ADVANCED 3D MODELLING FOR ANODE BAKING FURNACES

Dagoberto S. Severo1, Vanderlei Gusberti1, Elton C. V. Pinto1.

1 PCE Engenharia S/S Ltda, Rua Félix da Cunha, 322 Porto Alegre RS - Brazil

Keywords: Anode Furnace Baking, Mathematical Modelling, Conjugate Heat Transfer, Combustion Modelling

Abstract

The quality of the anode used in the aluminum industry depends strongly on the baking process. In general, it is desirable to achieve a more uniform temperature inside the anode during the heating process, combined with lower soaking time and energy consumption.

The main objective of this study is to present a 3D computational model able to take into account a large number of phenomena and parameters that play a role in the baking process, such as, coupled fluid flow and transient heat transfer, burning of volatiles, fuel combustion, radiation and control system representation. The simulations were done using the commercial CFD code CFX 5.6. This model can be used as a powerful tool in the development of new furnaces and retrofitting existing ones.

Introduction

Since the adoption of the prebaked anode technology in reduction cells by the smelters, the control of anode baking in the ring furnaces influences the pot performance.

In general, each ring furnace has 2 or more fire groups. Each fire group is composed of typically 10 to 15 sections: 3 anode preheating sections and 3 firing sections and 4 to 9 anode cooling sections, as shown in Figure 1.

The green anodes start at the ambient temperature and reach around 600°C at the end of the preheating phase, they continue heating up to 1000°C to 1200°C in the fire sections. Then the baked anodes cool down in the cooling sections, giving heat to the gas that preheats before it enters into the fire sections. The entire process takes about 240 - 300 h.

Anode baking is by far the most expensive step in anode production. Fuel supply and refractory repair represent approximately 15% of total anode manufacturing cost. Anode baking furnace fuel consumption in the range of 2 – 4 GJ/ton has been reported. The baking process is very complex and mathematical modelling is a good tool for understanding and control of the process as well as for the furnace design.

Many mathematical models of baking furnaces have been published. Some of these aim at the furnace operation [1-2]. These models are one or two dimensional and simulate the dynamics of the process. They assist in optimizing the furnace operation parameters. Other more detailed models are three dimensional and are used for furnace design or specific studies of part of the process [3-4]. These models are steady state and do not represent the dynamics of the process adequately to be used for practical furnace optimization and control.

In this paper we present a three dimensional transient model of the furnace and show how this model can be used for flue design and for optimization of the furnace operation.

Model Description

A three-dimensional model was built comprising a typical longitudinal slice of a section between the centerline of a flue and the centerline of a pit. In the flue, all tie-bricks and baffles are represented in detail. In the pit were placed 14 anodes of 1600 x 650 x 650 mm size for a total of 7.38 ton. The green anode contains 13% of pitch. In the model, there are 3 preheating, 3 fire and 6 cooling sections with a fire cycle of 25 h. Symmetry was assumed on the centerline of the flue and the pit (Figure 2). Thus, heat loss from the outside pit through the side wall is not taken into account.

This model is applied sequentially to every section of the furnace in order to represent one whole fire section from preheating to firing and cooling.
The geometry was discretized using the CFX5.6 Build Mesher in around 350000 tetrahedrons and all the transient differential equations solved using the finite volume method.

The differential equations solved [5] are:
- Continuity equation for fluid flow;
- 3 momentum transport equations for the velocity components;
- 2 equations for turbulence (k-ε model);
- Heat transfer equation for the flue and solids;
- Transport equations for CH₄, O₂, H₂O and CO₂;
- P1 Radiation Model;

The algebraic equations solved are:
- Reaction rate (Eddy Dissipation Model) for fuel combustion;
- Equation of state to take into account the density and viscosity variation of the air with temperature;
- Air infiltration for each section;
- Release and burning of volatiles, dependent on anode temperature and mass;
- Fuel massflow, dependent on gas temperature in the flues (control system representation)

Preheating Sections
The first three sections are the preheating sections, where the anode starts at ambient temperature and receives heat from two sources:
- from the hot gases coming out of the fire sections
- from burning of the volatiles, released by the anode

The main heat loss of the system occurs from the top surfaces exposed to the atmosphere due to natural convection and radiation. The convection heat transfer coefficient is calculated from a semi-empirical correlation found in the literature [6]. The radiation from the top surface is included as convection. The heat transfer coefficients are given by:

$$ h_{conv} = 0.15k \left( \frac{g \beta}{\alpha V} (T_s - T_\infty) \right)^{1/3} $$  
(1)

$$ h_{rad} = \sigma \epsilon \left( \frac{T_s^4 - T_\infty^4}{T_s^4 - T_\infty^4} \right) $$  
(2)

Where g is the gravity, β is the expansion coefficient, k, α and V are, respectively, the thermal conductivity, thermal diffusivity and viscosity of the air. T_s is local surface temperature and T_\infty is the environment temperature.

These coefficients are dependent on the surface temperature and the model recalculates them in each timestep according to the current surface temperature distribution. The environment temperature was assumed to be 30°C. This heat loss is present during all the simulation (combustion and cooling phase also). Another less significant heat loss is to the ground. This heat loss is taken into account by modelling all the soil (3.5 m) below the furnace structure, which is influenced by the temperature of the furnace.

Model Inputs. In the preheating sections, the gas temperature increases towards the fire sections. The inlet fluewall temperature was obtained by iteration in order to adjust the influence of the volatiles burned, this adds energy sources to the gas. Inside the flue, the gases gain extra heat given by the volatiles burned (CH₄, H₂ and tar). This heat is a function of the mass of volatiles present in the anode and the anode temperature. Typical values for the percentage of volatiles in the green anode can be found in the literature [4] and also the volatile release rate [1-2] and the percentage of the volatiles burned in the flue [1].

During the baking process, we assume that 32% of the lighter components of the pitch are evolved from the anodes as volatiles. In the model, the volatiles burn as soon as they are released. The amount of power obtained from volatile burn is a function of the anode temperature, shown in Figure 4 and obtained by a parabolic fitting to published curves [1]. The model applies heat sources in the flue volume according to the current anode average temperature following the curves of Figure 4.

Another phenomenon to consider is air infiltration. Infiltration is a function of the local pressure difference between the outside and the inside of the flue. It takes place in the preheat and fire sections since these have under-pressure in the flues. The under-pressure increases towards the exhaust and so do the infiltration and the mass flow rate.

There is a relationship between the pressure profile and the gas massflow along the furnace. In our sample furnace the pressure drops were specified from a simple flow model in all sections. The results for massflow are shown in Figure 5.
Fire Sections
Fire sections have an additional feature that was included in the model: the combustion of natural gas. In the fire sections the gas temperature is fixed as the target temperature.

First, we calculate the gas flow in the fire sections, including combustion, as a steady state flow in order to use these velocity fields as inputs into the transient simulation. This is a good approximation since the control system keeps the flue temperature nearly constant, and the infiltration in these sections is small due to low pressure difference between the flue and the outside air. For this, the massflow rate at the flue inlet is specified as the average of F1, F2 and F3 from Figure 5. A target temperature of 1210°C was assumed. The natural gas massflow rate was also specified.

Combustion of natural gas is simulated using the Eddy dissipation model, that is best applied to turbulent flows when the chemical reaction rate is fast relative to the transport processes in the flow. According to this combustion model the reaction rate is proportional to a mixing time defined by the turbulent kinetic energy $\kappa$ and dissipation rate $\varepsilon$. The expression for the reaction rate is:

$$\text{RATE} = -B \rho \left( \frac{\varepsilon}{\kappa} \right) \exp \left( -\frac{E}{RT} \right) m_{\text{fuel}} m_{\text{O2}}$$  \hspace{1cm} (3)

Where $m_{\text{fuel}}$ and $m_{\text{O2}}$ are fuel and oxygen concentrations respectively, $\rho$ is the density, $B$ is the pre-exponential factor, $E$ is the activation energy, $R$ is the universal gas constant and $T$ is the temperature.

In the next stage we perform the transient simulation of each fire section. The temperatures calculated in the last preheat section are the initial condition for the front fire. The velocity distribution is specified as described in the previous paragraphs. All the other variables are recalculated in each time step. The natural gas injection is now controlled by the target gas temperature shown in Figure 6, this modifies the respective inlet species concentration in the model.

Radiation heat transfer is an important phenomenon that occurs mainly in the firing sections. It is also included in the transient model using the P1 method available in the CFX solver.

Results for Different Modelling Cases and Discussion

Two models were built in order to show, results for a typical fluewall geometry found in the industry (Case1), and results for an improved fluewall geometry, including different tie brick and burner positions, 100 mm thicker insulation bricks in the bottom and in the top of the fluewall (Case 2). These geometries are shown in the Figure 7.

Results for Preheating Sections:
Figures 8 and 9 show the results at the end of the preheating. In both cases this part of the process took into account 3 preheating sections, i.e., 75 h.

The velocity distributions show that Case 2 has a more uniform velocity pattern, which leads to an improvement of the heat exchange between the gases and fluewall.
Results for Fire Sections:
In the fire sections, the combustion of the fuel (natural gas) was calculated. It was possible to model the flame shape, heat generation and the flame influence on the flow and wall temperature distribution. Results shown below are at the end of 75 h of firing (3 sections).

The velocity distributions show that the improved tie brick arrangement in Case 2 gives a more uniform velocity pattern, which leads to improved heat exchange between the gases and fluewall. This can be seen in Figures 12 and 13 that show the heat transfer coefficient “h [W/m²·°C]” on the fluewall which has a better distribution in Case 2. It can be seen that the regions with low heat transfer (dark blue) are smaller in the second case than in the first one.
The simulation gives also temperature (Figures 16 and 17) and gas concentrations (fuel, O\textsubscript{2}, N\textsubscript{2}, CO\textsubscript{2}, and H\textsubscript{2}O). It is important to note that the upstream burner has 50% more fuel injection, resulting in higher temperatures and bigger flames.

The final anode temperatures, presented in the Figures 18 and 19 show, in general, an improvement in the anode temperatures for the Case 2 with higher temperatures in the bottom, top and middle of the anode pit. Better heat transfer between gas and fluewall combined with reduced heat losses are responsible for the improved anode quality in Case 2.

<table>
<thead>
<tr>
<th>Geometry</th>
<th>Case 1</th>
<th>Case 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maximum temperature (°C)</td>
<td>1122</td>
<td>1124</td>
</tr>
<tr>
<td>Minimum temperature (°C)</td>
<td>961</td>
<td>1001</td>
</tr>
<tr>
<td>Maximum – Minimum (°C)</td>
<td>161</td>
<td>123</td>
</tr>
<tr>
<td>Average temp. (°C)</td>
<td>1086</td>
<td>1101</td>
</tr>
<tr>
<td>Standard deviation (°C)</td>
<td>24</td>
<td>18</td>
</tr>
<tr>
<td>% anode volume w/ temp. below 1050°C</td>
<td>8%</td>
<td>2%</td>
</tr>
</tbody>
</table>

In the anodes, the maximum average temperature is achieved a few hours (12 h in our cases) after the end of the fire due to the thermal inertia effect. At this time, the difference between maximum and minimum final anode temperature is 37°C smaller for Case 2. The improved flue can bake anodes more homogeneously.
The entire baking process was predicted, including the simulation of the cooling sections. Calculated gas and anode temperatures are given in Figure 21 for Case 2.

The Table II gives the main heat losses for the typical flue (Case 1) and the improved flue (Case 2).

Table II. Top and soil heat losses (GJ)

<table>
<thead>
<tr>
<th>Geometry</th>
<th>Soil</th>
<th>Top of Fluewall</th>
<th>Top of Coke</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case 1</td>
<td>1.104</td>
<td>3.464</td>
<td>2.391</td>
</tr>
<tr>
<td>Case 2</td>
<td>0.819</td>
<td>1.297</td>
<td>2.258</td>
</tr>
</tbody>
</table>

The extra insulation added to the top of the fluewall was very effective with respect to the heat losses as we can see in the Table II and Figure 22. The increased bottom insulation has a minor effect in the total heat loss but helps with a better anode temperature homogeneity.

The computational time necessary to simulate the whole process was approximately 50 h in a Pentium IV 3.0 GHz processor running on Windows 2000.

Conclusions

A transient three-dimensional model of the anode baking furnace was developed using the commercial CFD code CFX5.6. In this paper, a case study was presented comparing two different fluewall geometries with the same operating parameters.

The model is able to simulate the entire baking process (preheating, firing and cooling) using very detailed geometry. The model is also able to take in account specific burner configurations (type of fuel and burner geometry) that will influence the size, temperature and location of the flame. In the case study, it was shown how an improved flue design improves final anode baking temperature homogeneity.

The model has already been used and validated on two different industrial furnaces. It has been demonstrated that it is a good tool for flue geometry optimization in new and existing furnaces.

Acknowledgement

The authors wish to thank Mr. Vinko Potocnik for his encouragement, suggestions and contributions. Many thanks are also due to Mr. Raymond Perruchoud, Mr. Felix Keller and Mr. Peter Sulger from R&D Carbon Ltd. for their critical and careful review of the manuscript.

References

5. CFX 5.6 USER MANUAL - Solver Theory.