

## CFD - Modelling of the electrolyte flow in a copper electrolysis cell

I. Bacher, H. Antrekowitsch, Institute of Nonferrous Metallurgy, Montanuniversity Leoben

O. Zach, A. Filzwieser, RHI AG, Leoben and Vienna

Dipl.-Ing. Iris Bacher (31) studied metallurgy at the Montanuniversity Leoben, Austria. Since 2001 she has been working as a research assistant at the Institute of Nonferrous Metallurgy. During this period of time she works on her graduation with the topic of mathematical and physical Modelling in the copper electrolysis.

H. Antrekowitsch (34) studied from 1989 to 1994 metallurgy at the University of Leoben and worked from 1995 to 1999 as assistant at the Department of Nonferrous Metallurgy. He graduated at 1998. Since 1999 H. Antrekowitsch is working at the Department and since April 2002 he is the head of the Christian Doppler Laboratory of Secondary metallurgy of Nonferrous Metals.

Dipl.-Ing. Oliver Zach (32) studied process technology, Department of terotechnology, at the TU Graz and finished the diploma study at 2001. Since 2001 he is working at the CFD & FEM –division at the RHI AG, Leoben, Austria.

Dipl.-Ing. Dr. Andreas Filzwieser (34) studied Metallurgie at the University of Leoben and worked from 1996 to 2000 as assistant at the Department of Nonferrous Metallurgy. He graduated at 2000. Since January 2001 he is working at the RHI AG in Vienna for the Marketing Manager of the Business Unit Nonferrous Metallurgy and since March 2002 he is the General Manager of the RHI Non Ferrous Metals Engineering GmbH.

Abstract:

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) could be the method for mathematical modelling. For example in this article it is shown how the CFD calculation was used to optimise the placement of inlet and outlet of the electrolyte and the number of electrodes of a copper refining pilot plant. The goal was to run the process in the pilot plant as close as possible compared to the process of real copper refineries.

## **1 Introduction**

These days the use of mathematical modelling is far accepted in all industrial sectors. During the last years the use of CFD modelling for the copper electrolyse was developed at our institute [1,2,3,4].

An important point for starting experiments in near technical scale is the possibility to compare the conditions in the laboratory with industry conditions.

Especially at the hydrometallurgy the mechanism for flow conditions in the plant 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 Institute of Non Ferrous Metallurgy at the University of Leoben, Austria, 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 by hand with the RHI Research centre Leoben, Austria.

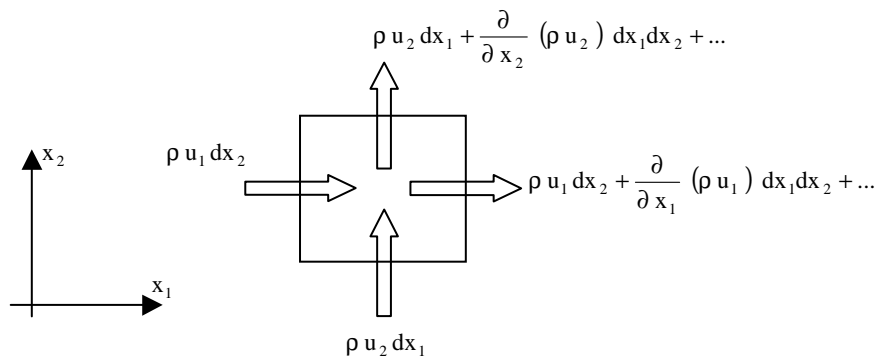
## 2 Fundamentals of the CFD-Modelling

The numerical calculations of fluid flow or heating- 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 depended variable. It is assumed that the different factors influencing the physical value are in equilibrium.

The dependent variable 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 that is often given as dependent variable is not a specific value, because it is compound in a fundamental equation with the energy/enthalpy equation.

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



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

Figure1:Two dimensional area field [5]

Looking at 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  $x_2$  with taking care of the changing in time of the density– according the mass input per volume in  $kg/m^3$ .

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

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

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

With  $\mu$  as the viscosity,  $p$  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 caused 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 to the Fick’s diffusion law:

The viscosity  $\mu$  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 will be called as Navies-Stokes-Equation.

The now described conservation of momentum equation is only working 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 is it possible to solve instational turbulent flows. At numerical flow calculation it is common to use time depended and ensembled values of the dependant variable, for example velocity, density, etc.

For these calculation it is use to think of that in the area of an average a fast and hazardous fluctuation will be there.

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 example B for the velocity component u:

(Equ. 3) 
$$u = \langle u \rangle + u'$$

The correlation of the velocity fluctuation is not known and that's why it must be calculated with a turbulence model. This kind of solving the equation system is called as closing 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 gives 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, that the transformation into the inner energy.

These transport of energy in the turbulence energy spectrum to bigger wave numbers will be called as turbulence energy cascade.

The feeder of energy into these cascades is not depending on the viscosity. It will taken place through transmission of energy of the average movement of the apparent shearing strain on the big turbulence elements. This energy will given from on 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 only less depend on the Reynolds number, although the energy loosing is caused by the viscosity.

In case of the 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 as local isotropic.

The dependent variables  $\phi$  must be transported into an average value  $\langle\phi\rangle$  and the fluctuated component  $\phi'$ :

(Equ. 4) 
$$\phi = \langle\phi\rangle + \phi'$$

For a static stationary flow  $\phi$  is the temporal average:

(Equ. 5) 
$$\langle\Phi(x_0)\rangle = \mathop{\text{Limit}}_{t_0 \rightarrow \infty} \frac{1}{2 \cdot t_0} \int_{t=-t_0}^{t=t_0} \phi(x_0, t) dt$$

Many different turbulence models are used. For practical and technical working with 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 kinetically energy k

$$\text{(Equ. 6)} \quad k = \frac{\overline{u_i^2}}{2}$$

and their dissipation ε

$$\text{(Equ. 7)} \quad \varepsilon = C_{\mu}^{3/4} \frac{k^{3/2}}{l_t}$$

In the equations which are described  $l_t$  is for a characterise turbulent whirl length. For going deeper into this field it is necessary to go into the special literature.

With an example the use of CFD-modelling will now be shown.

### 3 Example of the CFD-Modelling

#### 3.1 *Optimization of the flow conditions*

For this example an electrolysis cell was chosen. This cell should be used as a copper refining cell. The target was to construct and build a cell where the flow conditions are simulated to be very close to the conditions of a cell used in industry.

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 ( $\pm 0.5 \text{ A/m}^2$ ).

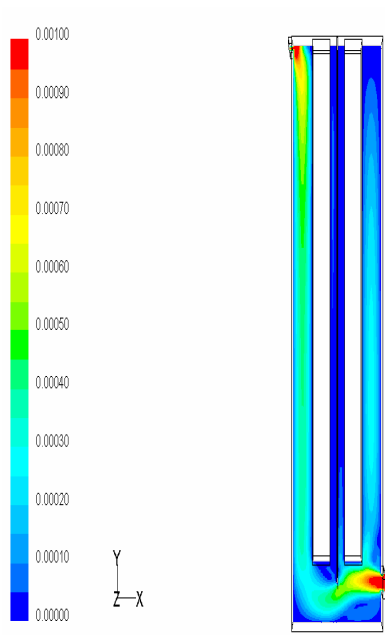


Figure 2 : Cell with one cathode

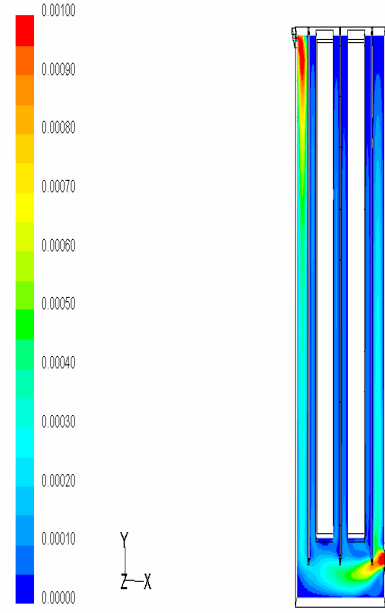


Figure 3: Cell with three cathodes

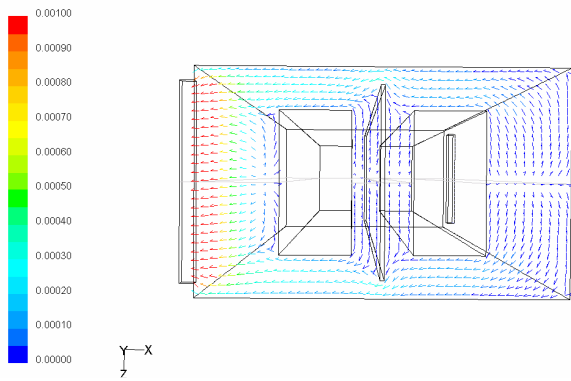


Figure 4: Fluid circulation at the top in I

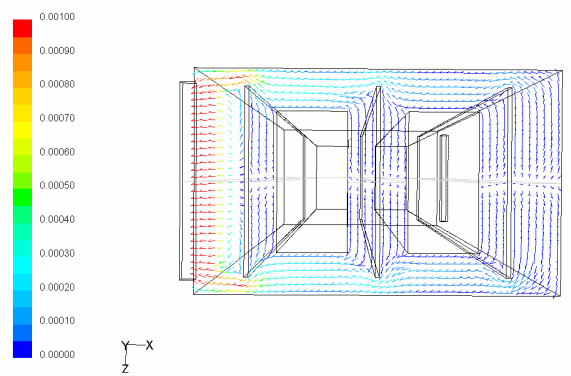


Figure 5: Fluid circulation at the top in II

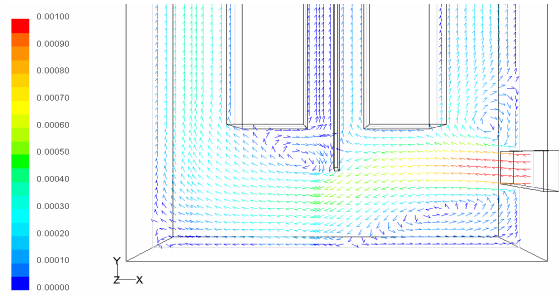


Figure 6: Fluid circulation at the bottom in I

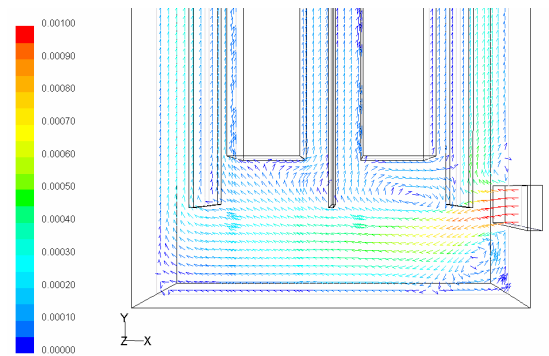


Figure 7: Fluid circulation at the bottom in II

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 there due to the insufficient durability of foil and glue. The latter dissolved in the acid electrolyte and went into the electrolyte as an impurity.

In case of variant II it is necessary to cover the outsides 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 current density at this cathode surface was not influenced by the two outer cathodes.

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). The volumes of the CFD calculation (Computational Fluid Dynamics) were built up by 200,000 single cells. 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 has to be observed in our future 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<sup>3</sup> and its viscosity as 0.00112586 kg/ms.

### *3.1 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 consists 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.

Our standard experimental parameters are:

Temperature: 65°C ±2°C

Current density: 350 A/m<sup>2</sup>

Inhibitor: 100 g of Glue per t of copper

Thiourea: 60 g per t of copper

Cl<sup>-</sup>: 50 mg/l  
 Electrolyte: 45 g Cu /l  
 175 g H<sub>2</sub>SO<sub>4</sub>/l  
 Duration: 6 days = 144 hours

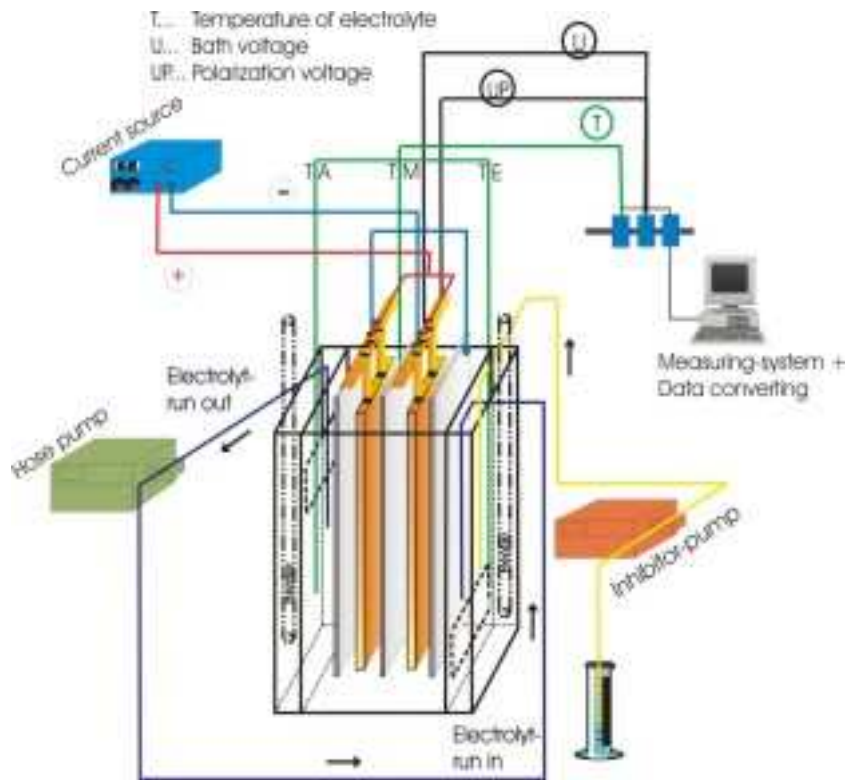


Figure 8: Electrolysis cell

We made tests with different anode qualities. For all experiments the cathode crop should be 6 days, only depending on the weight of the anodes.

For example some cathodic copper precipitations are shown in figure 9

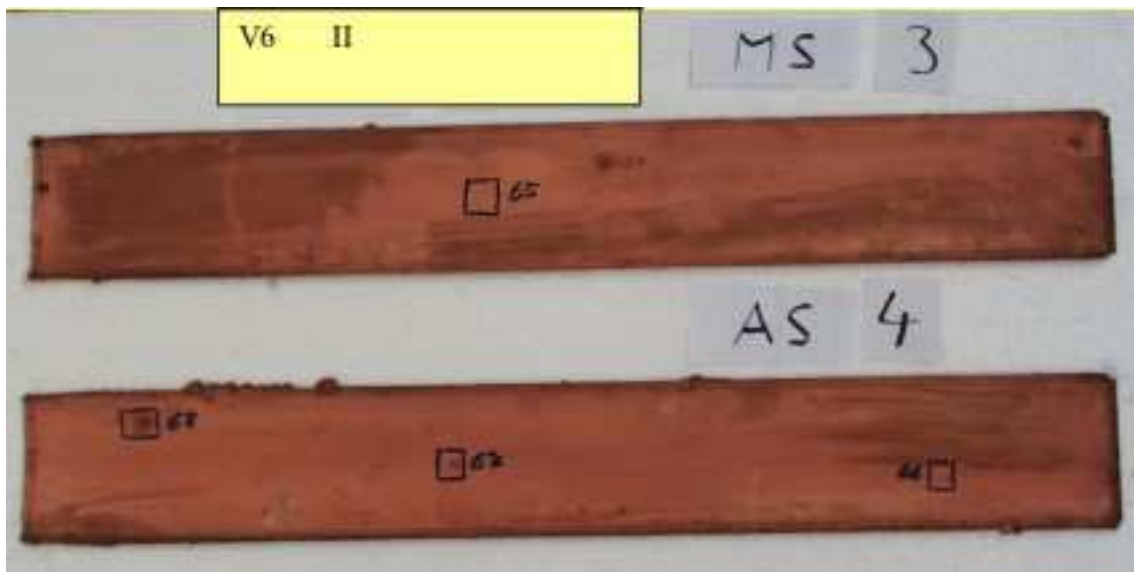


Figure 9: Cathodic copper precipitations

#### 4 Summary

The functionality and the repeatable investigations in a copper refinery 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 und 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 und M. Stelter: Untersuchungen zur Hydrodynamik bei der Kupfergewinnungselektrolyse. BHM 144 (1999), 6 – 13
  
- [3] Hanko, G., K. Hein und 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 modeling to an electrowinning cell. Proc. of the 4<sup>th</sup> Int. Conference „Copper 99 – Cobre 99“, Vol. III, Phoenix, Arizona, USA, 10.-13.10.1999, 695 – 709
  
- [5] F. Harlow, P. Nakayama: Transport of Turbulence Energy Decay Rate. Report LA-3845, Los Alamos Science Lab., University of California