Numerical estimation of heat affected zone in spirally laser quenched shaft

T Domański1, M Saga2, M Kubiak1, Z Saternus1 and P Novak2

1 Częstochowa University of Technology, Institute of Mechanics and Machine Design Foundations, Dąbrowskiego 69, 42-201 Częstochowa, Poland
2 University of Žilina, Faculty of Mechanical Engineering, Univerzitná 8215/1, 010 26 Žilina, Slovak Republic

E-mail: domanski@imipkm.pcz.pl

Abstract. This study concerns numerical modelling and computer simulation of thermal phenomena in spiral laser heating of shaft made of S355 steel. 3D numerical analysis is developed for the prediction of heat affected zone in Abaqus/FEA software. Power distribution of spirally moving heat source is implemented into additional DFLUX subroutine, written in Fortran programming language. Changing with temperature thermophysical properties are assumed in the analysis. The model of heat source motion takes into account the size of heat affected zone. Results of numerical analysis include temperature field, predicted heat affected zone and cooling rates. These results are a start point for the analysis of thermomechanical phenomena with the consideration of phase transformations in solid state.

1. Introduction

The work carried out numerical analysis of thermal phenomena of the laser hardening process of a cylindrical object. The material of the heated cylinder is carbon steel with increased strength S355. Numerical tests were carried out in the Abaqus FEA calculation program. The program developed a discrete model of the analyzed heated zone. In the material module of the Abaqus/CAE program, thermal properties variable with temperature were defined. The implementation of additional numerical procedures enabled modeling of the laser beam movement around the perimeter of the object. Based on the numerical calculations carried out, the temperature distribution in the heated zone was determined. The shape of the remelting zone and heat affected zone was numerically estimated.

2. Mathematical model

The heat flow equation in the Abaqus program is made on the basis of the energy conservation equation and Fourier law, expressed in the criterion of weighted residuals method. The notation of the equation in the program is as follows [1, 2]:

\[ \int_V \rho \frac{\partial U}{\partial t} \delta T \, dV + \int_V \frac{\partial \delta T}{\partial x_\alpha} \left( \lambda \frac{\partial T}{\partial x_\alpha} \right) dV = \int_V \delta T \, q_v \, dV + \int_S \delta T \, q_s \, dS \]  \tag{1}

where \( \lambda \) is a thermal conductivity (W m\(^{-1}\)°C\(^{-1}\)), \( U = U(T) \) is an internal energy (J kg\(^{-1}\)), \( q_v \) is a laser beam heat source (W m\(^{-3}\)), \( T = T(x_{ao}, t) \) is a temperature (°C), \( q_s \) is a boundary heat flux (W m\(^{-2}\)).
$\delta T$ is a variational function, $\rho$ is a density (kg m$^{-3}$), $V$ is a volume (m$^3$), $S$ is a surface (m$^2$) $T = T(x, t)$ is temperature (°C).

Equation (1) is completed by the initial condition and boundary conditions of Dirichlet and Neumann type. While the heat exchange with the environment is performed by Newton's condition, which takes into account the loss of heat by convection, radiation and evaporation [3, 4].

Most often in the literature on the numerical modeling of the laser beam hardening process to describe the power distribution of the welding source, the Gaussian mathematical model of the volumetric heat source is adopted (equation (2)). This model takes into account the linear decrease of energy intensity along material penetration depth [5, 6]. The shape of heat source is assumed as the shape of a truncated cone (figure 1) [7].

$$q_v(r, z) = \frac{Q}{\pi r_o h} \exp\left[\left(1 - \frac{r^2}{r_o^2}\right)\left(1 - \frac{z}{h}\right)\right] \text{ where } r_o = r_i - \left(r_i - r_b\right) \cdot \frac{z}{h}$$

where $Q$ is a laser beam power (W), $h$ is the laser heat source penetration depth (m), $z$ is actual depth (m), $r_i$ is a beam radius for $z = 0$, $r_b$ is a beam radius for $z = h$ and $r$ is an actual radius (m), where $r = \sqrt{x^2 + y^2}$.

![Figure 1. Diagram of a moving heat source.](image)

In Abaqus FEA, the moveable heating source is simulated using an additional DFLUX numerical subroutine written in the Fortran programming language [1]. The subroutine takes into account the power distribution of the beam, its location, beam movement and motion direction. The location of the center of the source is determined for each time step depending on the adopted source speed.

3. Numerical modeling

The three-dimensional discrete model of the analyzed object was developed in the Abaqus FEA calculation program. The object under consideration is a shaft with a diameter of 60 mm made of S355 steel. The discrete model analyzed and its division into finite elements is shown in figure 3.

The work analyzes the impact of heat source movement parameters on the size of a heated zone. The program developed a suitable three-dimensional discrete model, according to the diagram in figures 2 and 3. Technological parameters of the heating process were taken from the literature [6] for numerical calculations. For the heating process the following was assumed: beam power $Q = 700$ W and source speed $v = 1$ m min$^{-1}$. 
Based on the numerical verification in the case of heating, the beam radius \( r = 0.9 \) mm, and the penetration depth \( h = 2 \) mm.

The location of the heating source on the outer surface of the shaft depends on the adopted heating speed and time. The analysis is carried out in polar coordinates transposed into the Cartesian system:

\[
\begin{align*}
x &= R_z \sin(\varphi_0 + \omega \cdot t) \\
y &= R_z \cos(\varphi_0 + \omega \cdot t) \\
z &= z_0 + v_z \cdot t
\end{align*}
\]

where \( R_z \) is outer radius of the shaft (m), \( t \) is time (s), \( \varphi_0 \) is the angle of the initial position of heat source on the outer shell, \( \omega = \frac{v_1 \cdot t}{t} \) is angular speed in which \( v_1 \) = const. is peripheral speed, \( z_0 \) is an initial position on the axis \( z \), \( v_z \) is an axial speed along \( z \) axis. Heating speed \( v \) is the result of the peripheral \( v_1 \) and axial \( v_z \) speed \( v = \sqrt{v_1^2 + v_z^2} \).

![Diagram of the analyzed object](image1)

**Figure 2.** Diagram of the analyzed object.

![Finite element mesh](image2)

**Figure 3.** Finite element mesh.
Figure 3 shows the numerical model with a finite element grid marked on it. The highest mesh density occurs at the place of operation of the heating source. To reduce the duration of the simulation at a further distance from the heating line, the mesh has a much larger dimension. Numerical calculations were performed using a separable method. First, calculations of thermal phenomena were made, followed by calculations of structural and mechanical phenomena.

Numerical analysis of thermal phenomena occurring in the hardening process is difficult to perform because it requires taking into account the specific conditions of the laser beam heating process.

The 355 steel material model was included in the Abaqus material module. The calculations took into account variables with temperature, thermomechanical properties of the adopted steel grade [5, 8, 9]. The figure 4 shows the temperature-physical thermophysical properties of 355 steel. For 355 steel solidus temperature \( T_S = 1400 \, ^\circ\text{C} \), liquidus \( T_L = 1455 \, ^\circ\text{C} \), latent heat of fusion \( H_L = 260 \cdot 10^3 \, (\text{J kg}^{-1}) \). The ambient temperature is \( T_0 = 20 \, ^\circ\text{C} \) and the heat transfer coefficient with the environment \( \alpha_k = 100 \, \text{W m}^{-2} \).

![Figure 4](image)

**Figure 4.** Thermophysical properties of S355 steel.

4. Results and discussion

Simulation calculations were performed on the basis of developed numerical models for the adopted process parameters. Temperature distribution was determined for the analyzed heating technique and the shape and size of remelting zones was determined. In figures 5 and 6, a solid line indicates the \( A_{c3} \) temperature exceeding limits.

Figure 5 shows the temperature distribution on the cylinder surface after \( t = 3 \, \text{s} \) when the heating source made one turn, after \( t = 60 \, \text{s} \) when the heating source made six turns, after \( t = 115 \, \text{s} \) when the heating source made eleven turns. The maximum temperature we observed on the surface was 1103 °C. This means that the austenitizing temperature has been exceeded.

On the other hand, figure 6 shows the temperature distribution in the views of the shaft and in the cross section of the centre of melting zone. Where the numerical estimated shape of the \( A_{c3} \) temperature overflow zone is marked with a solid line.

Figure 7 presents thermal cycles for 4 selected measuring points located in the middle of the length of the heated shaft. Selected points are located at different distances from the symmetry axis as shown in the figure. Based on the results presented, it can be concluded that the shaft surface has been heated to a depth of 2.1 mm (temperature above 1000 °C), which is the temperature that guarantees the transformation of the structure into austenite. The shaft core was also heated to around 180 °C, which does not change its structure.
Figure 5. Temperature distribution over the surface of the shaft at the time 3, 60, 115 s.

Figure 6. Obtained temperature field in the views of the shaft.

Figure 7. Longitudinal temperature distribution in points 1, 2, 3, 4.
5. Conclusions
During the analysis, the adopted parameters of the heating source simulating laser heating allow achieving appropriate temperature distributions enabling hardening of the shaft surface layer at a depth of 2 mm. Developed mathematical and numerical models in Abaqus software enable the analysis of thermal and mechanical phenomena occurring during laser hardening, give the possibility of numerical prediction of temperature field and the geometry of heating source. Numerical simulation of hardening process is Abaqus program requires implementation of additional numerical subroutines. Based on the developed model, it is possible to estimate the temperature distribution in spirally hardened shafts of any diameter, which allows forecasting the surface properties of hardened objects. The presented results are the basis for the development of the model that allows estimation of phase shares and mechanical properties of hardened objects.

6. References
[1] SIMULIA 2007 Abaqus FEA theory manual. Version 6.7, Dassault System
[2] Domanski T, Sapietova A and Saga M 2017 Procedia Engineering 177 64–69
[3] Cheng H, Xie J and Li J 2004 Computational Materials Science 29 453–458
[4] Piekarska W, Kubiak M, Saternus Z and Rek K 2013 Archives of Metallurgy and Materials 58 1237–1242
[5] Piekarska W and Saternus Z 2017 Procedia Engineering 177 196–203
[6] Arif A, Al-Omari A, Yilbas B and Al-Nassar Y 2011 J Mater Process Tech 211 675–687
[7] Heming Ch, Xieqing H and Honggang W 1999 Journal of Materials Processing Technology 55 339–343
[8] Pacheco P M, Savi M A and Camarao A F 2001 Journal of Strain Analysis 36 507–516
[9] Huiping L, Guoqun Z, Shanting N and Chuanzhen H 2007 Material Science and Engineering: A 452–453 705–714