

CFD Modelling of a Near Technical Scale Cell for Copper Refining Electrolysis

I. Filzwieser, O. Zach and A. Filzwieser

Today it is possible to describe a lot of metallurgical processes with the use of mathematical and physical models. The finite element method (FEM) or the method of the finite volumes (Computational Fluid Dynamics, CFD) are used methods for mathematical modelling. For example in this article it is shown how the CFD calculation was used to optimize the bath circulation in the inlet and outlet of the electrolyte and the number of electrodes of a copper refining electrolysis. The goal was to run the process in the pilot plant as close as possible to the process of technical scale copper refineries.

CFD-Modellierung einer halbtechnischen Zelle einer Kupferraffinationselektrolyse. Heutzutage ist der Einsatz von mathematischen und physikalischen Modellen zur Beschreibung metallurgischer Prozesse üblich. Zu den gängigen Rechenmodellen gehören die Finite-Elemente-Methode (FEM) und die Methode der Finiten Volumina (CFD). Nachfolgend wird gezeigt, wie anhand einer CFD-Modellierung eine optimale Bauweise einer Kupferraffinationsversuchsanlage gewährleistet werden kann. Es galt, Elektrolytz- und -abfluss sowie auch die Anzahl der zu verwendenden Elektroden zu optimieren. Ziel war es, eine Pilotanlage zu bauen, deren Prozessparameter möglichst vergleichbar mit denen einer Betriebsraffinationselektrolyse sind.

1. Introduction

Nowadays the use of mathematical modelling is widely accepted in all industrial sectors. During the last years the use of CFD modelling for the copper electrolysis was developed at the Department of Nonferrous Metallurgy at the University of Leoben¹⁻⁴. An important point for starting experiments in near technical scale is the possibility to compare the conditions in the laboratory with industrial conditions. Especially the mechanism of flow conditions is of big interest. To define the fluid flow conditions in the laboratory cell in a proper way means that the lab scale experiments are better comparable with the plant parameters.

The above-mentioned department is now working on a big international project in copper refining electrolysis. For this project new electrolysis cells were built. To get good comparable conditions a CFD calculation was done hand-in-hand with the RHI Refractories Technology Centre Leoben, Austria.

2. Fundamentals of CFD Modelling

The numerical calculations of fluid flow or heat and temperature distribution is possible, if the physical laws can be transported into mathematical equations, normally into differential equations.

Each single differential equation is linked to a certain conservation law and consists of one physical value as a dependent variable. It is assumed that the different factors influencing the physical value are in equilibrium.

The dependent variables of these differential equations are used as specific value, this means, it is calculated of one device volume. Examples for these are the velocity (momentum per device volume), the mass or rule of agitation and the specific enthalpy/energy. The temperature, which is often given as dependent variable,

is not a specific value, because it is a component in a fundamental equation with the energy/enthalpy equation.

The single terms of a differential equation show the influence of an imaginary unit volume. An example of the continuity equation, the conservation of mass, will be calculated of a two dimensional field.

Fig. 1. Two dimensional area field⁵

In Figure 1 an element in the two dimensional space with the edge dx_1 and dx_2 is shown. The continuity equation is the result of the calculation in both directions x_1 and x_2 taking care of the density change with time according to the mass input per volume in kg/m^3 .

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_1}(\rho u_1) + \frac{\partial}{\partial x_2}(\rho u_2) = 0 \quad (1)$$

The differential equation, which describes the conservation of momentum for one direction of one volume element of a Newton's fluid, can be calculated in a similar way as described before. With u for the velocity vector and u_1 as the velocity component in x_1 direction it can be described:

$$\frac{\partial \rho u_1}{\partial t} + \text{div}(\rho u u_1) = \text{div}(\mu \text{grad } u_1) - \frac{\partial p}{\partial x_1} + B x_1 \quad (2)$$

with μ as the viscosity, p as the pressure and B as "body force" in x_1 direction, for example the gravitational acceleration. The singular parts of this differential equation describe the different kinds of momentum transportation. The first part of the term $\delta(\rho u_1)/\delta t$ means the rate of momentum exchange (changing the velocity) per time interval, the term $\text{div}(\rho u u_1)$ the convective flow of the momentum, which will be transported by the general velocity field u . The term $\text{div}(\mu \text{grad } u_1)$ shows the diffuse part of the momentum transport, with the meaning of $\mu \text{grad } u_1$ in combination with Fick's diffusion law. The viscosity μ corresponds to the "diffusion coefficient" of the momentum and $\text{grad } u_1$ the differential of the momentum of the volume element. The term $\delta p/\delta x_1$ shows the part of the pressure force to the complete momentum calculation. The momentum conservation equation is called Navier-Stokes-Equation.

Dipl.-Ing. Iris Filzwieser, Department of Nonferrous Metallurgy, University of Leoben, Franz-Josef-Straße 18, A-8700 Leoben; *Dipl.-Ing. Oliver Zach*, RHI AG, Magnesitstraße 2, A-8700 Leoben, *Dipl.-Ing. Dr. Andreas Filzwieser*, RHI Non Ferrous Metals Engineering AG, Magnesitstraße 2, A-8700 Leoben.

The described conservation of momentum equation is only valid in the case of laminar conditions, but turbulent flows occur in nearly all technical processes. Only in very special cases and under extreme computing time it is possible to solve instational turbulent flows. At numerical flow calculation it is common to use time dependent and ensemble values of the variables, for example velocity, density, etc.. For these calculations it is usual to assume that in the area of an average a fast and hazardous fluctuation exists.

In the equation of conservation the current variables on a volume element through the sum of the ensemble calculated values $\langle u \rangle$ and a deviation u' of the calculated value replaced for the velocity component u is:

$$u = \langle u \rangle + u' \quad (3)$$

The correlation of the velocity fluctuation is not known and therefore it must be calculated with a turbulence model. This kind of solving the equation system is called to close the system. In this case it is important that the apparent transverse strain is mainly caused by the bigger turbulence conglobes. These big turbulence conglobes transfer through vortex filament extension and in case of the shear instability of the flow their energy to smaller whirl, till the velocity gradient of the smallest turbulence elements is very steep, a transformation into the inner energy. This transport of energy into the turbulence energy spectrum to bigger wave numbers is called turbulence energy cascade.

The feed of energy into these cascades is not depending on the viscosity. It will take place through transmission of energy of the average movement of the apparent shearing strain on the big turbulence elements. This energy is given from one small turbulence element to the next till it is at the dissipation. This is also the reason that the distribution of the average velocity at turbulent flow depends only slightly on the Reynolds number, although the viscosity causes the energy.

In case of going through the energy cascade the information of direction of the velocity will be lost. This means that the fluctuation energy during the cascade will be distributed in all directions. The smallest turbulence elements are isotropic. This is common at turbulent flows with big velocity gradients or not twist used rotating flows. These flows are called local isotropic.

The dependent variables Φ must be split into an average value $\langle \Phi \rangle$ and the fluctuated component Φ' :

$$\Phi = \langle \Phi \rangle + \Phi' \quad (4)$$

For a static stationary flow f the temporal average is:

$$\langle \Phi(x_0) \rangle = \text{Limit}_{t_0 \rightarrow \infty} \frac{1}{2 \cdot t_0} \int_{t=T_0}^{t=t_0} \Phi(x_0, t) dt \quad (5)$$

Many different turbulence models are used. For practical and technical work the so-called k - ϵ -Model from Harlow and Nakayama [5] will be taken. The k - ϵ -Model solves two more partial differential equations for the turbulent kinetic energy k

$$k = \frac{\overline{u_i'^2}}{2} \quad (6)$$

and their dissipation ϵ

$$\epsilon = C_\mu^{3/4} \frac{k^{3/2}}{l_t} \quad (7)$$

In the equations l_t stands for a characteristic turbulent whirl length. For going deeper into this field it is

necessary to familiarize with the special literature. The use of CFD modelling will now be shown by an example.

3. Example of the CFD Modelling

3.1 Optimization of the flow conditions

For this example an electrolysis cell for copper refining was chosen. The target was to construct and build a cell where the flow conditions are simulated to be very close to the conditions of an industrial cell.

In order to carry out near-industrial tests a semi-technical experimental plant was built. In this installation the height of the test electrodes corresponds to that of industrial ones. The width was chosen to be 10 cm for the anodes and 11 cm for the cathodes. To find out the best cell/electrode arrangement two variants have been tested:

- I: One cathode between two anodes
- II: Three cathodes with two anodes in between

The evaluation was done by the following criteria: All experiments have to be performed as near as possible to industrial scale; that means that the arrangement of the electrodes leads to similar fluid flow conditions as industrially. The different behaviour of the anode air side and mould side and its influence on the cathodic copper deposition must be reflected. For a systematic investigation of this effect it has to be taken into consideration that a mixing-up of the electrolyte between cathode and anode air side on the one hand and between cathode and anode mould side on the other hand has to be minimized. The cathodic current density – one of the most important parameters – must be set exactly ($\pm 0.5 \text{ A/m}^2$).

The advantage of variant I is that the difference in electrolyte composition between cathode – air side and cathode – mould side should be more pronounced, because the active anode surface is lower than with variant II. The disadvantage, however, is that one half of each anode has to be deactivated. In the first tests this was done by covering with a foil. But problems arose due to the insufficient durability of foil and glue. The latter dissolved in the acid electrolyte as an impurity.

In case of variant II it is necessary to cover the outside of the two cathodes to achieve industrial conditions. These sides could be covered by a Macrolon® plate without glue. For the test results only the cathode in the middle was used and no inaccuracies could occur since the two outer cathodes did not influence the current density at this cathode surface.

Fig. 2. Cell with one cathode

Fig. 3. Cell with three cathodes

Fig. 4. Fluid circulation at the bottom in I

Fig. 5. Fluid circulation at the bottom in II

To get a better understanding of the fluid flow dynamics in the semi-technical cells a fluid flow simulation was done for both variants.

The experimental equipment was modelled (scale 1:1). 200,000 single cells built up the volumes of the CFD calculation. The following limiting conditions have been assumed: Only the forced convection was taken into consideration. The natural, free convection resulting from the density differences of the electrolyte near to the electrode surfaces will only be observed in the final work. Furthermore a pure steady streamline flow was taken

into account. For the upper surface of the electrolyte (a "free" surface) the term "specified shear stress = 0" was used, which represents a wall without shear stress. The electrolyte density was taken as 1181.81 kg/m^3 and its viscosity as 0.00112586 kg/ms .

The two different flow conditions are presented in Figure 2 to Figure 5. The electrolyte with the inhibitor is better spread over the whole electrode surfaces and distances in arrangement II. So, for the set-up of the near-technical scale cells this arrangement was chosen.

3.2 Set-up of near-technical scale cell

The cell is built from Macrolon® of 1 cm thickness and has the following dimensions: length 40 cm, width 16 cm, height 100 cm, all inside measures. The whole cell is joined together with glue. It can be seen that the cells consist of three parts: An inlet box (right), the electrolysis cell (middle), and an outlet box (left). The electrolyte stream enters the cell by an opening between inlet box and cell, 4 cm above the bottom and 2 cm wide over the total width. The same opening exists for the outlet 4 cm under the upper surface, so that the electrolyte has to flow through the cell diagonally.

The standard experimental parameters are:

Temperature:	$65^\circ\text{C} \pm 2^\circ\text{C}$
Current density:	350 A/m^2
Inhibitor:	100 g of glue per t of copper
Thiourea:	60 g per t of copper
Cl-:	50 mg/l
Electrolyte:	45 g Cu /l 175 g H_2SO_4 /l
Duration:	6 days = 144 hours

Figure 6. Electrolysis cell

Tests were made with different anode qualities. For all experiments the cathode crop was six days, only depending on the weight of the anodes.

For example some cathodic copper precipitations is shown in Figure 7.

Fig. 7. Cathodic copper precipitations

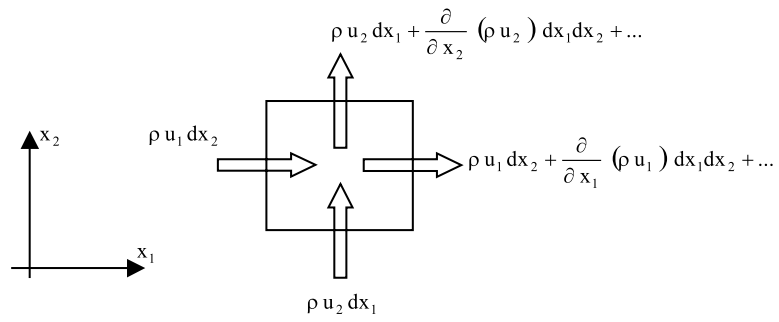
4. Summary

The practically orientated and the reproducible investigations in a copper refining electrolysis in a pilot plant are mainly driven by the fluid flow conditions in the cell. It is, therefore, necessary to closely monitor the construction of the pilot plant. Results with the described cell are comparable with those from the industry.

Literature

- ¹ Filzwieser, A., A. Lackner, and K. Hein: Möglichkeiten zur Strömungsberechnung in Elektrolysezellen unter besonderer Berücksichtigung des Stofftransportes. Schriftenreihe der GDMB, Heft 81 (1997), 161–172. – ² Hein, K., G. Hanko, A. Filzwieser, and M. Stelter: Untersuchungen zur Hydrodynamik bei der Kupfergewinnungselektrolyse. BHM 144 (1999), 6–13. – ³ Hanko, G., K. Hein, and A. Filzwieser: Visualisierung und Quantifizierung der Strömungsverhältnisse in einer Kupfergewinnungselektrolyse. Erzmetall 52 (1999), 226–235. – ⁴ Filzwieser, A., K. Hein, G. Hanko, and H. Grogger: Application of two phase hydrodynamic modelling to an electrowinning cell. Proc. of the 4th Int. Conference „Copper 99 – Cobre 99“, Vol. III, Phoenix, Arizona, USA, 10.–13. 10. 1999, 695–709. – ⁵ Harlow, F., and P. Nakayama: Transport of Turbulence Energy Decay

42/Filzwieser



$$\frac{\partial}{\partial t} (\rho dx_1 dx_2) = \rho u_1 dx_2 + \rho u_2 dx_1 - \rho u_2 dx_1 - \frac{\partial}{\partial x_2} (\rho u_2) dx_2 dx_1 - \rho u_1 dx_2 - \frac{\partial}{\partial x_1} (\rho u_1) dx_1 dx_2 - \dots$$

Abb. 1

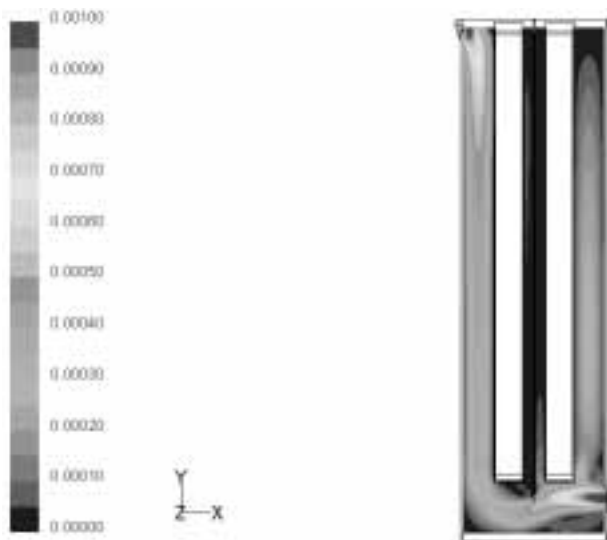


Abb. 2

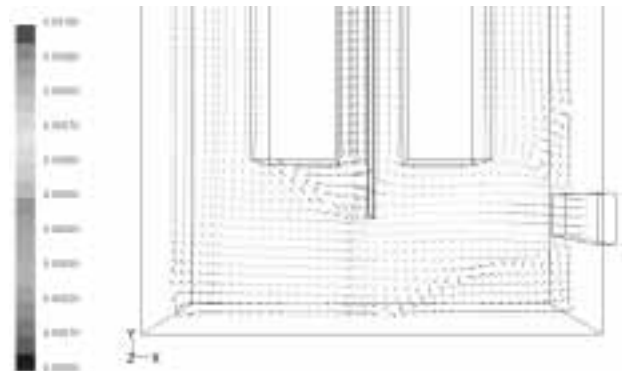


Abb. 4

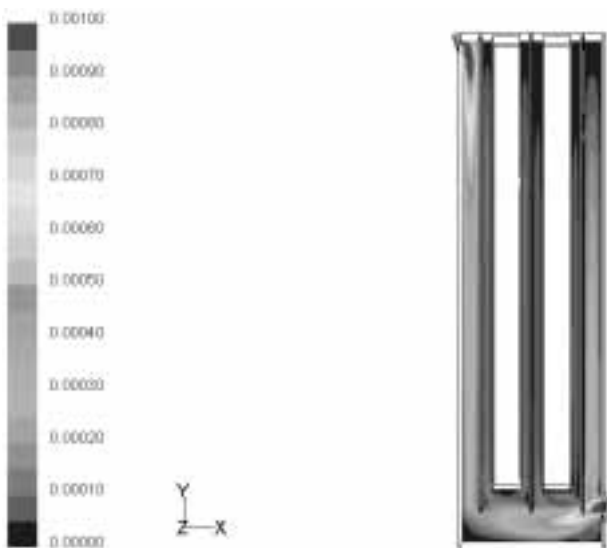


Abb. 3

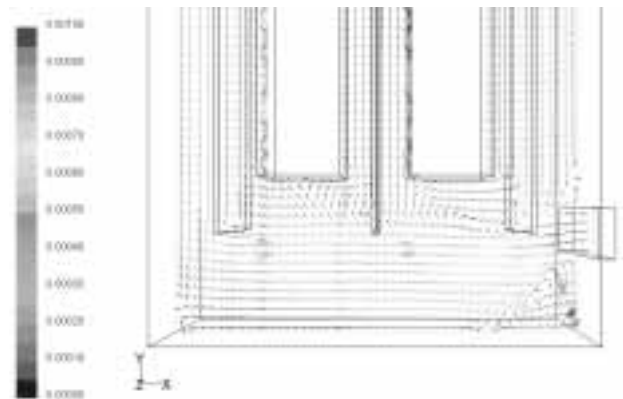


Abb. 5

42/Filzwieser

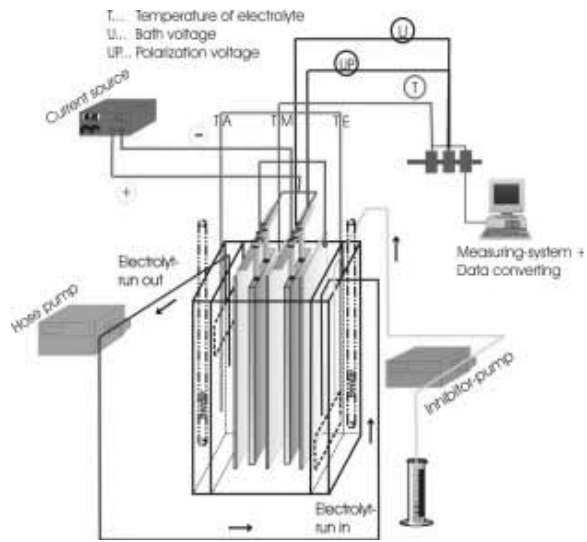


Abb. 6

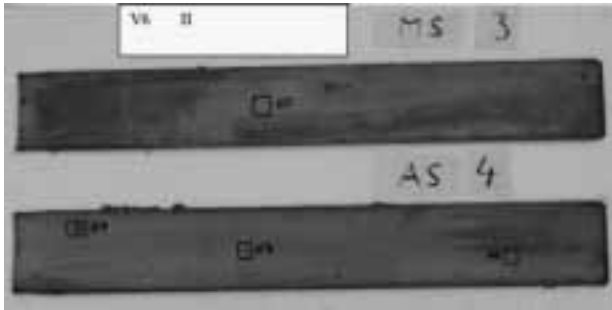


Abb. 7