LES of Heat and Mass Exchange in Induction Channel Furnaces

Abstract. This paper introduces LES model verification for heat and mass transport in Induction Channel Furnace (ICF) and its comparison with two-equation turbulence models. LES simulation results show very good accordance with experimental data including both flow structure and temperature distribution in the melt. Transit flow velocity in the channel is also estimated.

Streszczenie. Praca przedstawia weryfikację modelu LES do transportu masy i ciepła w piecu indukcyjnym kanałowym i jego porównanie z dwuwspółczesnymi modelami turbulencji. Wyniki symulacji LES wskazują na bardzo dobrą zbieżność z danymi doświadczalnymi zarówno pod względem struktury przepływu jak i rozkładu temperatury w kapieli. Prędkość przepływu w kanale jest także oszacowana.

Keywords: channel induction furnace, induction heating, heat and mass transport.

Introduction

Induction channel furnaces (ICF) are often used not only for copper and other non-ferrous alloy production but also for cast steel melting. This equipment is very effective but till now it has several important limitations concerning melt overheating in the channel and slag formation in different parts of the furnace.

ICF is build like transformer with the iron core where the melt channel acts as a secondary coil (Fig. 1). The iron core has large magnetic permeability and therefore reduces magnetic flux leakage increasing electrical efficiency. Primary coil or inductor is usually orthogonal to the symmetry plane of the channel. The channel is surrounded by the thermal insulating ceramics with outer metallic shield forming one block, which can be easily replaced when necessary. This part of the furnace represents heating element and is called induction channel unit, because main ohmic power losses in the melt occur here. Melt bath is connected to the channel through the neck and can have any shape. Usually the bath is cylindrical with rounded bottom surface, which reduces surface area to minimize thermal losses.

Power range of the industrial channel furnaces starts at about 300 kW and extends to 1200 kW and above for each induction heating unit. Maximal channel furnace power is limited by maximal melt overheating, which leads to technological problems (higher thermal losses, infiltration into ceramics and possible leakage). This overheating can be reduced by highly intensive turbulent heat exchange and by transit flow of the melt through the channel, which intensifies heat exchange in the bath-channel system. The transit melt flow is affected by buoyancy and by distribution of Lorentz forces, which can be influenced by the channel design. The basic aim of this LES research is to determine proper overheating of the melt and create some preconditions for channel shape optimization for stronger transit flow in industrial ICF.

Literature review

Very detailed theoretical and experimental investigation of the physical processes in ICF was performed in [1, 2], where theoretical estimations of transit flow velocities in the adiabatic channel with the one and two metal loops were performed. Adiabatic boundary conditions for the channel means that \( \frac{\partial^2 T}{\partial i^2} = 0 \) (or \( T = T_{in}/L^2 \)), where \( i \) is a coordinate along the channel for one induction unit. The temperature maximum \( T_{in} \) is shifted to the channel side due to some initial fluctuation by the transit flow and temperature changes along the channel assumed to be are linear. With these considerations the next expression is provided in [1] for the channel transit velocity estimation:

\[
v_0 = \frac{1}{2}\sqrt{\frac{d g \beta L f^2}{2 \rho c_p \lambda \sigma}}.
\]

where \( g \) is the gravity acceleration, \( c_p \) is the heat capacity, \( L \) is the channel length, \( f \) is the channel diameter, \( \lambda \) is the thermal conductivity and \( \beta \) is the thermal expansion coefficient. For the overheating estimation the next expressions are proposed:

\[
T_M = \frac{1}{2}\sqrt{\frac{2 \lambda L f^4}{\rho c_p L \sigma}} = \frac{\sqrt{\frac{1}{2}}}{\sqrt{\rho c_p L \sigma}}.
\]

Experimental study of horizontal channel furnace models [2] showed presence of many transversal vortices in the channel, which should also present in the vertical channel because they are created by EM force. The authors stated that temperature distribution is determined mainly by the turbulent mixing in the horizontal channel while thermal conductivity effects and therefore Prandtl number can be excluded. The transit flow existence was not found in the horizontal model for several symmetrical channel furnaces (one loop) with Wood’s melt, however it was proposed that concentric (to inductor) channel with straight exits is the most efficient channel design for optimal heat transport to the bath.

The further development of the one-dimensional channel model described bath as a part of the channel to close hydrodynamic equation integrals and exclude pressure [3]. This model had power dependent constants, which were determined from experimentally measured maximal temperature and its location. Model equations were also solved with the finite-difference method and temperature difference and transit flow velocity have been approximated with expressions \( \Delta T \sim P^\varepsilon_1 \) and \( v_0 \sim P^\varepsilon_2 \), where experimentally and theoretically determined coefficients \( \varepsilon_1 \) and \( \varepsilon_2 \) are in the range 0.57–0.78 and 0.27–0.34 correspondingly, which are in accordance with equations (1) and (2). Several important conclusions were also made: the transit flow in the channel is a result of action of the EM force together with the buoyancy; the turbulent oscillations are very important for heat and mass transport between the channel and the bath; the stability of the transit flow becomes better when the ICF power increases.
An interesting simplified theoretical model of induction furnace is presented in [4], where EM field equations are exactly solved for circular cross-section torus channel and circular cross-section iron core. A primary flow-field in the channel's circular cross-section was studied using the finite-difference method for Navier-Stokes equations with constant viscosity and without thermal dependence. The transit flow was later estimated introducing some perturbations into the primary field.

Fig. 1. Dimensions in mm of the melt region in experimental ICF with local coordinate system for EM and LES simulation.

Industrial channel furnaces were studied experimentally using similarity theory in [5], where a fourth-tenth scale physical model of 1300 kW inductor unit without thermal convection was described. The authors made experimental setup with a mercury and water-cooled design (mercury temperature was homogeneous within 1°C) of two-loop induction unit with a small bath. It was found that flow pattern in the bath and in the channel is not stable and it is difficult to find an averaged flow pattern. Transit flow through the channel legs was found to change its direction and magnitude several times during a test of 1 minute duration and possibly here is an analogy for low-frequency vortices oscillations in coreless induction furnaces [6].

Fig. 2. Experimentally measured velocity distribution in the bath y=0 for 62 kW electrical power.

All above mentioned authors conclude that 3D EM field and correct heat and mass transfer simulations in ICF are extremely difficult due to complicated geometry and 3D nature of inner physical processes. The most recent development in 3D simulation of channel induction furnace was made utilizing coupled EM and thermal and steady HD calculations for twin-loop ICF with aluminum [7].

Table 1. The Physical properties of the Wood’s melt

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Melting temperature ( T_{melt} )</td>
<td>72°C</td>
</tr>
<tr>
<td>Density at ( T_{melt} ) ( \rho )</td>
<td>9400 kg/m³</td>
</tr>
<tr>
<td>Dynamic viscosity ( \eta )</td>
<td>4.2·10⁻³ kg/m·s</td>
</tr>
<tr>
<td>Electrical conductivity ( \sigma )</td>
<td>1·10⁶ S/m</td>
</tr>
<tr>
<td>Thermal expansion coefficient ( \beta )</td>
<td>1·10⁻⁴ 1/K</td>
</tr>
<tr>
<td>Heat capacity ( c_v )</td>
<td>168 J/kg K</td>
</tr>
<tr>
<td>Thermal conductivity ( k )</td>
<td>14.05 W/m·K</td>
</tr>
</tbody>
</table>

Furnace geometry was presented by one quarter part and two-equation turbulence models were used. This simulation has shown that all selected turbulence models provide similar results for flow velocity and temperature fields without presence of transit flow in the channels. Flow intensities in the bath are much smaller than in the channels and temperature gradients in the liquid melt are about 12 K.

EM model

Due to small magnetic Reynolds number \( R_{mag} = \mu_0 \mu \sigma L V \ll 1 \) electromagnetic field can be decoupled from the hydrodynamic problem. Hence, smaller geometry model containing only the channel and part of the bath entrance can be used in EM calculations.

EM calculations were performed in “ANSYS Multiphysics” using half symmetry of the model geometry. The model had several regions defined (Fig. 3), where material electrical properties are important:

- channel itself and a bath with the Wood’s metal;
- copper cooling jacket between the inductor and the channel;

Fig. 3. Sketch of EM ANSYS model.

The presented numerical simulation is based on geometry (Fig. 1) and problem formulation from experiments and calculations of large scale ICF [4, 8, 9, 10]. The furnace was filled with the Wood’s melt (Table 1). The channel was made from refractory ceramics but the bath walls were made from stainless steel with built-in water cooling for better control of thermal regime. Flow velocities were measured using permanent magnet probes [11] with 18 Hz sampling rate and with maximal spatial resolution 0.01 m (Fig. 2). The measured maximal velocities in the bath were in the range 20÷40 cm/s and 50÷90 cm/s in the channel depending on the channel cross-section shape and on the furnace power, which had range 27÷74 kW. The experimentally measured average velocity data will be used for LES model verification.
• copper inductor;
• iron core for EM field amplification;
• surrounding air with infinite boundary.

Inductor had 32 (16 for half model) turns with rectangular cross-sections. Current distribution was calculated with respect to skin effect. The magnetic field is homogenized by presence of the laminated core, in which electrical power losses were not taken into account. The model was verified using total electrical power on two grids (coarse and fine) for \( I_{\text{amp}} = 1200 \text{ A (50 Hz)} \), which gives approximately 62 kW total power. Calculated total electrical power and power losses in the melt (Joule heat) were almost equal for two different grids.

Simulation results show very symmetrical Lorentz force (Fig. 4, 5) distribution in the channel. The maximal values of Lorentz forces are noticed on the inner bottom surface of the channel, which is the closest to the inductor. Lorentz force direction is collinear to the radii direction on a symmetry plane and force intensity in the bath is much lower than in the channel. EM force distribution in the channel cross-section should lead to two-vortex structure inside the channel. It also should ensure upward flow directly above the channel in the symmetry plane (Fig. 4).

**HD Calculations**

Calculations were performed in "ANSYS CFX" on a large model, which had full 3D geometry of the channel and the bath. The model mesh had above 3 million elements with refined boundary layers on all no-sleep walls. A constant temperature 80°C was kept on the bath walls during the experiments to ensure the same conditions for all measurements and the temperature is fixed on the bath walls for simulation. Free surface had heat transfer coefficient 20 W/m·K. Channel was stated to be adiabatic.

Material properties also included temperature dependent density \( \rho = \rho_0 (1 - \alpha(T - T_0)) \) [12] because thermal convection seems to be very important for heat and mass transport from the channel to the bath. Gravity vector was directed to negative \( z \). Appropriate high resolution numerical schemes were selected for precise calculations.

Calculations were also performed with \( k-\varepsilon \) or SST \( k-\omega \) two-equation turbulence models for the half geometry 3D model with respect to a symmetry plane \( y = 0 \).

\( k-\varepsilon \) model calculated velocity distribution in the symmetry plane \( y = 0 \) (Fig. 7 top) satisfies EM force distribution in the melt. Flow is directed radially outwards in the channel in this cross-section and there is an upward flow just above the channel forming two vortex loops in the bath with smaller flow velocity comparing to the channel. These loops have complex 3D structure and are closed at the side walls of the bath. There is also less intensive upward flow near the left and right sides of the bath. Different intensity of this side flow can be explained by the buoyancy.

**Fig. 4. Distribution of Lorentz force density in the melt, \( y = 0 \).**

**Fig. 5. EM force density distribution in radial channel cross-section.**

**Fig. 6. SST \( k-\omega \) calculated velocities in radial channel cross-section.**

**Fig. 7. \( k-\varepsilon \) and SST calculated velocity distributions on a vertical symmetry plane \( y = 0 \).**

SST model (Fig. 7 bottom) produces similar results for flow structure in the bath, but there is completely different flow pattern in the channel, where flow is represented by complex 3D vortices, which looks like Benard cells. There are about 8 such structures with characteristic sizes 1.2÷1.5 channel diameters along the channel length. Such vortical structures are not seen in \( k-\varepsilon \) model because according to
the standard \( k-\varepsilon \) model without enhanced wall functions overpredicts near-wall shear and therefore the flow is additionally stabilized. The flow intensity calculated by SST model is about 30\% higher especially in the bath, where the upward flow in the central part above the channel is more expressed.

Velocity distribution in the channel is represented by the two-vortex structure (Fig. 6) due to strong symmetry condition, which reduces flow degree of freedom. Characteristic velocity magnitudes in the channel are approximately the same for both models (\( \approx 0.5 \) m/s in maximum). The distance between the vortex centers is about one channel radius.

Temperature distribution in the melt is completely coupled with the flow, because the melt density is temperature dependent. Very strong turbulent heat exchange is expectable in the furnace while thermal conductivity effects should be smaller.

Melt temperature in the bath is very close to the wall temperature, which can be explained by the reduced heat exchange between the turbulent vortices in a standard \( k-\varepsilon \) model. Temperature distribution in the channel is not symmetric, with respect to plane \( y = 0 \), having local maximum at one channel side. Temperature maximum localization in the channel and temperature difference between the channel and the bath depend on the selected turbulence model (table 2).

SST model produces averaged temperature pattern with smaller temperature gradient near the channel exit to the bath comparing to \( k-\varepsilon \) model because 3D vortical structures are more developed in SST, which intensify vertical heat transport to the bath.

Full 3D calculations were performed with SST and LES turbulence models. All boundary conditions and material properties were the same. 3D SST solution repeats the previously described flow structure and temperature distribution for the half geometry simulations.

Temperature distribution in the channel has maximum on the right side with overheat about 45 K in full SST. Temperature maximum location has switched side comparing to the half geometry SST results. It seems that the maximal temperature position can not be determined exactly for symmetrical channel furnaces. It means that the direction of the transit flow velocity is determined by random fluctuations or disturbances in the flow itself neither on mesh and geometry factors.

LES was started from zero velocity field and 80\(^\circ\)C initial melt temperature. About 77 s of the flow were calculated. Velocity and temperature averaging was started from \( t = 10 \) s. Averaged velocity vectors and temperature contours were compared with \( k-\varepsilon \), SST model results and with the experimental data. Both LES and two equation models produce averaged velocity field, which structure is very close to experimental. LES calculated maximal velocity magnitudes in the channel are about 60÷70 cm/s. LES averaged characteristic flow velocities in the bath (Fig. 9) are about 30 cm/s and are of the same value like in experiment (Fig. 2).

Channel exits are characterized by the presence of two characteristic recirculating vortices. Such structures in the turbulent flow usually have strong low frequency interaction between, which intensify heat and mass transfer between the characteristic averaged vortices [6] and therefore spread temperatures near the channel exit, which is in accordance with LES model and experimental results. However, large temperature gradients near the channel exit are noticed in \( k-\varepsilon \) model results (Fig. 11), which also over predicts channel-bath overheat.

LES is also capable to establish averaged \( \approx2 \) cm/s transit flow along the channel, which is in accordance with
convection. The temperature maximum in the channel is some initial flow fluctuation together with thermal theoretical predictions [1] (Table 2). This flow is created by random fluctuation. Direction of this shift and of the transit flow can be different depending on initial random fluctuation. The temperature maximum in the channel is shifted from the symmetry point closer to one of the channel exits by this transit flow (Fig. 10). Direction of this shift and of the transit flow can be different depending on initial random fluctuation.

**Figure 10. LES averaged temperature distribution on a vertical symmetry plane y = 0.**

The temperature in the channel grows almost linearly from one of channel exits till the maximum (Fig. 11) and LES calculated temperature in the channel for Wood’s melt is about 34 K, which is very close to experimentally measured value [8]. The similar overheat is also calculated with half 3D SST model.

**Figure 11. Calculated temperature distributions along the channel.**

Conclusions

LES calculated velocity field pattern and temperature distribution along the channel are very close to experimentally determined, which allows usage of the model for industrial applications. Temperature maximum is shifted to the side from the symmetry position in the channel by the transit flow, which calculated intensity is about 2±5 cm/s in the studied case (Table 2).

SST turbulence model can be recommended for fast engineering analysis, where more realistic averaged temperature distributions are needed.

Proposed simulation approach allows to make very extensive study of the industrial channel furnaces where transit flow velocity need to be as high as possible for intensification of the heat transport to the melt bath, which should be achieved mainly by geometry optimization.

<table>
<thead>
<tr>
<th>Model</th>
<th>$V_0$, cm/s</th>
<th>$V_0$ theor, cm/s</th>
<th>$\Delta T$, K</th>
<th>$T_m$, K</th>
<th>$T_m$ location</th>
</tr>
</thead>
<tbody>
<tr>
<td>SST half</td>
<td>-5.4</td>
<td>1.42</td>
<td>37.7</td>
<td>393.4</td>
<td>left</td>
</tr>
<tr>
<td>k-ε half</td>
<td>5.2</td>
<td>1.36</td>
<td>53</td>
<td>408.8</td>
<td>right</td>
</tr>
<tr>
<td>SST full</td>
<td>0.3</td>
<td>1.4</td>
<td>45</td>
<td>398.6</td>
<td>right</td>
</tr>
<tr>
<td>LES t=77s</td>
<td>-1.8</td>
<td>1.4</td>
<td>36.6</td>
<td>397.2</td>
<td>left</td>
</tr>
<tr>
<td>LES averaged</td>
<td>-2.2</td>
<td>1.42</td>
<td>35.5</td>
<td>393.2</td>
<td>left</td>
</tr>
<tr>
<td>Experiment [8]</td>
<td></td>
<td></td>
<td>33.5</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Channel erosion and growing of the oxide formations are other problems, which can be investigated with the proposed method. Further results will include LES of several industrial channel furnaces and mass transport analysis on Lagrangian particle transport basis.

REFERENCES


**Authors:** Maksims Kirpo, University of Latvia, Faculty of Physics and Mathematics, Laboratory for mathematical modelling of environmental and technological processes, Zellu Str. 8, Riga, LV1002, E-mail: Maksims.Kirpo@lu.lv; prof. dr ing. Bernard Nacke, Leibniz Universität Hannover, Institut für Elektroprozesstechnik, Wilhelm-Busch-Str. 4, 30167, E-mail: ewh@ewh.uni-hannover.de.

**Table 2. The calculated parameters of the transit flow**