Modelling and Design of a Forced Convection Network for Hall-Héroult Cells

S. Guérard
Rio Tinto Aluminum – CRDA, 1955 Boulevard Mellon, Jonquière, Qc, Canada, G7S 4K8
Corresponding author: sebastien.guerard@riotinto.com

Abstract

Forced convection networks (FCN) are now commonly used in the aluminum reduction technology to increase the heat transfer from the sidewalls of the cells. This both cools down the potshell and increases the ledge thickness, potentially leading to additional amperage creep in the smelter. Proper design of a FCN requires a combination of modelling tools: a computational fluid dynamics (CFD) model to predict the air flow pattern and heat transfer coefficients on the shell, a thermal model to evaluate the ledge response inside the cell, and pressure loss calculations to design the pipe network. In this work, we present the approach that has been developed at Rio Tinto Aluminum to design and optimize the FCN configuration. The models are validated based on measurements taken in the potroom. Finally, a case study illustrates how the approach can be applied.

Keywords: Forced convection network; computational fluid dynamics; cell modelling.

1. Introduction

Most aluminum smelters are steadily increasing their line amperage. In order to maintain the cells' thermal balance, such increases often need to be accompanied by higher heat dissipation. This can be achieved in a variety of ways, for example by modifying the cathode design (eg. more conductive sidewalls) or operating parameters (eg. higher metal levels).

One of the most interesting options to increase heat dissipation is to install forced cooling around the cell. In 2001, Pechiney patented a forced convection network (FCN) [1]. Since then, most high productivity plants in Rio Tinto Aluminum (RTA) have been equipped with this type of system. A schematic representation is shown in Figure 1. FCNs are intended to cover the entire pot and to be operated continuously. They consist of a main pipe surrounding the cell, which is connected to a series of nozzles providing air cooling approximately at the level of the bath-metal interface. These systems are operated at relatively low pressures (a few kPa) to minimize energy requirements. The main pipe should be large enough to have small head losses compared to the nozzles in order to maintain as uniform a flow rate as possible around the pot.

A FCN also confers additional benefits beyond simply extracting heat: it cools down the sideshell, which could otherwise become too hot when highly conductive sidewalls are used; it can be designed to target known weak points with more intense cooling, thus improving cell robustness; and the flow rate can be adjusted as needed, for instance to offset seasonal variations or to compensate for periods at different intensities.

This paper aims to present how such systems are designed within RTA. We will begin by describing the modeling and calculations tools that have been developed, followed by a comparison with experiment measurements, and we will end with a case study showcasing how such tools can be applied in practice.

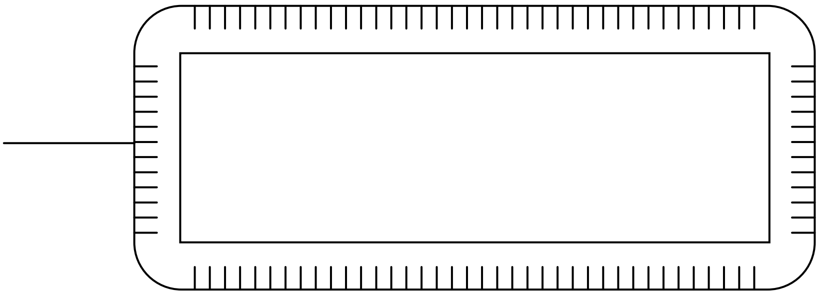
Figure 1. Forced convection network surrounding the electrolytic cell.

2. Modelling Approach

The three most important aspects to consider in the design of a FCN are the configuration of the nozzles, the impact on the thermal state of the cell, and the behavior of the pipe network. Our approach was to develop a separate model to study each of these components. This can lead to some back-and-forth, but offers the considerable advantage of making each model easy to manipulate and computationally inexpensive.

2.1 Nozzle configuration

Perhaps the most crucial step is to find a nozzle configuration that maximizes the heat transfer from the cell. Parameters of interest include the nozzle diameter, angle, distance from the shell, number per inter-cradle, as well as the position of the point of impact. Also relevant are the velocity and air temperature of the jet.

Several methods can be used. The simplest one is to rely on semi-empirical correlations. Jambunathan et al. [2] provides a good review of the available literature, and Martin [3] presents an easy-to-use correlation that predicts the heat transfer coefficients as a function of geometrical parameters and air flow rate. This approach can give a good first approximation of the performance that can be expected from the FCN. That being said, it’s also severely limited: it cannot account for several important aspects of the system, most notably the jet angle, the influence of the cradles or fins on the shell, the interactions between neighboring jets, the natural convection in the potroom, the large temperature gradient in the shell, and the exact nozzle geometry.

Another approach is to perform experimental parametric studies, either in a laboratory [4] or on a full-scale pot [5]. This is more reliable, but also time and resource intensive.

We have instead decided to rely on a computational fluid dynamics (CFD) model of the air jet and side shell. This allows us to account for all the relevant parameters, while still being able to rapidly test many different configurations. The basic geometry is shown in Figure 2. A half or a complete inter-cradle may be represented, depending on the exact configuration being studied. The model includes the nozzles, the air region surrounding the shell, as well as most of the cell’s lining. Both the air flow field and the conduction through the shell and lining material are explicitly calculated.

The air flow rate is imposed at the inlet of the nozzles. An additional inlet is present at the bottom of the air region, with an upward velocity corresponding to the natural convection measured in the potroom. The k-ε model of turbulence is used. Periodic boundary conditions are present on both sides of the inter-cradle. Radiation between all surfaces is included. On the solid side, the liquidus temperature is imposed on the ledge-liquid interface and on the top surface of the cathode. Adiabatic boundary conditions are used for all the other surfaces.

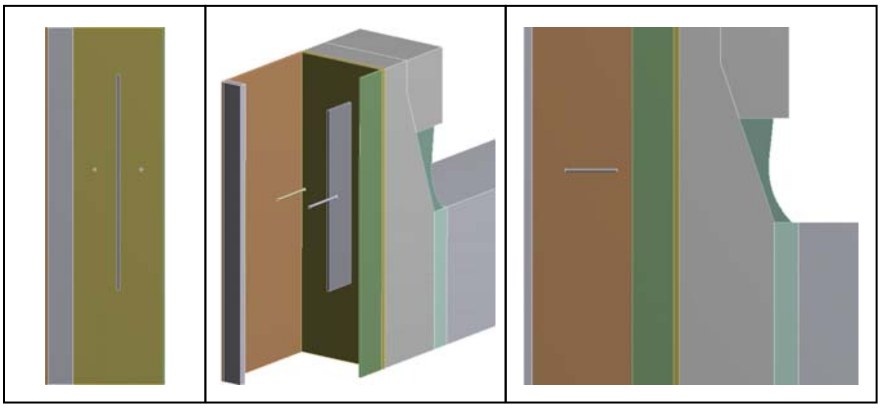

**Figure 2. Nozzle and cell geometry used in the CFD model
(note: the details of the lining have been removed in these images).**

2.2 Thermal impact on the cell

The most important simplification in the CFD model is that the ledge profile is fixed, while in reality it will vary with the FCN configuration and operating parameters. Therefore, while the model can be used to obtain the heat transfer coefficients – which are not very sensitive to the exact ledge profile or shell temperature – it cannot give us the net thermal impact on the cell, or even the correct impact on the shell temperature. For that, we need to couple it with another model that includes the ledge response.

One of our main tools for pot design is a finite-element thermo-electric model of the cell. Originally, the shell heat transfer coefficients were predicted by a combination of the previously mentioned correlation by Martin for impinging jet [3], correlations for natural convection, and analytical calculations for radiation. Now, however, we can perform a one-way coupling with the CFD results to apply a more accurate mapping of the heat transfer coefficients. The thermo-electric model can then be used to evaluate the impact of the FCN on the cell's operating conditions, such as ledge profile, bath superheat, and shell temperature.

A further possibility is to update the CFD model with the new ledge profile. In practice, however, the benefits of this two-way coupling have turned out to be negligible.

2.3 Pressure loss and mass flow rate distribution in the FCN

Taking a step back to look at the FCN as a whole, it is also important to be able to predict the pressure and flow rate throughout the system. The FCN is simply a pipe network with one fan and multiple outlets. These calculations can therefore be carried out by applying the well-known head loss equation and finding the solution that balances both the mass flow rate and the pressure drop at every point.

A Matlab® tool was developed for this task. It is shown in Figure 3. The network characteristics are imported from a spreadsheet, and the solution is then calculated iteratively until convergence is achieved. The results are shown in a graphical user-interface and can also be exported back to a spreadsheet.

Minor loss coefficients are estimated from the literature. In some specific cases, a local CFD model was built to complement this information. For instance, one variation considered eliminated the nozzles and simply used a perforated pipe. For this particular case, CFD was used to estimate the loss coefficient associated with the orifice as a function of the air velocity in the main pipe (see Figure 3).

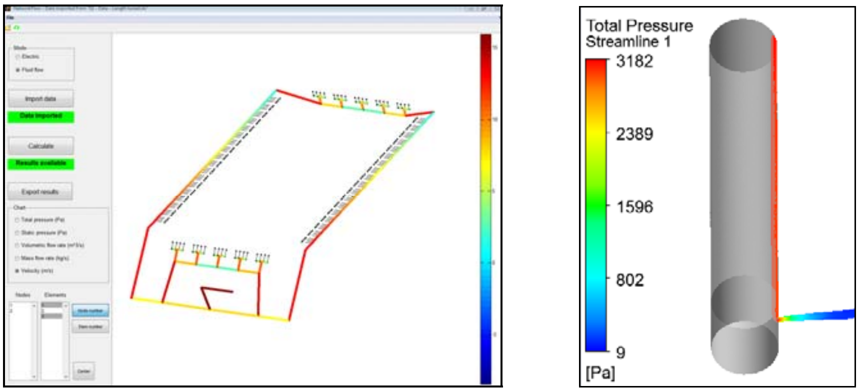

Figure 3. Matlab interface to perform the pipe network analysis (left) and determination of minor loss coefficient though an orifice exit (right).

By including the fan's pressure curve in this tool, we can predict the total mass flow rate, pressure, and energy requirements. The other important result is the variation from one end of the cell to the other. Ideally, we want a jet velocity that remains as constant as possible. If we find a large variation around the cell, this indicates that the design must be reevaluated.

3. Model validation

The CFD model accurately predicts the heat transfer coefficients, but not the shell temperature, and as such its results cannot be directly compared with experimental measurements. To validate the approach, we must instead use the results of the thermo-electric model. Figure 5 presents a comparison between a thermal image of the sideshell and the predictions of the model for the same region.

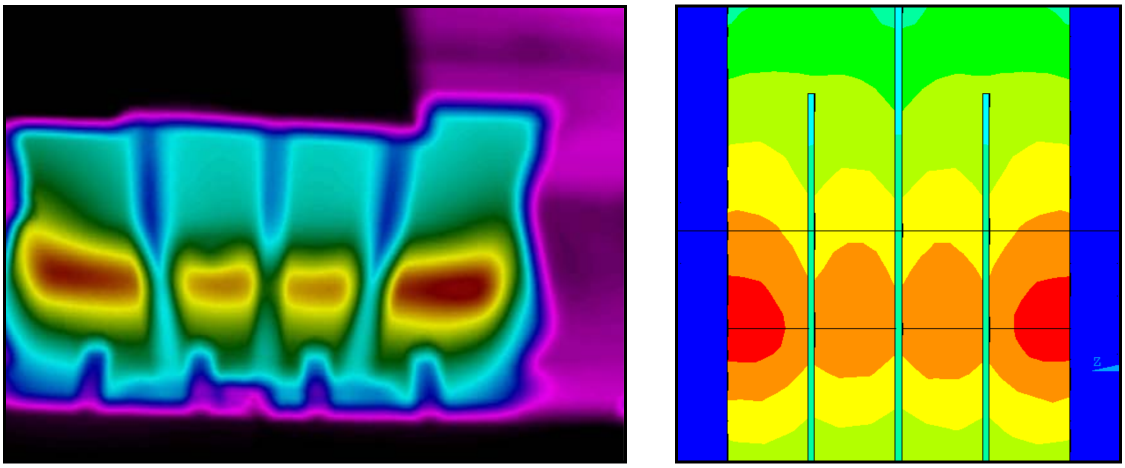

Figure 5. Comparison between a thermographic image of the sideshell (left) and the predictions of the thermo-electric model (right).

There are some differences – for instance, the isotherms seem more horizontal in the measurements – but overall the contours are quite similar. In particular, the model correctly captures the fact that the maximal temperature is found next to the cradles, with the section between the fins being somewhat colder.

To be more quantitative, we can measure the temperature along a vertical line and compare it to the model. A typical result is shown in Figure 6. For this type of comparison, systematic offsets aren't especially meaningful: this is a model of the entire pot, and any source of error in the cell's energy balance will cause a translation of the temperature profile. The more crucial aspect to consider is the profile shape. In this respect, using the results of the CFD has brought the model into much closer alignment with the measurements, which suggests that the underlying physics is well represented by this approach.

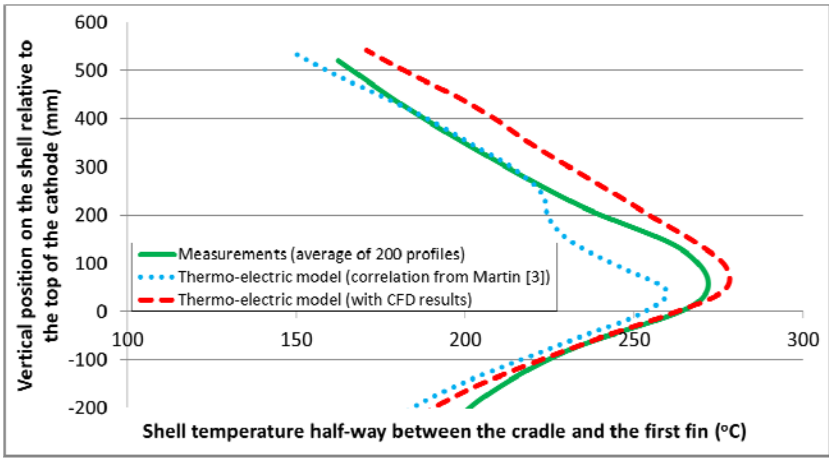

Figure 6. Shell temperature half-way between the cradle and first fin.

To validate the pipe network calculations, the pressure was measured at the entrance of the system for different total flow rates. The results show very close agreement between the measurements and the calculations, as illustrated in Figure 7.

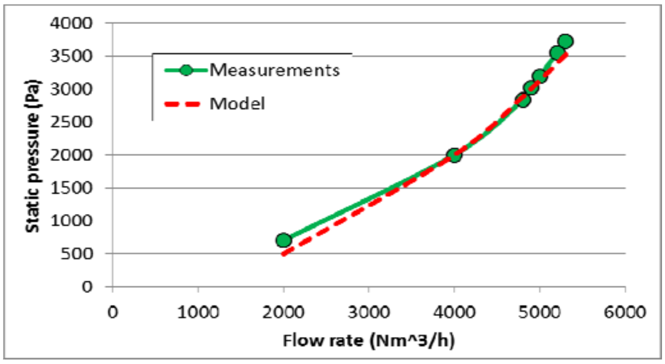

Figure 7. Pressure at the entrance of the FCN as a function of the flow rate.

4. Case Study

Let us now consider the design of a hypothetical FCN to see how these tools can be applied. In practice, a complete design requires several iterations between the different models until the best solution is identified. In this example, however, we will simplify the approach by only looking at each component once, starting by the pipe network.

The FCN must meet several distinct goals: it must cool down the cell efficiently, which requires either a large flow rate or a large nozzle exit velocity (or a combination of both); it must have relatively low energy requirements; and it must distribute its air as uniformly as possible so that positions further downstream experience the same cooling as those closer to the entrance. Obviously some of these constraints are in tension with one another, and an optimal tradeoff must therefore be found.

For simplicity, let's assume that there will be four nozzles per inter-cradle, and that the diameter of the main pipe surrounding the pot is fixed by geometrical considerations. In that case, there are only two main design parameters to consider at this stage: the nozzle diameter and the total flow rate. We can then use our pipe network calculation tool to see which configurations seem most promising.

This is shown in Figure 8. Each chart shows some parameter as a function of the total volumetric flow rate and nozzle diameter. The leftmost chart illustrates the variation between the minimum and maximum nozzle flow rate. If we specify that we can only accept a maximum variation of 10%, then everything above the red line must be rejected. The middle chart shows the energy requirements of the system, expressed in kWh per ton of aluminum. For this example, we can fix the maximum acceptable value at 15 kWh/t Al. Finally, the rightmost chart shows the combination of both criteria, with the green region corresponding to the acceptable design space. Obviously this will need to be recalculated with the specific performance of the fan and with the final dimensions, but for now it specifies which combination of parameters can be considered in more details in the CFD model.

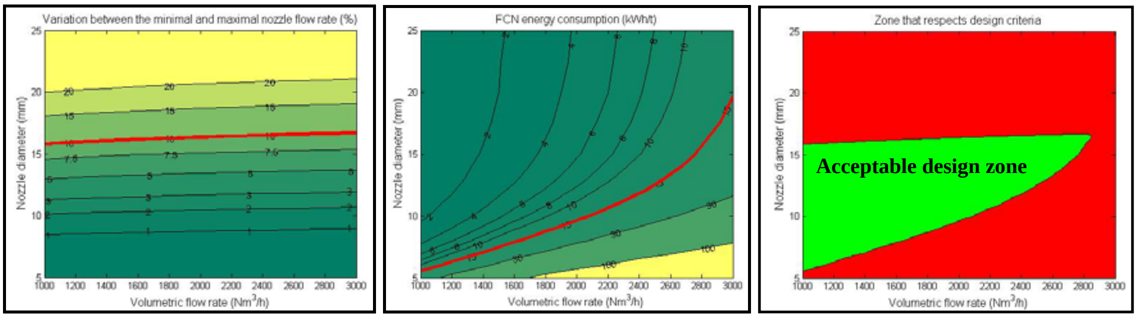

Figure 8. FCN performance as a function of flow rate and nozzle diameter: Variation between the minimum and maximum flow rate (left), energy consumption (middle), and acceptable design zone (right).

The next step is to find the exact configuration of the nozzle by using the CFD model. Parameters to study include the nozzle diameter, the distance from the shell, the angle, and the position of the impingement point. As an example, let’s consider the nozzle diameter. Using the previous results, we can try to determine if, for a given energy consumption by the fan, it is preferable to have a small nozzle (and thus a higher velocity but lower total flow rate) or a bigger one (and thus a lower velocity but higher total flow rate).

Results are shown in Figure 9. Although the sensitivity is limited, the best performance is obtained with a diameter of approximately 10 mm. For this particular case, it should be noted that the pressure drop in the main pipe becomes important at higher flow rates. The optimal diameter would be different if we had the space to accommodate a larger main pipe.

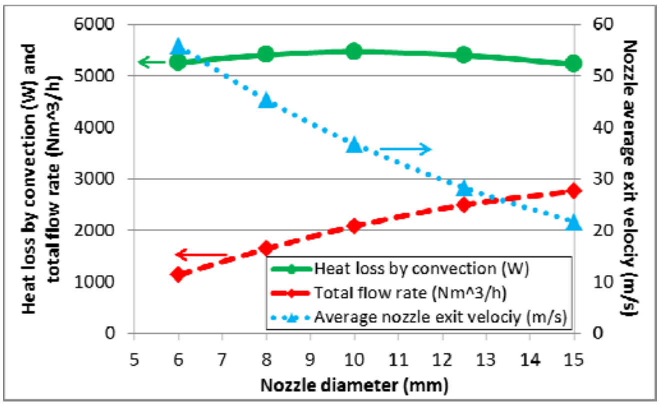

Figure 9. Performance of the FCN at different nozzle diameters, with the total flow rate adjusted to have the same energy requirements.

We can also consider the distance between the nozzle and the shell. Figure 10 shows the heat transfer coefficients and shell temperature at different distances. A closer nozzle creates a more localized cooling, leading to lower shell temperatures around the point of impact at the cost of higher temperatures further away. The optimal distance can be found by looking at the total heat dissipation.

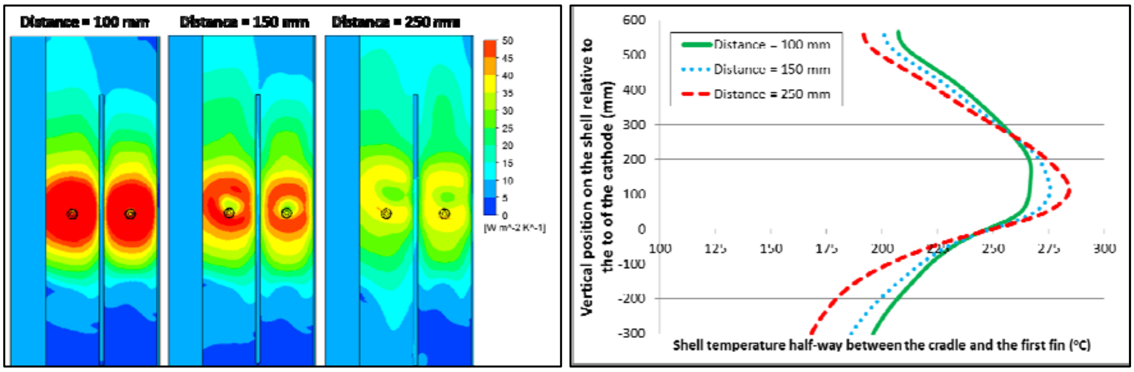

Figure 10. Heat transfer coefficients (left) and shell temperature (right) for different distances between the nozzle and the shell.

Finally, once all the parameters have been fixed, we can evaluate the impact on the pot by transferring the heat transfer coefficient to the cell thermo-electric model. Figure 11 presents the operating window – the region in which the cell can operate while maintaining an adequate thermal state – with and without a FCN.

There are three important points to note from this figure. First, the FCN allows the pot to increase its intensity by10 kA. Second, the range of available intensities at a given anode-cathode distance (ACD) also increases, going from 13 kA to 16 kA. This happens because the impact of the FCN is not constant; it extracts more heat when the pot is hot and has a higher shell temperature. Finally, in practice the air flow rate can be modulated as required, leading to a partial or complete transition between the two operating windows. In effect, then, the available operating region at a fixed ACD now covers 25 kA, instead of 13 kA initially. The ultimate result is that a well-designed FCN not only leads to operation at higher intensities, but also improves cell robustness, allowing it to operate under a wider range of parameters.

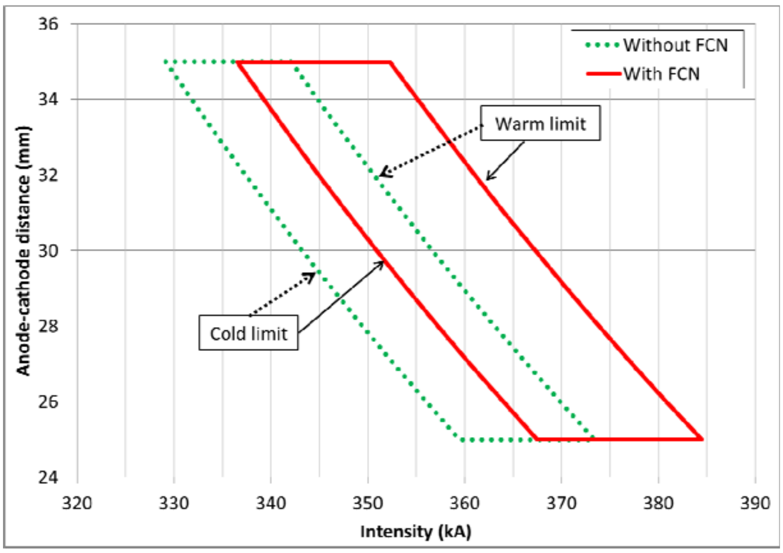

Figure 11. Operating window with and without FCN.

5. Further Developments

The tools presented thus far capture most of the physical phenomena relevant to the FCN. Still, there remains one important aspect that has not been considered. In an operating cell, the magnetically-induced metal flow leads to significant variation in the ledge thickness and shell temperature around the pot. Some zones are systematically warmer than others, and will be affected differently by the FCN.

To take this into account, the next step would therefore be to couple the previous calculations with our magneto-hydro-dynamic (MHD) model of the entire cell [6]. Such an approach would predict the FCN impact at every position around the pot, and might eventually lead to more sophisticated designs that aim to partially compensate for the disparity caused by the MHD forces.

6. Conclusion

A FCN is a technological option that can be used to increase cell intensity, robustness and operating range. To assist in its design, we have developed a combination of modelling and calculations tools, each focusing on a specific part of the system. The results of the models have been compared with experimental measurements, and show good agreement. We are now using this approach to optimize existing FCNs and design future ones.

7. References

1. Jerome Bos, Benoit Feve, and Pierre Homsi, Electrolytic pot for production of aluminum using the Hall-Héroult process comprising cooling means, U.S. Patent 6,251,237, 2001.
2. K. Jambunathan et al., A review of heat transfer data for single circular jet impingement, *International Journal of Heat and Fluid Flow*, Vol. 13, No. 2, (1992), 106-115.
3. Holger Martin, Heat and mass transfer between impinging gas jets and solid surfaces, *Advances in Heat Transfer*, Vol. 13, (1977), 1-60.
4. Robert J. Wallace et al., Efficient operation of compressed air jets for sidewall cooling, *Light Metals* 2007, 445-450.
5. Abdulla Habib, K.S.R. Raghavendra, and Barry Welch, Improving heat dissipation and cell life of aged reduction lines at Aluminium Bahrain (ALBA), *Light Metals* 2010, 285-290.
6. Steve Langlois et al, 3D coupled MHD and thermo-electrical modelling applied to AP technology pots, *Light Metals* 2015, 771-775.