is the capability to treat a wide area of the workpiece at once. The shape of the workpiece and induction coil, given by parameters such as the workpiece thickness and the surface curvature, has great influence on the distribution of the hardening area and residual stresses. The local heating of the workpiece causes complex thermal strains and stresses which finally lead to residual stress and distortion. Since the residual stress greatly influences the mechanical properties of the workpiece surface, e.g. corrosion resistance, wear resistance etc., the prediction of the residual stress and its distribution is one of the major concerns of this hardening process.

In the present work, the high frequency induction hardening process of a turbine blade mould was analysed. The finite element method was used to determine the heat flow and thermal stress during surface hardening of the turbine blade mould. Calculations of the transient temperature distribution were based on the two-dimensional finite element model, while the analysis of the thermal and residual stresses was carried out using a modified two-dimensional finite element model which satisfies the self-equilibrium of the resultant force in the longitudinal direction [1]. The effects of the volume change and plasticity in the phase transformation were also considered in the finite element analysis of this hardening process [2].

2. Finite element formulation

2.1. Heat transfer analysis

2.1.1. Formulation of heat transfer analysis

A typical cross-section of the turbine blade mould is represented in Fig. 1, which shows its upper and lower part. The shape of the turbine blade mould was derived
W.-B. Kim, S.-J. Na / Residual stresses in surface hardening of a blade mould

Fig. 1. Schematic diagram of the mould section.

from a real turbine blade which has approximate surface equations as follows (see Fig. 2): upper surface

\[ Z_{u}(y) = 26.2 - 0.1086(y - 3.2) - 0.035(y - 3.2)^2 + 0.000034(y - 3.2)^3 \]  

(1)

and lower surface

\[ Z_{l}(y) = 12.58 + 0.1674(y - 3.2) - 0.0342(y - 3.2)^2 + 0.0000799(y - 3.2)^3 + 0.0000561(y - 3.2)^4 \]  

(2)

Calculation of the transient temperature distribution was based on the quasi-stationary condition, which occurs when the heat source is moving at a constant speed on a regular path and the end effect resulting from either initiation or termination of the heating process is neglected. The temperature distribution is then stationary with respect to the moving coordinate system whose origin coincides with the point of application of the heat source. Considering the solution domain in Fig. 2, the temperature at any point in the workpiece is expressed functionally as follows:

\[ T(x, y, z, t) = T(x - v_h t, y, z) \]  

(3)

where \( v_h \) is the speed of the moving heat source. Thus, if the transient temperature distribution at any one section of the workpiece is given, for example at \( x = 0 \), the temperature at any other section can be determined by an appropriate shift of the time scale as follows:

\[ T(x, y, z, t) = T(0, y, z, t - x/v_h) \]  

(4)

The problem is therefore reduced to finding the two-dimensional unsteady-temperature field at a section normal to the heating line. A planar analysis may be used for this purpose when the speed of the heat source, compared with the characteristic diffusion rate of the material, is sufficiently high that the amount of heat conducted ahead of the heat source is very small relative to the total heat input. In this case, the heat flow across any infinitesimally thin slice of the workpiece normal to the heating line is assumed to be negligible compared with the heat being dissipated within the slice itself. Two-dimensional thermal analysis, normal to the direction of the heating line, was thus used in the present study. The governing isoparametric finite element equations for this type of heating process problem can be written in matrix form as follows [3]:

\[ [C][T] + [K][\dot{T}] + [F] = 0 \]  

(5)

where \([C]\) is the conductance matrix, \([K]\) the stiffness matrix, \([F]\) the load vector and \([T]\) the nodal temperature vector. The numerical scheme employed to integrate eqn. (5) was based on the Crank–Nicholson method.

2.1.2. Boundary conditions and heat generation for temperature analysis

The boundary conditions for the temperature analysis of the high frequency induction hardening for the turbine blade mould are given as follows (see Fig. 3):

\[ k \frac{\partial T}{\partial n} = h(T_i - T_a) \]  

(6)

on the surface, where \( k \) is the heat conductivity, \( n \) the outer normal vector to the boundary, \( h \) the heat convection coefficient, \( T_a \) the ambient temperature and \( T_i \) the surface temperature on the boundary.
Fig. 3. Boundary conditions of the solution domain for thermal analysis: (a) upper part; (b) lower part.

The appropriate mesh size is necessary to give good accuracy in the analysis results using little computing time. The more severe the heat generation rate is in a region, the finer the elements required. Thus variable meshes, which are very fine near the mould surface and expand gradually away from it, are indispensable to obtain good accuracy with the limited elements. This can also reduce the error in calculations of the amount of heat generation, which is very large in the surface zone and decreases steeply away from it. The finite element mesh system adopted is shown in Fig. 4.

In high frequency induction hardening, the distribution of the heat generation is an important factor in solving the temperature distribution, because the heat generation rate in the interior of the workpiece varies with the distance from the surface, the distance between the coil and workpiece etc. The distance between the coil and workpiece depends on the coil shape. In this study, the coil shape assumed was such that the distance between the workpiece and coil had a constant value except at the corner of the mould. The shape of the high frequency induction coil, represented in Fig. 3, can be approximately expressed as follows: upper part

\[ Z_1(y) = 31.2 - 0.090(y - 31.2) - 0.029(y - 31.42) - 0.000089(y - 31.2)^2 \]

and lower part

\[ Z_2(y) = 17.58 + 0.1674(y - 31.2) + 0.0347(y - 31.42) + 0.0006066(y - 31.2)^3 + 0.0001801(y - 31.2)^4 \]

The heat supplied by the high frequency induction is generated inside the workpiece, and the amount of heat generation is proportional to the strength of the current density which is determined by the magnitude of the magnetic field. The current density \( J(z) \) can be found using the following formula [4]:

\[ J(z) = J_{\text{on}} \exp(-xz) \cos(\omega t - xz) \]

where \( J_{\text{on}} \) is the peak value of the strength of the current density at the workpiece surface, \( x \) is a constant \((= \mu \omega / 2r)\), \( \mu \) is the permeability of the workpiece, \( r \) is the resistivity of the workpiece and \( \omega \) is the angular velocity \((= \omega / 2\pi)\).

Based on eqn. (9), the depth of penetration \( \delta \) of the magnetic field in the workpiece can be expressed as follows [4]:

\[ \delta = \left( \frac{2r}{\mu \omega} \right)^{1/2} \]

According to eqn. (10), the depth of penetration of the
magnetic field is smaller, the higher the frequency used. In this calculation, the magnitude of the heat generation of the element was determined as the average value for ten subelements into which each element was divided in the depth direction. The load vector of the heat generation was computed for the whole elements and added to the global load vector \([F]\) in eqn. (5).

Another factor influencing the amount of heat generation is the distance of the coil from the workpiece surface. The relationship between the current density and the distance of the coil from the workpiece surface can be expressed in the following form (see Fig. 5):

$$J_s = \frac{l}{\pi h (1 + x_c/h)^2}$$

(11)

where \(J_s\) is the current density at the surface, \(l\) the current in the coil, \(h\) the height of the working coil from the workpiece surface and \(x_c\) the distance on the workpiece from the coil centre.

Then the distribution of the generated heat can be expressed by the following formula:

$$P = \rho_m [1/(1 + (x_c/h)^2)]^2$$

(12)

where

$$\rho_m = \frac{\rho}{\delta} \left( \frac{1}{\pi h} \right)^2$$

(12a)

is the maximum power density generated by the induced current in the workpiece.

By integrating eqn. (12) with respect to \(x_c\), the total power between \(+x\) and \(-x\) can be calculated to be

$$P_{\pm x} = \frac{rl^2}{\pi^2 \delta h} \left[ \tan^{-1} \left( \frac{x}{h} \right) \right]_0^x$$

(13)

giving the total power input to the workpiece, when \(x = \infty\), as follows:

$$P_{\text{total}} = \frac{rl^2}{\pi^2 \delta h} \left[ \frac{1}{2} \left( \frac{rl_0}{h} \right)^{1/2} \right]$$

(14)

2.2. Thermal stress analysis

2.2.1. Formulation of thermal stress analysis

Material subjected to the thermal cycle of high frequency induction heating is postulated to behave mechanically as an initially isotropic, elastoplastic, strain hardening continuum, such that a component \(\varepsilon_{ij}\) of the total strain is given by the following formula:

$$\varepsilon_{ij} = \varepsilon_{ij} + \varepsilon_{ij} + \varepsilon_{ij}$$

(15)

where \(\varepsilon_{ij}\), \(\varepsilon^p\) and \(\varepsilon^{th}\) are the components of the elastic, plastic and thermal strain respectively.

Considering the virtual work equation for an isoparametric finite element assemblage and the constitutive equations for a thermoelastic and plastic material with isotropic hardening, all the equations can be expressed in matrix form as follows:

$$\sum_{m=1}^n \int \mathbf{[B]}^T [\sigma] \, dV = [R]$$

(16)

$$[\sigma] = [C][\varepsilon] - [\varepsilon^p] - [\varepsilon^{th}]$$

(16a)

$$[\varepsilon] = [B][U]$$

(16b)

where \([B]\) is the total strain–displacement transformation matrix, \([C]\) the material stiffness matrix, \([U]\) the nodal point displacement vector, \([R]\) the nodal point external load vector, \(n\) the number of elements in the assemblage, and \(v\) the volume of the \(m\)th element. Although eqn. (16a) is valid at any point in the structure or continuum, only the stresses and strains at the element integration points will be of interest. Substituting eqns. (16a) and (16b) into eqn. (16) results in the following equation:

$$[K][U] = [R] + \int_v \mathbf{[B]}^T [C][\varepsilon] + [\varepsilon^{th}] \, dV$$

(17)

where

$$[K] = \int_v [B]^T [C] [B] \, dV$$

(17a)

is the stiffness matrix and \(v\) the volume of the solution domain.

2.2.2. Boundary conditions for stress analysis

For analysing thermal and residual stress, the plane strain boundary condition has been widely used along the longitudinal direction (x direction) in the workpiece, because of its simplicity and ease in modelling. However, the model using the plane strain boundary condition could not satisfactorily describe the thermomechanical behaviour of real situations unless the workpiece could be set up between two rigid constraining walls facing the heating direction. In many real situations, however, the length of the workpiece along the longitudinal direction is finite and not much larger than its transverse
(y direction) size. Moreover, there would be no constraining walls at the end of the workpiece facing the moving direction. Although it is possible to analyze the thermal and residual stress by using a full three-dimensional finite element model for real situations, a long computation time is needed for this method. To calculate the thermal and residual stress in an unconstrained workpiece more effectively, the modified two-dimensional finite element model was adopted in this work [1, 5]. In this model, the sliced solution domain was considered for calculating the thermal stress as shown in Fig. 2.

The structure should be properly restrained to eliminate all possible modes of the rigid body motion; otherwise the stiffness matrix will not be positive definite. The boundary conditions used in the analysis allow the free expansion of the workpiece in the transverse direction, and two nodal points (y = 0 and W on z = 0) were locked in the z direction (see Fig. 6).

3. Simulation results and discussion

The incremental analysis of the elastoplastic stress was carried out by using the finite element mesh. The material parameters such as elastic modulus $E$, plastic modulus $P$, yield stress $Y$ and thermal expansion coefficient $z$ needed for the mechanical analysis were adopted from the data for AISI 1045 steel, where the values below 700 °C were determined by the tensile test and those above 700 °C were extrapolated from the given data [3]. The dimensions of the turbine blade mould for the simulation were $W = 60$ mm and $D = 28$ mm for the upper part and $W = 60$ mm and $D = 40$ mm for the lower part. In this study, the basic processing parameters chosen for the analysis were as follows: high frequency power $P = 20$ kW, frequency of the power source $f = 10$ kHz, traverse speed $v_n = 20$ mm s$^{-1}$ for the upper part and $v_n = 10$ mm s$^{-1}$ for the lower part, and the height of the working coil from the workpiece surface $h = 5$ mm.

For rapid heating, as is usually observed in high frequency induction heating, the austenitic transformation temperature varies according to the heating rate. In this study, the heating rate observed at the surface of the mould centre was about $4.5 \times 10^3$ °C s$^{-1}$, which approximates to the typical heating rate calculated for high frequency induction heating [5]. For this heating rate, the $A_1$ temperature could be approximated to about 825 °C and the $A_1$ temperature to about 950 °C [7].

In high frequency induction heating, the cooling rate is high enough for all the material that undergoes austenitic transformation to be transformed into martensite. In the simulation, the volume change in the austenitic and martensitic transformation was attributed to the thermal dilatation for which the equivalent linear thermal expansion coefficients $2.8 \times 10^{-3}$ and $8.0 \times 10^{-3}$ respectively were adopted [8]. To determine the progress of the martensitic transformation the following kinetic equation proposed by Koistinen and Marburger was used [9]:

$$m = 1 - \exp[-k(M_s - T)]$$

where $m$ is the volume of the martensite, $T$ the instantaneous temperature, $k$ a constant which was found to be equal to $0.011$ K$^{-1}$ for most steels and $M_s$ the martensite start temperature, for which 360 °C was used.

Transformation plasticity is a phenomenon in which a permanent strain occurs when a phase transformation takes place under an applied stress, even for stresses lower than the yield stress of the material. In the high frequency induction heat treatment, the workpiece...
undergoes the transformation when thermal stresses are imposed, so that the transformation plasticity needs to be considered to obtain accurate simulation results.

In this study, the effect of the transformation plasticity was considered by adding an additional strain in the constitutive equation. The total strain was modified as follows [10]:

$$
\varepsilon_{ij} = \varepsilon_{ij}^t + \varepsilon_{ij}^a + \varepsilon_{ij}^p
$$

where $\varepsilon_{ij}^p$ is the transformation plasticity strain.

The typical shape of the hardening zone caused by the high frequency induction heating is shown in Fig. 7, in which the isothermal line of the maximum temperature was drawn for 825 °C. The simulated shape of the hardened zone in the upper part shows a very uneven hardening depth, which varied from 0.6 to 2.0 mm. A relatively thin hardening zone was built up at the corner of the upper part when compared with the other part. This is probably because the distance between the coil and workpiece was approximated to be larger at the corner area than at the other surface zone. A relatively thick hardening zone was produced at the centre part, because the amount of heat dissipation is small in this area due to the symmetry. However, the simulated hardening zone shape of the lower part, for which a constant distance from the coil to the workpiece could be well maintained, shows a fairly uniform hardening depth of about 1.4 mm along the mould surface.

For the comparison of simulation results, an analytical method for predicting the hardened zone was considered. The penetration depth $\delta$ of the magnetic field in the workpiece could be obtained by using eqn. (10). The used physical properties of AISI 1045 steel are as follows [11]: permeability $\mu = 1.5 \times 10^{-3}$, resistivity $r = 1.111 \times 10^{-6}$ $\Omega$ m at 800 °C. Then the penetration depth of each part is 0.965 mm. Owing to the heat conduction in the workpiece during heating, the overall depth of penetration is larger than the penetration depth of the magnetic field. It is possible to calculate the additional penetration due to heat conduction from the following expression [12]:

$$
d_i = 0.2t^{1/2}
$$

where $d_i$ is the depth of penetration and $t$ the time. The total depth of penetration is $\delta + d_i$. By using the above equations, it can be determined that the additional penetration by heat conduction in the upper part is $d_i = 0.245$ mm and in the lower part $d_i = 0.346$ mm. Consequently the total average penetration depth of the upper part is 1.21 mm and of the lower part 1.311 mm.

The difference between the simulation and analytical results is about 15%-40%. This large difference is probably due to the fact that the efficiency of the high frequency induction heating was assumed to be 100%
in the calculations, the physical properties were presumed for high temperatures, and the analytical expressions give only a rough estimate for the penetration depth.

Figures 8–11 show the isolines of the longitudinal and transverse residual stress in the upper and lower part of the mould, with and without considering the transformation plasticity on the assumption that the $A_t$ temperature is 825 °C. Fairly uniform compressive longitudinal and transverse stresses were formed in the surface area of the hardened zone. The compressive residual stresses exist only in the region close to the surface within the hardened area, and then the stress is suddenly changed to tensile stress as the distance from the top surface increases. The severe variation of the residual stress distribution in the boundary of the hardened zone is due to the equilibrium of forces within the workpiece which undergoes a sudden temperature change and martensitic phase transformation. The compressive residual stress seems to originate from the martensitic transformation in the hardening zone of the workpiece. In the inner part of the upper and lower mould, the tensile residual stress was produced to satisfy the force equilibrium condition, while the position of the maximum tensile stress is located near the boundary of the hardened zone. The compressive residual stress in the surface zone of the hardened mould has a beneficial effect on mechanical characteristics such as wear resistance, fatigue strength etc., whereas the tensile residual stress in the interior of the mould possesses the potential to generate internal microcracks.

The transformation plasticity has a considerable influence on the distribution of the residual stress, reducing the level of the compressive residual stress in the hardened area (Figs. 9 and 11). This is because the elastic strain is changed to plastic strain when the plastic deformation occurs during the phase transformation in the temperature range between $M_t$ and $M_f$. In the non-hardened zone, however, the distributions of the residual stresses were very similar to each other. These figures show that the transverse residual stress in the hardened zone is smaller than the longitudinal residual stress regardless of whether the transformation plasticity is considered.

4. Conclusions

The transient heat flow and residual stress during surface hardening of the turbine blade mould were
the force equilibrium due to the severe temperature gradient. The maximum tensile stress occurred along the boundary of the hardened zone which has the potential to cause internal cracks.

The differences between the distribution of residual stresses considering the transformation plasticity and that obtained without considering it are large, especially in the hardened zone with the compressive residual stress.

Acknowledgment

This work was partially supported by the Korea Science and Engineering Foundation under Contract 89-02-01-06.

References