УДК 621.01(06)

W. Piekarska, M. Kubiak, Z. Saternus
Czestochowa University of Technology, Poland
Tel. +48 (034) 325 06 99; E-mail: piekarska@imipkm.pcz.pl

ANALYSIS OF THERMAL PHENOMENA IN LASER WELDED CONSTRUCTION PARTS USING ABAQUS SOFTWARE

Computer modelling of welding using commercial computer software requires special numerical procedures to determine chosen thermal phenomena or the implementation of specific boundary conditions. Basic versions of commercial computer solvers used for the analysis of stress and strain in constructions usually dont take into account the full complexity of phenomena accompanying welding processes. Abaqus FEA allows to include special procedures for such an analysis. Modelling of movable heat source in DFLUX subroutine is presented in this study and the computer simulation of temperature field during laser welding of various construction joints, performed in Abaqus/Standard solver. On the basis of obtained results the shape and size of charasteristic zones of welded joints are estimated.
***Keywords**: laser welding, movable heat source, temperature field, numerical simulation.*

Introduction

In recent years there has been a significant growth in the use of a laser beam in different types of welded joints. Laser beam welding allows performing various joints at any position, with or without additional material. Specific properties of laser radiation gives a very rapid melting of the material which contributes to obtaining a very high welding speed with a small amount of molten material and a small zone of thermal influence [1-4]. Lasers are being increasingly used in many branches of industry, especially in automotive industry and ship building. Involved in the production laser welding allows for new and innovative connections of construction parts [2, 3] with high technological requirements taken into account for the quality and reliability of welded joints.

Study concerning the influence of particular welding parameters on weld geometry is necessary to understand and optimize the process [5-10]. Experimental determination of basic process parameters is expensive. Therefore, the mathematical and numerical models of physical phenomena in welding become a powerful tools supporting engineers. An important stage of thermal modeling is the determination of suitable heat source power distribution. The problem becomes even more complex in T-joints where welding heat source must take into account the direction of laser beam. [10].

Abaqus finite element analysis (FEA) is a commercial software based on finite element method used in numerical analysis of physical phenomena in wide range of industrial processes. Using Abaqus/Standard solver for linear and nonlinear engineering simulations or Abaqus/Explicit solver for brief transient dynamic events, engineering work groups are able to consider phenomena in the field of continuum mechanic, electricity, acoustic, etc. This commercial modelling tool leads the engineers through the entire computer aided design process, starting from the creation of analyzed object geometry and definition of simulation parameters (material definition, boundary conditions, loads, etc.) – preprocessing, to solution of created task (processing). Finally, ending at graphical presentation of simulation results (postprocessing).

Abaqus FEA used in simulations of welding processes requires an appropriate modelling of movable welding heat source, which must be implemented into DFLUX subroutine [11, 12]. Algorithm for creation of computational welding model is presented in Fig. 1.

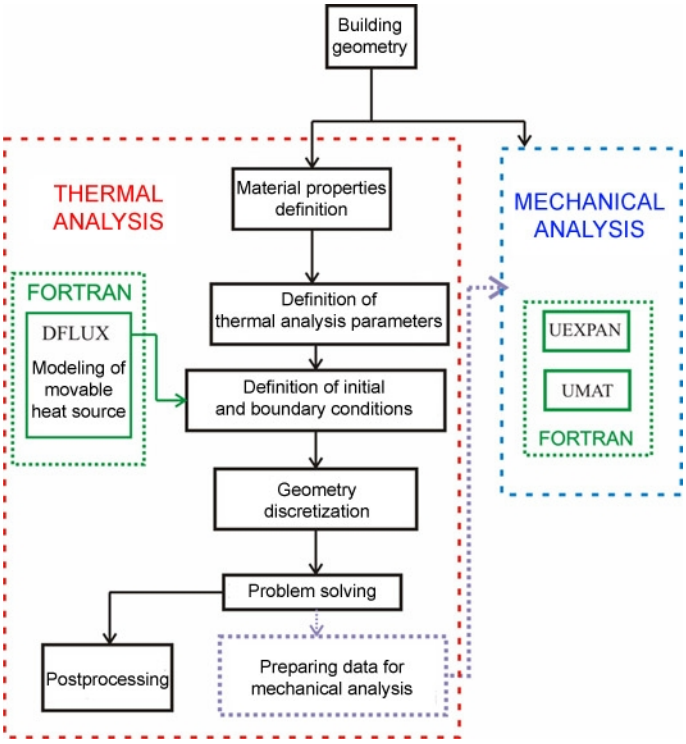

Fig. 1. Computational welding model in Abaqus FEA

This study contains mathematical and numerical models of thermal phenomena in laser beam welding process. Numerical analysis of temperature field was performed for butt-joints, lap-joints and T-joints. Computational model as well as numerical analysis was performed in Abaqus FEA. Thermo-physical parameters varying on temperature and latent heat of fusion were assumed in calculations. The distribution of movable heat source was implemented into DFLUX subroutine. On the basis of obtained temperature field, the size and shape of fusion zone and heat affected zone is estimated.

Mathematical models and numerical algorithms

In modelling of laser beam welding process, the intaraction of laser energy on the material is usually considered as a volumetric heat source in the shape of a cylinder, cone or truncated cone [6, 7, 9, 13] with gaussian distribution in radial direction. From analysis of this welding process it is observed that power decreases with increasing depth of penetration. In this study linear decrease of laser energy along material penetration depth is assumed. Volumetric heat source model is defined as follows [6]:

$$q_v(r,z) = \frac{Q_L}{\pi r_o^2 d} \exp\left[\left(1 - \frac{r^2}{r_o^2}\right)\left(1 - \frac{z}{d}\right)\right], \qquad (1)$$

where Q_L is laser beam power [W], d is a penetration depth [m], z is current depth [m], r_o is beam radius [m] and $r = \sqrt{x^2 + y^2}$ is current radius [m].

Simulation of laser beam welding process performed in Abaqus FEA requires the implementation of subroutines written in FORTRAN programming language. For the implementation of movable heat source, a DFLUX subroutine is used where power distribution is modeled as well as the motion of the heat source along welding direction (x-direction) using constant welding speed *v* [m/s] and simulation time t [s] passing into the subroutine from Abaqus solver.

Additionally heat source system is transformed by an angle α (Fig. 2) in T-joint computer simulation. In Fig. 3 heat source power distribution (1) is presented at the top surface and in cross-section of welded joint before and after transformation.

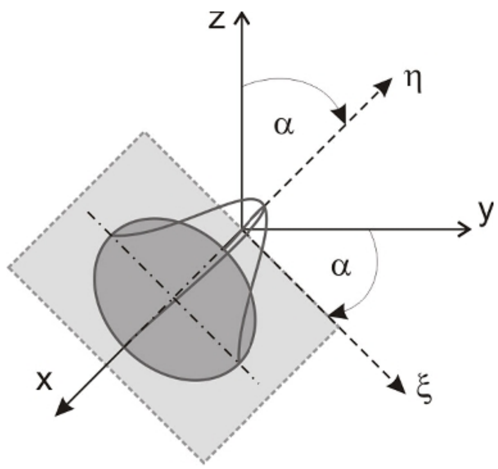

Fig. 2. Transformation of heat source power distribution

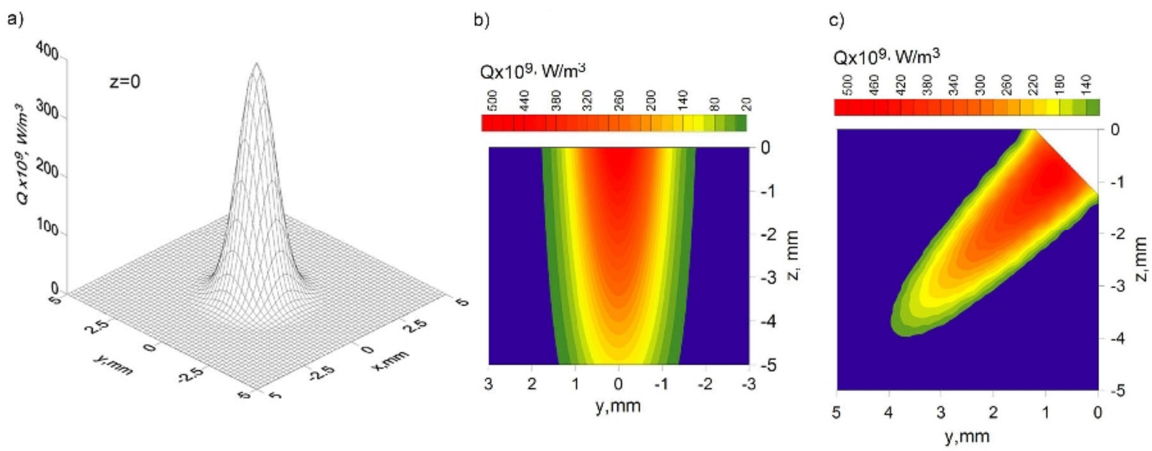

Fig. 3. Heat source power distribution a) at the top surface and in cross section of welded sheets b) in cartesian and c) rotated coordinate system

Analysis of temperature field is performed using „Uncoupled heat transfer" [11] in Abaqus/Standard solver. The model is based on the energy conservation equation and Fourier's law [12] taking into account thermo-physical parameters, such as thermal conductivity, density and specific heat dependent on temperature. Temperature field in variational formulation is expressed as follows:

$$\int_V \rho \dot{U} \delta T \, dV + \int_V \frac{\partial \delta T}{\partial x_\alpha} \cdot \left(\lambda \frac{\partial T}{\partial x_\alpha} \right) dV = \int_V \delta T \, q_V \, dV + \int_S \delta T \, q_S \, dS \,, \tag{2}$$

where U is a internal energy, q_S is a heat flux toward element surface [W/m^2], δT is the variational (weight) function.

Equation (2) is completed by initial condition $t = 0 : T = T_0$ and boundary conditions of Neumann and Newton type, with heat loss due to convection and radiation taken into account, according to the following equation [14]:

$$q_S = -\lambda \frac{\partial T}{\partial n} = \alpha_k (T|_\Gamma - T_0) + \varepsilon\sigma(T|_\Gamma{}^4 - T_0^4) - q(r,0) \,, \tag{3}$$

where α_k is a convective coefficient, assumed as α_k=50 W/m^{2}K, ε is radiation coefficient (ε=0.5), and σ is Stefan-Boltzmann constant and $q(r,0)$ is the heat flux towards the top surface of welded workpiece ($z=0$) in the source activity zone.

The body is approximated geometrically with finite elements. Temperature for every finite element is described by shape functions $\varphi_j(x_\alpha)$:

$$T^e(x_\alpha, t) = [\varphi_1 ... \varphi_n] \begin{bmatrix} T_1 \\ T_n \end{bmatrix} = \varphi_j T_j \,. \tag{4}$$

Variational field is also interpolated by shape functions (The Galerkin approach):

$$\delta T^e(x_\alpha) = \begin{bmatrix} \delta T_1 \\ \delta T_n \end{bmatrix} = \delta T_i = \varphi_i \,. \tag{5}$$

Time integration of internal energy material derivative is performed using the backward difference algorithm [12].

In equation (2) internal energy U takes into account the latent heat of fusion (H_L), thus assuming linear approximation of a solid fraction (f_S) between solidus (T_S=1750K) and liquidus (T_S=1800K) temperatures, specific heat $c(T) = dU / dT$ (Fig. 4) is defined as:

$$c(T) = \begin{cases} c_S & \text{for} \quad T < T_S \\ \dfrac{c_S + c_L}{2} + \dfrac{H_L}{(T_L - T_S)} & \text{for} \quad T_S \le T \le T_L \,, \\ c_L & \text{for} \quad T > T_L \end{cases} \tag{6}$$

where subscripts $_S$ and $_L$ stay for solid and liquid phase respectively (assumed as c_S=650 and c_L=840 J/kgK), H_L is a latent heat of fusion which was set to 270×10^3 J/kg.

Density is calculated from the ratio of density in solid and liquid state: $\rho = \rho_S f_S + \rho_L (1 - f_S)$, where density of solid and liquid state was set respectively to ρ_S=7800 kg/m^3 and ρ_L=6800 kg/m^3.

Thermal conductivity varying with temperature is assumed in analysis. In the solid state $\lambda=\lambda(T)$ is defined according to data form the literature [15]. Much higher value of $\lambda(T)$

is assumed in high temperatures (Fig. 4), which corresponds to the motion of liquid material in the welding pool [14].

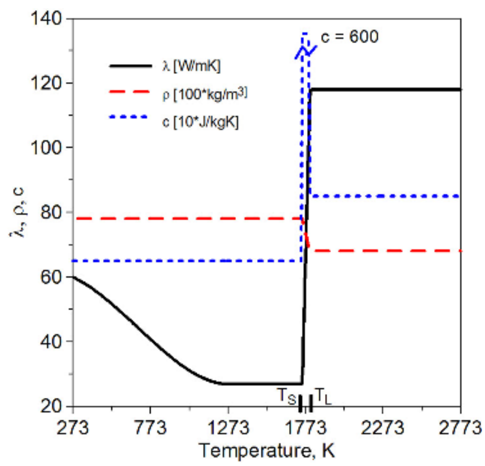

Fig. 4. Thermal conductivity, density and specific heat assumed in calculations

Results of calculations

Numerical analysis was performed for butt-joint, lap-joint and T-joints made of steel welded by the laser beam without additional material. Welding parameters assumed in calculations are described in table 1.

Table 1.Welding parameters assumed in calculations

Parameters	**butt-joint**	**lap-joint**	**T-joint** ($\alpha = 45^o$)
Welding speed	v=0.7 m/min	v=0.6 m/min	v=0.8 m/min
Laser beam power	Q_L=3.8 kW	Q_L =2.2 kW	Q_L =1.0 kW
Beam (spot) radius	r_0=1 mm	r_0=1 mm	r_0=0.4 mm
Penetration depth	s=7 mm	s=4 mm	s=1.4 mm

Simulation of laser welded butt-joint

Computer simulation was made for sheets with dimensions L=250mm, a=30mm and g=5mm (Fig. 5a). Created finite element mesh, consist of 16200 cubic elements (Fig. 5b), is dense in the range of heat sources activity zone with linear increase of spatial step in x-y plane. In order to reduce computational time, symmetry of the joint was used assuming only a half of the mesh with thermal isolation constraint in the plane of symmetry.

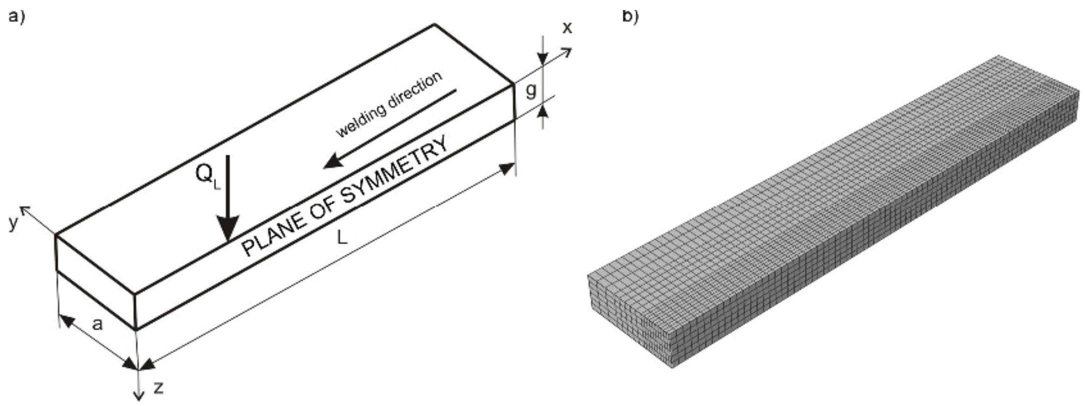

Fig. 5. Laser beam welding of butt-joints: a) scheme of considered system, b) finite element mesh

Fig. 6 presents temperature profile and temperature distribution in the cross-section of the joint. Liquidus isotherm (T_L) determines the shape and size of welding pool, whereas temperature T_g≈1000K determines heat affected zone boundary.

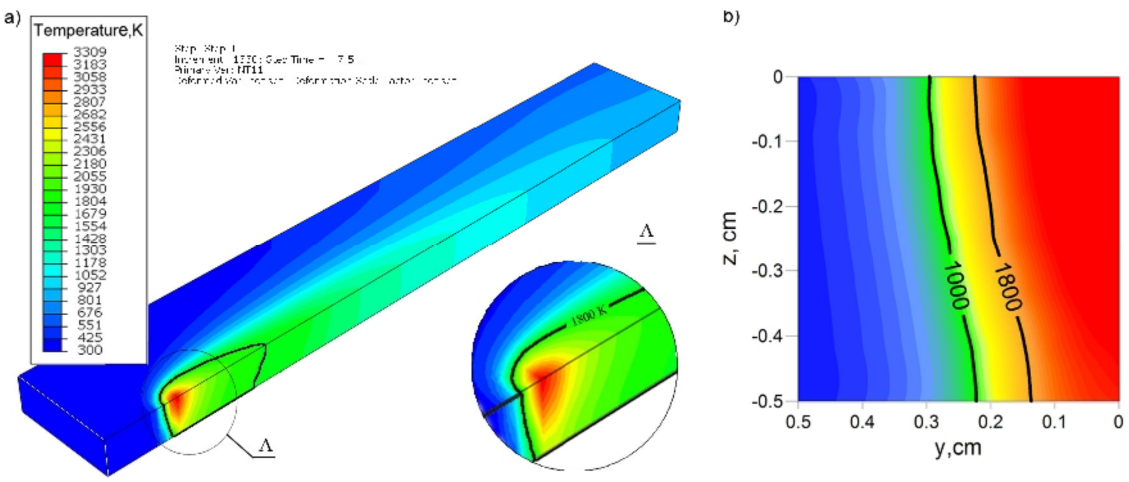

Fig. 6. Temperature field in laser beam butt-welded joint: a) entire profile, b) in the cross section of the joint

Simulation of laser welded lap-joint

Numerical analysis of temperature field in laser welded lap-joint was performed for plates with dimensions L=50mm, h=20mm and g=1.5mm. It was assumed that plates overlap in length L_1=25mm with a perfect contact of joined surfaces (Fig. 7a). Joined sheets were discretized using 22 187 cubic finite elements (Fig. 7b) with increasing spatial step in direction perpendicular to the welding line (starting from 0.2 mm in the range of heat sources activity zone).

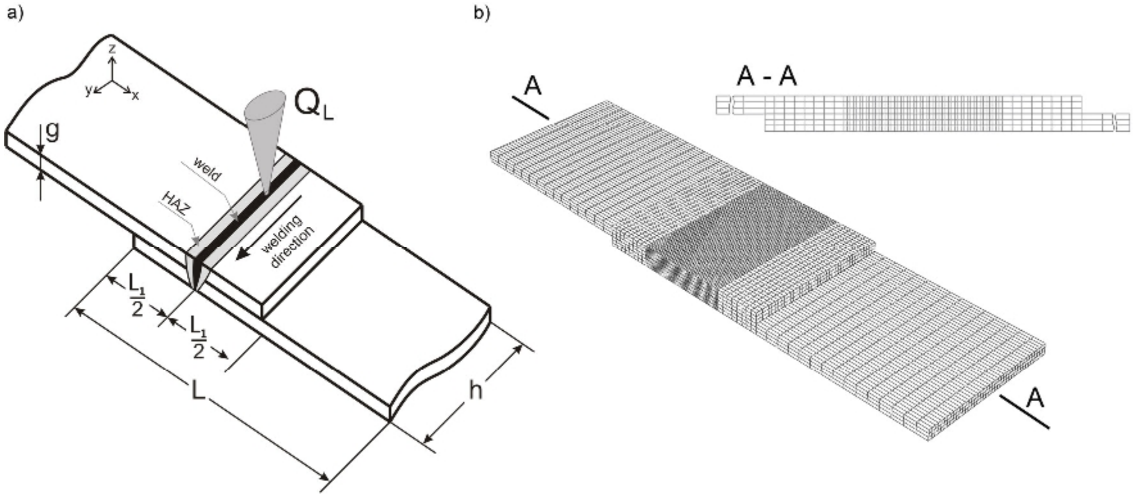

Fig. 7. Laser beam welding of lap-joints: a) scheme of considered system, b) finite element mesh

Temperature distribution in the lap-joint is presented at the top surface (Fig 8a), in the longitudinal section (Fig. 8b) and at the cross section (Fig. 9) in the heat source activity zone (x=0).

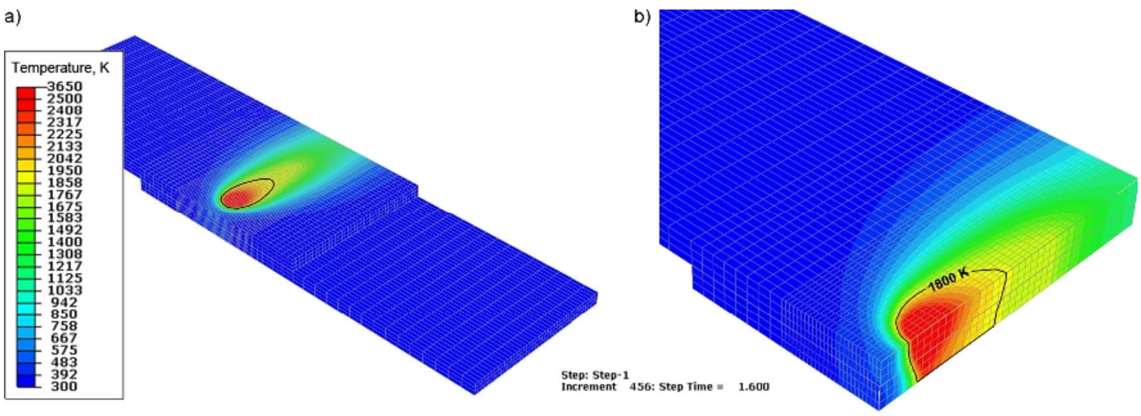

Fig. 8. Temperature field a) from the face of the weld and b) in longitudinal section of laser welded lap-joint

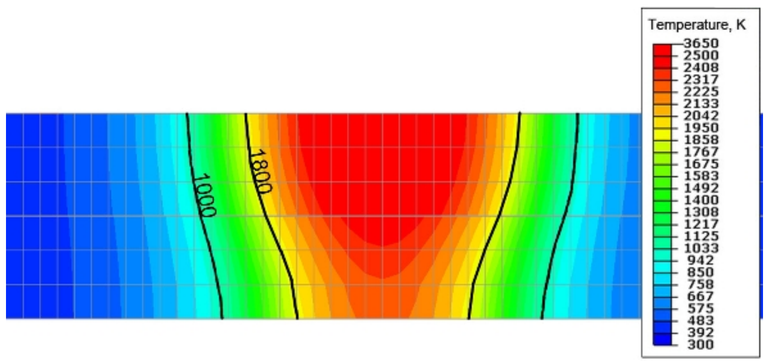

Fig. 9. Temperature field in the cross section of the weld

Simulation of laser welded T-joint

Schematic sketch of laser welded T-joint and finite element mesh consist of 80 000 cubic element is presented in Fig. 10. Dimensions of plates was set to *L*=60mm, *h*=20mm, *b*=20mm and *g*=1.6mm.

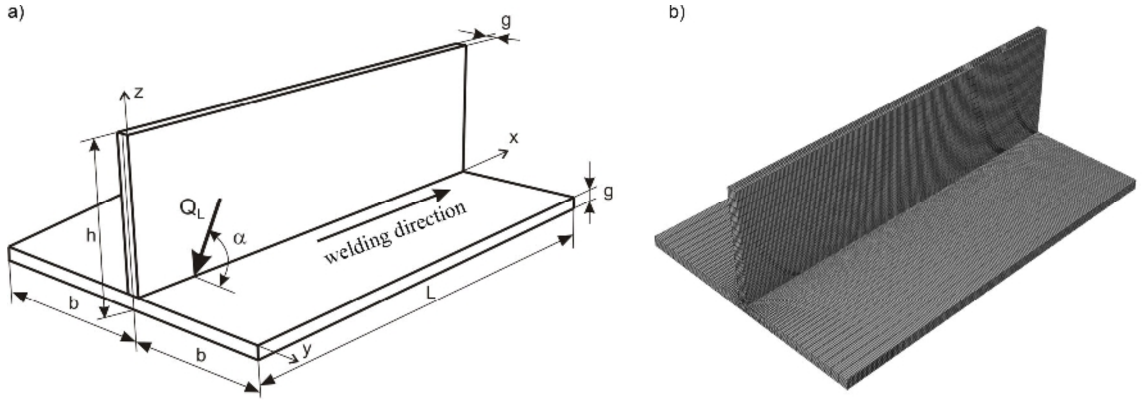

Fig. 10. Laser beam welding of T-joints, a) scheme of considered system and b) finite element mesh

Profile of temperature in welded T-joint and temperature field in the cross section of the weld, in the centre of heat source activity zone is presented in Fig. 11, with marked fusion zone boundary (liquidus isotherm) and heat affected zone boundary.

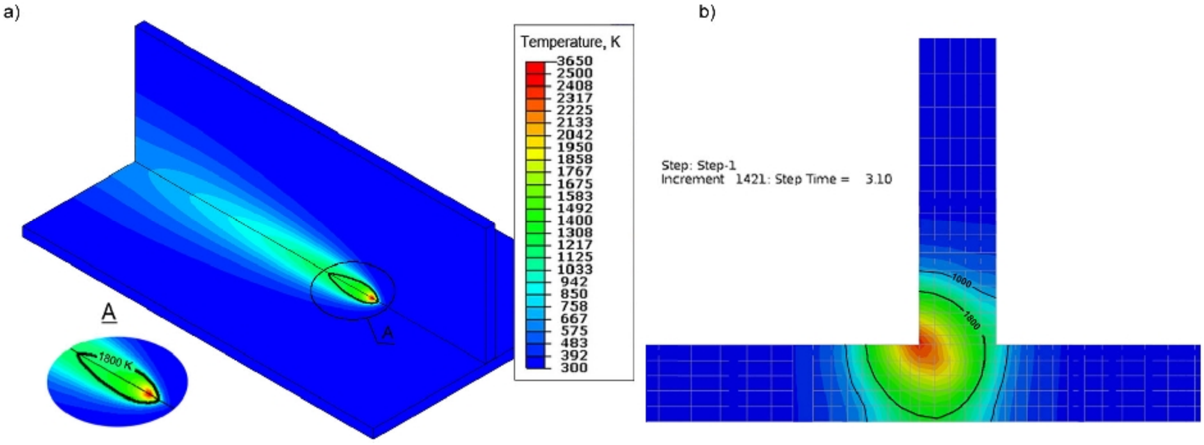

Fig. 11. Temperature distribution in T-joint: a) temperature profile and b) in the cross section of the joint

Conclusions

This study presents three dimensional computational model of thermal phenomena accompanying laser beam welding of construction parts. Numerical analysis, performed in Abaqus FEA commercial solver, required implementations of additional procedure DFLUX written in FORTRAN programming language. The subroutine allowed the modeling of movable heat source with the assumption of appropriate heat source power distribution.

Computational analysis was performed for laser welded butt-joints, lap-joints and T-joints. On the basis of obtained temperature field the geometry of fusion zone and heat affected zone in welded joint was estimated.

Presented models allows for the optimization of laser beam welding process in construction parts design, having as input data laser welding process parameters and dimensions of joined sheets.

Numerically estimated temperature field can be the basis for determining the deformation of joined elements, as well as to analyze the kinetics of phase transformation in solid state and the prediction of microstructure composition, which can be helpful for engineers in prediction of mechanical properties of laser welded construction elements.

References:

1. Deng D. Determination of welding deformation in fillet-welded joint by means of numerical simulation and comparison with experimental measurements / Deng D., Liang W., Murakawa H. // Elsevier: Journal of Materials Processing Technology. – 2007. – Vol. 183, – 219-225 p.
2. Fatigue strength in laser welding of the lap joint / Choa S.K., Yanga Y.S. Sona K.J. et al. // Elsevier: Finite Elements in Analysis and Design. – 2004. – Vol. 40. – 1059-1070 p.
3. Laser beam welding and formability of tailored blanks / Baysore J.K., Williamson M.S., Adonyi Y. et al. // American Welding Society: Welding Journal. – 1995. – Vol. 74. – 345-352 p.
4. Technological applications of laser beam welding and cutting at the Instytut Spawalnictwa / Pilarczyk J., Banasik M., Dworak J. et al. – Warszawa: Przeglad Spawalnictwa, 2006. – Vol. 5-6. – 6-10 p.
5. Makhnenko V.I. Role of mathematical modelling in solving problems of welding dissimilar steels (Review) / Makhnenko V.I., Saprykina G.Y. // E.O. Paton Institute: Paton Welding Journal. – 2002. – Vol. 3. – 14-25 p.
6. Numerical simulation of the laser welding process in butt-joint specimens / Tsirkas S.A., Papanikos P., Kermanidis Th. // Elsevier: Journal of Materials Processing Technology. – 2003. – Vol. 134. – 59-69 p.
7. Chang W.S. A study on the prediction of the laser weld shape with varying heat source equations and the thermal distortion of a small structure in micro-joining / Chang W.S., Na S.J. // Elsevier: Journal of Materials Processing Technology. – 2002. – Vol. 120. – 208-214 p.
8. Prediction of welding distortion in butt join of thin plates / Long H., Gery D., Carlier A. et al. – Elsevier: Materials & Design, 2009. – Vol. 30. – 4126-4135 p.
9. Gery D. Effects of welding speed, energy input and heat source distribution on temperature variations in butt joint welding / Gery D., Long H., Maropoulos P. // Elsevier: Journal of Materials Processing Technology. – 2005. – Vol. 167. – 393-401 p.
10. Transient finite element simulation of the temperature and bead profiles of T-joint laser welds / Shanmugam N., Buvanashekaran G., Sankaranarayanasamy K. et al. – Elsevier: Materials and Design. – 2010. – Vol. 31. – 4525-4542 p.
11. Abaqus analysis user's manual. Version 6.7. – Dassault System: Abaqus Documentation, 2007.
12. Abaqus theory manual. Version 6.7. – Dassault System: Abaqus Documentation, 2007.

13. Bokota A. Modeling of residual stresses in laser welding / Bokota A., Piekarska W. // E.O. Paton Institute: Paton Welding Journal. – 2008. – Vol. 6. – 19-24 p.

14. Piekarska W. Numerical analysis of thermo-mechanical phenomena during laser welding process. The temperature fields, phase transformations and stresses. – Czestochowa University of Technology: Monograph / W. Piekarska. – 2007. – N. 135. – 157 p.

15. Serajzadeh S.A., Karimi T.A Study on austenite decomposition during continuous cooling of a low carbon steel / Serajzadeh S.A., Karimi T.A // Elsevier: Materials and Design. – 2004. – Vol. 25. – 673-679 p.

Надійшла до редколегії 20.02.2012.

В. Пекарска, М. Кубяк, З. Сатернус

ИСПОЛЬЗОВАНИЕ ПРОГРАММЫ ABAQUS ДЛЯ АНАЛИЗА ТЕПЛОВЫХ ЯВЛЕНИЙ ЛАЗЕРНОЙ СВАРКИ ЭЛЕМЕНТОВ КОНСТРУКЦИИ

Компьютерный процесс сварки с использованием коммерческих компьютерных программ требует специальных численных процедур для определения некоторых тепловых явлений, или введение конкретных граничных условий. Коммерческие программы для анализа напряжений и деформаций в структуре, в своей основной версии не учитывают всей сложности явлений процесса сварки. Включение таких специальных процедур к общей основной программе позволяет программа Abaqus FEA. В работе представлено моделирование движущегося источника тепла, используя процедуру DFLUX, и с помощью программы Abaqus FEA приведено численное моделирование распределения температуры при лазерной сварке в различных конструктивных комбинациях. Рассчитаны формы и размеры различных зон сварных соединений.

Ключевые слова: *лазерная сварка, движущийся источник тепла, температурное поле, численное моделирование.*

В. Піекарска, М. Кубіак, З. Сатернус

ВИКОРИСТАННЯ ПРОГРАМИ ABAQUS ДЛЯ АНАЛІЗУ ТЕПЛОВИХ ЯВИЩ ЛАЗЕРНОГО ЗВАРЮВАННЯ ЕЛЕМЕНТІВ КОНСТРУКЦІЇ

Комп'ютерний процес зварювання з використанням комерційних комп'ютерових програм вимагає спеціальних чисельних процедур для визначення деяких теплових явищ, або введення конкретних граничних умов. Комерційні програми для аналізу напружень і деформацій у структурі, у своїй основній версії не враховують всієї складності явищ процесу зварювання. Включення таких спеціальних процедур до загальної основної програми дозволяє програма Abaqus FEA. У роботі представлено моделювання рухомого джерела тепла, використовуючи процедуру DFLUX, і за допомогою програми Abaqus FEA наведено чисельне моделювання розподілу температури при лазерній зварці в різних конструктивних комбінаціях. Розраховані форми і розміри різних зон зварювальних з'єднань.

Ключові слова: *лазерне зварювання, рухоме джерело тепла, температурне поле, чисельне моделювання.*