TWO-FLUID MODEL OF MELTING DYNAMICS OF METALS IN COLD CRUCIBLE

V. Frishfelds[(1)], A. Jakovics[(1)], and B. Nacke[(2)]

[(1)]LMMETP laboratory, University of Latvia,
Zellu 8, LV-1002 Riga, Latvia

[(2)]Institute for Electrothermal Processes, University of Hanover
Wilhelm Busch Str. 4, D-30167 Hannover, Germany

ABSTRACT. Numerical modelling of melting of metals like TiAl in cold crucible is considered. Combined calculations of harmonic magnetic field, thermal transfer and flow dynamics have been made in 2D axis-symmetrical approximation using the effective value of magnetic permeability for split crucible. Distribution of magnetic field is performed acconting the massive coils of the inductor. Because the top boundary is varying due to large electromagnetic compression of metalllic load, two-fluid model has been applied. There the first fluid is the metallic load, while the second fluid is assumed as much lighter and neutral incompressible fluid, where no body forces are present. Dynamics of both fluids is calculated by means of direct solution of both momentum and continuity equations, which however needs large computational time. Because of intense turbulant flow of melt, turbulent viscosity has beeen used.

INTRODUCTION

High frequency (~10 kHz) skull melting in induction furnace with cold (slitted and water cooled) crucible is well suited for treatment of alloys with high melting point such as TiAl. The principial scheme of the furnace is shown in Figure 1. Thin skull layer is formed between the melt and cooled crucible. The power requirements depend significantly on the properties of this layer and layer of solid material at the water-cooled bottom. Such cold crucible furnaces have several advantages, e.g., high purity of final material. The properties of the processed material are well

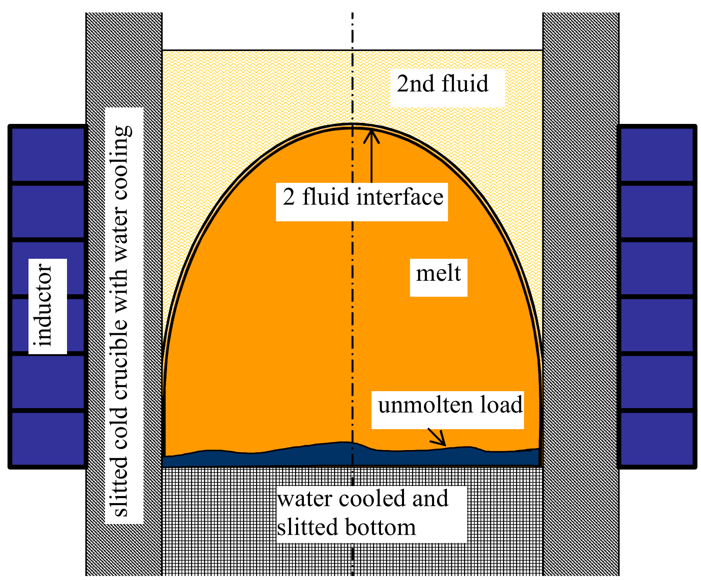


Figure 1. The principal scheme of the cold crucible. The length of the cold crucible is chosen as infinite. The coils of the inductor are put close separated by very thin insulating layer.

TELEPHONE +371-7033780

FAX +371-7033781

suited for novel applications, e.g., in car industry.

The conductivity of metals is high (~10^5-10^7 1/(Ω·m)). Therefore, the melt is pressed towards the centre of the furnace by electromagnetic force of the inductor [1] that reduces heat transfer losses to the inductor as the contact area becomes smaller. The dominating force in metal melt flow is electromagnetic rather than thermo-gravitational despite of small skin depth. Usually, the velocities are quite high reaching around 0.3 – 0.5 m/s and flow regime is highly turbulent. Therefore, model of turbulent viscosity has to be used. One of the simplifications in comparison with oxides [1] is that change of porosity can be neglected. Essential parts of the numerical model have been compared with package ANSYS as well as with other simulations. Further comparisons with real experimental data [3], [6] are planned in the future.

ELECTROMAGNETIC FIELD WITH SEVERAL MASSIVE COILS

Axial-symmetric model of simulations has been made for melting in inductor crucible. Azimuthally split crucible is modelled by use of effective relative magnetic permeability lower than one as suggested in [3] in order to escape from 3-dimensional description. The distribution of vector potential A in axial symmetric coordinates can be find form the equation

$$\frac{\partial}{\partial r}\left(\frac{1}{\mu_r r}\frac{\partial(rA)}{\partial r}\right)+\frac{\partial}{\partial z}\left(\frac{1}{\mu_r}\frac{\partial A}{\partial z}\right)=\mu_0\sigma(T)\left(i\omega A+\frac{1}{r}\frac{\partial U}{\partial \varphi}\right), \tag{1}$$

where U is scalar potential, the azimutal gradient of which differs from zero only in the coils, μ_r – is relative magnetic permeability, which is equal to 1 except the slitted crucible, where it is by an order of magnitude smaller. The bottom of the cruccible also consists of slitted water colled metal, but here the relative magnetic permeability is close to 1.

The height of crucible is chosen infinite in order to simplify the geometry, as its characteristic length exceeds the height of load significantly. Because the melt boundary in two-fluid model can be spread a little, the approximation of small skin depth may not work. Moreover, little spreading of the metal boundary can increase slightly the total amount of produced Joule heat.

If the load distribution changes rapidly so that voltage distribution is changing too, it is better to add equations that require equal current in each coil directly to the system of equations. In order to keep the complete electromagnetic matrix symmetric, it is necessary to set the current rather than voltage. The resulting matrix in direct method is solved by Gaussian method for sparse matrixes as in [1].

Direct method is still quite time consuming describing the temporal changes of load distribution. Therefore, it is better to use some kind of iteration method. As the resulting matrix described in previous sub-section is symmetric the conjugate gradient method is preferred. However, the calculations showed that added equations that require equal current in each coil could lead to slow divergence of the solution process in some cases. The reason appeared to be that added equation to the electromagnetic matrix makes the complete matrix non-positive, i.e., some of the eigenvalues are negative. This explains the instability of conjugate gradient method for the total system of equations. Therefore, the equations that require equal current in each coil are omitted, and approximated voltages in each coil are used. The voltages are obtained in first step from the direct method and further they are adjusted to make the currents more close to the required ones. The new voltages after each conjugate gradient step are approximated in following way in each coil j:

$$U_j = U'_j + \theta(I - I'_j)\frac{U'_j}{I'_j}, \tag{2}$$

where θ is some coefficient. This coefficient is chosen to be ~0.1 for better convergence. The parameters with " ′ " are the obtained values in current step and without it – the new approximated values. The current deviations from the given one in this method do not differ more than by 1 %. To be even more sure about the accuracy of the electromagnetic calculations, the accurate direct method is used if the deviation of current from the given one in each coil falls behind 3 %.

HEAT TRANSFER

Usual heat transfer equation with convection is used to describe the transient heat transfer in axially symmetric coordinates:

$$\rho c_p\left(\frac{\partial T}{\partial t} + v_r\frac{\partial T}{\partial r} + v_z\frac{\partial T}{\partial z}\right) = \frac{1}{r}\frac{\partial}{\partial r}\left(r\lambda(T)\frac{\partial T}{\partial r}\right) + \frac{\partial}{\partial z}\left(\lambda(T)\frac{\partial T}{\partial z}\right) + \frac{\sigma(T)|A|^2\omega^2}{2}. \tag{3}$$

The heat transfer to the inductor and bottom is dominating due to high thermal conductivities of considered metals, while radiation losses usually give smaller contribution to total energy balance. Nevertheless, radiation heat transfer is included too, because of the significant importance of the effects occuring at two-fluid interface. Despite the two-fluid interface can be slightly spread, the boundary for radiation is well estimated by appropriate choice of separating threshold. The curvature of surface in axial-symmetric coordinates is approximated by that best fitting curve (e.g., parabola) of top surface using least square method. Because there are no significant temperature differences between the melt and second fluid, heat transfer by conduction in stationary case gives only a small part in total heat balance. The description of phase change is of significant importance in the melting process, which makes the numerical calculations more laborious. Enthalpy function is used to account the change of the phase. The detailed time dependent description of phase change is also a major improvement in comparison with different models (e.g. [4], [5]) of metal melting in cold crucible.

TiAl melt is not in direct contact with the sectioned metallic cooler, but there is a thin insulating skull layer with much lower electrical and thermal conductivities. If the temperature of the metallic cooler is constant T_0 and thickness of skull layer δ is small with respect to the size of load, we can set the boundary condition *bnd* of the third kind at the surface with skull:

$$\left.\left(\frac{\lambda\delta}{\lambda_{sc}}\frac{\partial T}{\partial x} + T\right)\right|_{bnd} = T_0, \tag{4}$$

where λ_{sc} – heat conductivity of the skull. The similar skull layer forms also at the bottom of the load. Because the thickness of the skull layer δ is in millimeter scale or smaller, we have to know only the heat transfer coefficient λ_{sc} / δ .

TWO INCOMPRESSIBLE FLUID MODEL

In order to use previously developed model for ZrO_2 melting [1], we want to escape from detailed description of dynamics of top boundary. The detailed description could create quite irregular shape of the top surface of TiAl in intermediate phases. For this reason, we will assume that system consists of two incompressible fluids:

1) TiAl,
2) other fluid (like paraffin or air), with lower density, viscosity, heat conductivity, heat capacity, and negligible electrical conductivity and thermal expansion. This fluid has no melting and boiling points and mentioned quantities are not temperature dependent. The fluid is transparent and radiation from the separating surface freely goes through.

Thus, the considered area of axial cross-section can be rectangular. The meshing is made homogeneous for better convergence in fluid dynamics. Because of negligible electrical conductivity and thermal expansion, there are no body forces in pure fluid of the second fluid. The characteristic boundary between two fluids with approximately equal pressure should shape automatically. If the kinematic viscosity and density of the second fluid is considerably different from the first one, numerical difficulties occur. Let us denote the local volume fraction of the second fluid as $\Pi=\Pi(\mathbf{r},t)$. This will be called further as void fraction. The values of density, dynamic viscosity, heat capacity, thermal and electrical conductivities can be approximated by following equation of linear interpolation:

$$\mathrm{X} = \mathrm{X}_1(1-\Pi) + \mathrm{X}_2\Pi \,, \tag{5}$$

where the indexes 1 – first fluid (metals like TiAl), 2 – second fluid (gas); X denote density ρ, heat capcity c, thermal conductivity λ, electrical conductivity σ or logarithm of dynamic viscosity $\ln\eta$. Let us denote the combined local velocity of both fluids as $\mathbf{v}$. Then we have continuity equation

$$\nabla\mathbf{v} = 0 \tag{6}$$

and momentum equations

$$\frac{\partial\mathbf{v}}{\partial t} + (\mathbf{v}\nabla)\mathbf{v} = \frac{1}{\rho}\nabla(\eta\nabla\mathbf{v}) + \frac{\mathbf{f}-\nabla P}{\rho}\,. \tag{7}$$

Considering the thermal expansion, we cannot use just buoyancy force approach, because both the fluids have different density. Instead the complete gravitational force $\rho\mathbf{g}$ must be added to the force, and the pressure corresponds to the total one. This leads to further difficulties for choosing the right boundary conditions for the pressure in SIMPLER method. Therefore, simultaneous solution of momentum equations and continuity equation are performed, where the pressure boundary conditions are not required. Transient calculations do not require additional iterations as in SIMPLER method except at high velocities when non-linear term in momentum equations is dominating. Hence, direct solution scheme of complete matrix made by both velocity components and pressure is solved by methods for sparse matrixes. Significant spatial variations in viscosity in the system make the total matrix close to singular. There are several possibilities how to escape from this singularity by adding an additional term in continuity equation, e.g. laplacian of pressure or artificial compressibility [2]. We have chosen such kind of continuity equation with added lapplacian of pressure

$$\nabla \vec{v} = \varepsilon \nabla \left(\frac{1}{\rho} \nabla p \right), \tag{8}$$

where ε is small parameter. Its value should be chosen in such a way that singularity is absent and continuity condition of the flow is still nearly satisfied.

The void fraction in the system changes due to two factors: motion of two-component mixture as a whole and filtration of one component into another. The first factor is described by change of void fraction with following equation

$$\frac{\partial \Pi}{\partial t} + (\gamma \vec{v}) \nabla \Pi = 0, \tag{9}$$

where $\gamma \leq 1$ is added to make the fluid interchange slower. It will not change stationary distribution since the gradient of void fraction in direction of velocity should cancel in this case. Filtration of one component into another is caused by the force, which acts only to the metal component but not the second. Therefore, the interchange of both fluids between neighbouring cells i-1 and i has following proportionality

$$\sim \frac{1}{\eta} f_{i-1,i}. \tag{10}$$

It is hard to obtain a proper coefficient before this rate of filtration, and it chosen so that boundary between two fluids is not too much spread, while keeping the flow of two-component mixture as dominant. The last factor is important because it gives the correct stationary distribution.

In order to be closer to actual situation, where the second fluid is neglectable, its properties are set in such a way that density, thermal conductivity, heat capacity, dynamic viscosity are by order of magnitude lower, while kinematic viscosity is of the same order of magnitude.

The stability of the surface is higher, if surface tension is included. The surface tension slightly influences the interface position [3]. If the fluids are well separated the height of the load depends on the radial argument $\varphi(r)$, and the local curvature used for pressure difference is following:

$$\frac{1}{R_1} + \frac{1}{R_2} = \frac{1}{\sqrt{1+\varphi'^2}} \left(\frac{\varphi''}{1+\varphi'^2} + \frac{\varphi'}{r} \right), \tag{11}$$

which gives the required pressure difference between two fluids. Different method must be used in two-fluid model. Here the additional term

$$-\alpha \sigma_{srf} \nabla \Pi \tag{12}$$

is added to the force, where α is geometrical factor. It tends to direct the metal flow towards its higher concentration.

RESULTS

Results will be shown using the data taken from the reference [3]. The value of turbulent viscosity is set 100 times higher as the value of physical viscosity and the relative magnetic permeability of the cooled sectioned layer is set to 0.1. Let us consider the example of TiAl melt, where the electromagnetic skin depth is higher than in Al melt. The radius of load is 0.078 m and height of load is 0.11 m (7.88 kg) in an exmaple of simulation in Figure 3.

Figure 2 shows the characteristic velocity pattern in the initial stages. The places where the velocity is shown corresponds to the molten part of the load, because the velocities inside the second fluid are not drawn. Thus, the velocity distribution also shows the charateristic development of two fluid interface and gradual melting of the load. The iniatial surface of the load is assumed to be flat as can be seen from Figure 2 – left.

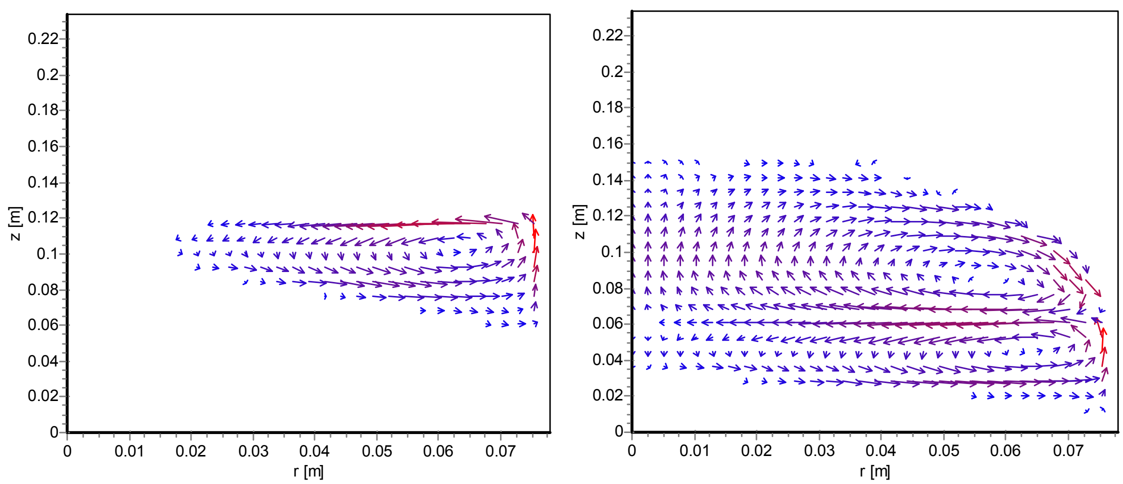


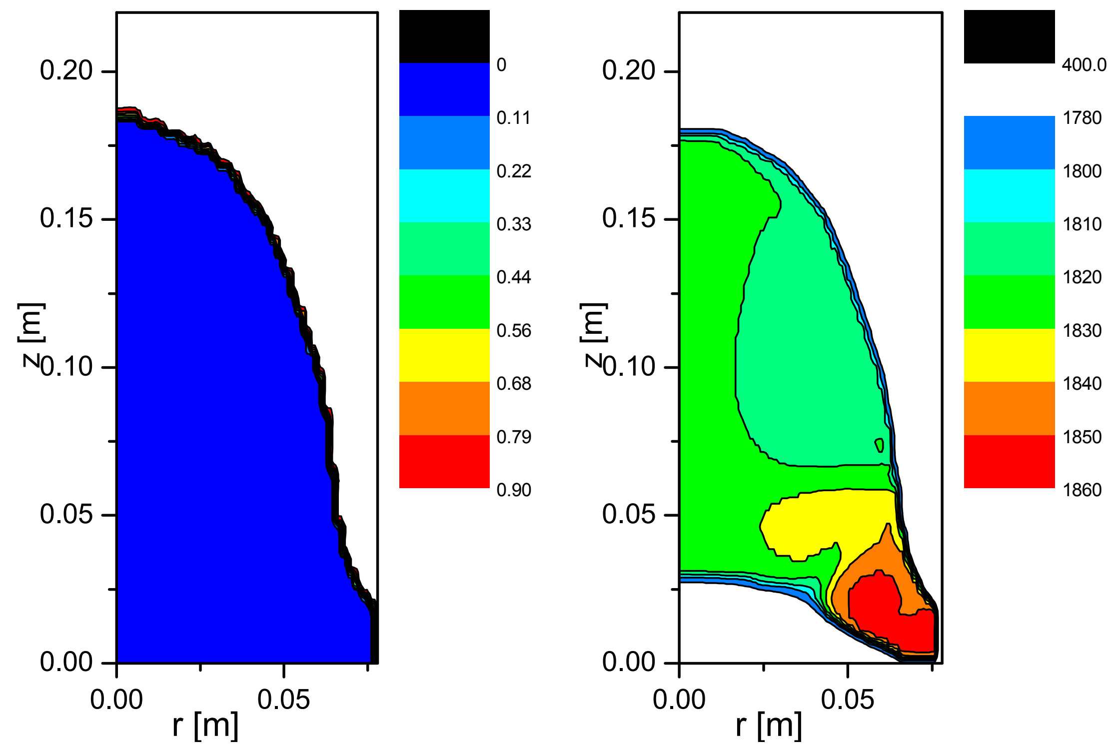
Figure 2. The characteristic flow patterns at the starting stages of inductive heating. The velocity is shown only in the melt, not in the second (neutral) fluid.

The inductance of the system is good parameter characterising how much the melt is pressed towards the centre as shown in Figure 3d. The fluctuations of the maximal velocity is caused by the fact that melt can flow along the boundary between two fluids. This causes the dissipation of the two-fluid boundary in the finite difference or element methods, which leads to considerable local stresses. These stresses can produce considerable local velocities and subsequent fluctuations in integral parameters. In order to suppress these fluctuations one can use smaller spatial step in numerical methods.

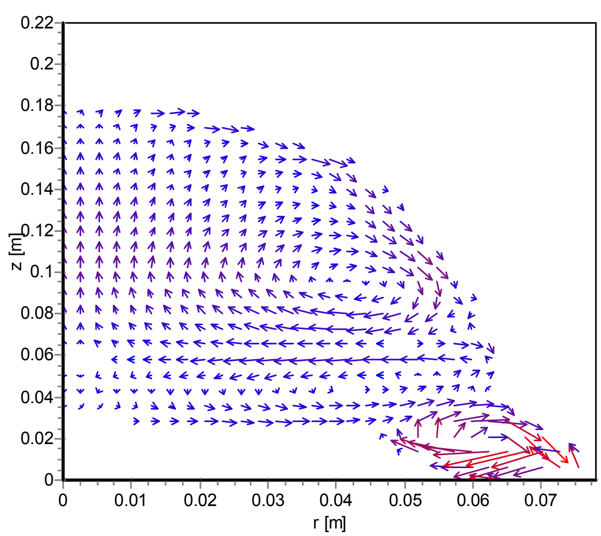
Figure 3 shows that with thermal resistance layer (skull), both at bottom and sidewalls, the temperature is almost homogeneous in the melt. The flow pattern is relatively stable and is mostly driven by electromagnetic rather than thermo-gravitational force due to small variations of temperature. The heat transfer resistance is set to 0.0007 (m^2K) /W of bottom and side walls but its difficult to measure its actual value while the skull layer is extremely thin.

As can be seen from Figure 3b, the maximum of temperature in final distribution is placed either near the inductor, where maximal Joule heat liberated. But in some cases it can be placed at the top of melt, where hot upward flux is directed. Below the position with temperature maximum is quite cold, where cold flux along condesed interface is flowing. Figure 3 shows that bottom of the load remains condensed at amplitude of current 5100 A.

Figure 4 shows characteristic feature that maximal velocity is initially increasing, but it becomes lower when the melt is forced back from the sidewalls, where the electromagnetic force becomes lower.



a) b)



c)

Figure 3. Stationary field distributions (after 10 min) at inductive heating with 5100 A (absolute value) of 7.88 kg of TiAl melt: a) void fraction, b) temperature distribution in melted part of TiAl (melting point is 1773 K), c) velocity distribution.. Maximal velocity on the axis is around 0.5 m/s.

CONCLUSION

Calculations showed that two-fluid model can be applied sucessfully to account for the motion of top boundary of the load due to electromagnetic compression. If the density of the second fluid is by order lower than of the first fluid, the calculations of fluid dynamics are still stable.

The slight presence of Archimedian force can be accounted by setting the density of load little higher. The boundary of two fluids is found to be sufficiently sharp and both fluids are well separated. One of the main problems is the tangential motion of two-fluid front along the grid. Due to the grid slight mixing of both fluids is present that creates large local stresses and, consequently, accounts for flow instabilities. These instabilities are prevented numerically. The thin skull layer and radiation is found to be very important for the overall heat balance.

Further calculations of the melting

process are required with finer grid that is quite important for the correct two-fluid interface and melts with smaller electromagnetic skin depth.

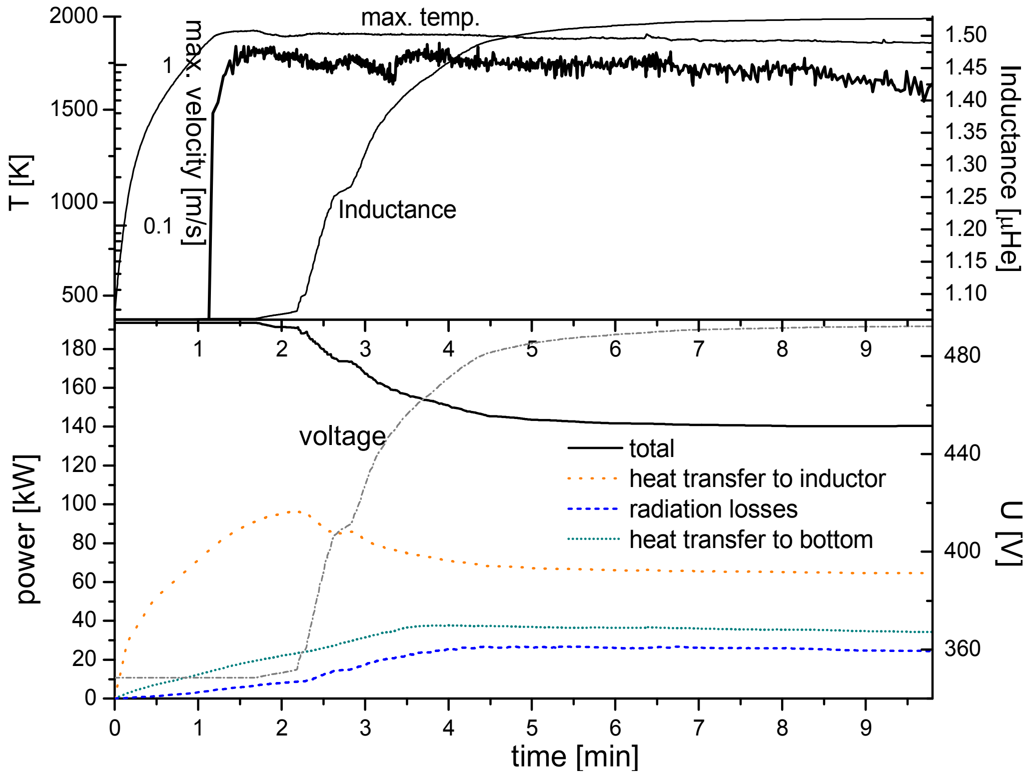


Figure 4. The change of power parameters, voltage (absolute), maximal temperature, maximal velocity and inductance by inductive heating with 5100 A of 7.88 kg of TiAl melt.

REFERENCES

[1] Nacke, B., Frishfelds, V., Jakovics, A. (2002). Modelling of inductive melting of oxides in inductor crucible furnace. *Elektro-Wärme International*, No. 3, 105-109.

[2] Fletcher, C.A.J. (1991). *Computational techniques for fluid dynamics*. Vol. II, Springer-Verlag, Berlin.

[3] Vogt, M. (2001). *Einsatz des Kaltwand-Induktions-Tiegelofens zum Schmelzen und Gießen von TiAl-Legierungen*. VDI Verlag, Reihe 19, Nr. 132, p. 160.

[4] Baake, E., Nacke, B., Umbrashko, A., Jakovics, A. (2002). LES Modelling of Heat and Mass Transfer in Turbulent Recirculated Flows, Proceedings of the *5th International PAMIR Conference*, Ramatuelle, France, Vol. I, 173-178.

[5] Bojarevics, V., Djambazov, G., Harding, R.A., Pericleous, K., Wickins, M. (2002). Investigation of the Cold Crucible Melting Process, Proceedings of the *5th International PAMIR Conference*, Ramatuelle, France, Vol. II, 77-82.

[6] Umbrashko, A., Baake, E., Nacke, B., Jakovics, A. (2004). Experimental investigations and numerical modeling of the melting process in the cold crucible, Proceedings of the *4th International HES Symposium*, Padua, Italy.