Light Metals 2011 *Edited by: Stephen J. Lindsay*
TMS (The Minerals, Metals & Materials Society), 2011

Modern Design of Potroom Ventilation

Anastasiya Vershenya[1], Umesh Shah[1], Stephan Broek[1], Tom Plikas[1], Jennifer Woloshyn[1] and Andre Felipe Schneider[2]

[1]Hatch, Sheridan Science & Technology Park, 2800 Speakman Drive,
Mississauga, Ontario, Canada, L5K 2R7
[2]Hatch, 5 Place Ville-Marie, Suite 200, Montréal, Québec, Canada, H3B 2G2

Keywords: Potroom Ventilation, Cell Emissions, Heat Release Rates, Computational Fluid Dynamics (CFD)

Abstract

A typical natural ventilation design of a potroom incorporates incoming ambient air flow dividing into two main streams. The 'under the pot' stream passes through basement infrastructure and sweeps the heat/emissions from the pots upward to roof. The 'over the pot' stream flows through the Claustra wall and pushes the emissions/heat away from the working zone. This paper describes the use of CFD modeling to accurately predict the ventilation air flow split between these two main streams and improve the ventilation design, thus meeting the defined workplace hygiene standards.

The model's success greatly depends on reliable input conditions such as cell emissions and heat release rates. An innovative approach for the calculation of cell emission rates is thus introduced. Furthermore, the ventilation model can be coupled with pot heat balance and busbars thermoelectric analyses in order to obtain proper Joule heat generation and release rates.

Introduction

During development of a smelter, potroom ventilation design needs to address both heat and fugitive emission issues in order to comply with workplace standards. Computational fluid dynamics (CFD) is a commonly used technique which aids in design and further evaluation of the ventilation system performance as discussed in [1] and [2]. In addition, to correctly incorporate the cell emissions and heat release rates inside the potroom, CFD modeling needs to account for ambient conditions outside of the potroom that can effect ventilation such as local terrain, wind speed and direction. Often two separate CFD models are built in order to properly predict wind velocity, temperature and pressure outside of the building with one way coupling to the building's detailed interior model. Due to complex interactions between multiple boundary conditions, it is critical to use correct inputs for both models. To ensure success, different disciplines have to come together to outline the various boundary conditions and use CFD analysis to comprehensively design the ventilation system.

External and Internal CFD Models

Purpose of the external model is to establish wind flow patterns in close proximity to the building based on ambient temperature, wind velocity and direction. The geometry of an external model should include the main buildings on plant site and the surrounding terrain. Results of the external CFD model will provide pressure and velocity profiles for the two streams identified in the potroom slice model: 'under the pot' and 'over the pot'. An improper external model set up could result in an incorrect representation of the pressure differentials between the air inlets and the roof vent, which would lead to erroneous ventilation flow rate predictions. Since neither ambient temperature nor wind characteristics can be assumed constant over long periods of time, it is necessary to identify specific outdoor design conditions for the potroom ventilation. Based on weather data, a set of prevailing wind conditions has to be selected, thus affecting the flow inside the potroom in the most adverse way. It is up to the CFD modeling expert to identify those scenarios and use known meteorological data to define the ambient conditions for those simulation runs.

The second CFD model represents the inside of the potroom, including detailed geometry of the pot infrastructure, to accurately capture flow resistance. In addition to applying boundary conditions based on external model predictions, the internal potroom model needs to include pot cell emissions and heat release rates that reflect the total calculated heat load per pot. Since the internal model is used to determine whether working zone conditions are meeting the workplace hygiene standards, Hydrogen Fluoride (HF) emissions need to be captured as well. Both heat releases and HF emission are specified as boundary conditions for surfaces within pot geometry. Separate pot heat balance and busbar thermoelectric analyses can be used to determine the correct heat generation rates.

Cell Emissions

Cell emissions are an important input parameter for any ventilation model, which traditionally are taken care of by prescribing arbitrary cell emission rates at specific locations. In order to obtain a set of boundary conditions that holds a better connection to actual potroom practice, an in-house computer model for the prediction of emissions profiles was developed. Such a model is based on:

- The main production parameters;
- The gases produced from the electrolysis process;
- The configuration of the cell including covers and possible gaps;
- The duration of each step in the production process (tapping, anode change, sampling, etc);
- Normal draft and/or high draft ventilation;
- Anode effects;
- Secondary emissions from spent anodes and bath material removed from the cell.

All of these lead to a set of emission values that are applied at specific points on a cell where it is most likely that the emissions occur (see also Figure 9).

Cell and Pot-to-pot Busbars Heat Releases

The incoming fresh air that enters the building cools down the cells and busbars and, in turn, has its own temperature risen by the released thermal energy. One possible way to take these heat sources into account is by estimating the heat generation through measurements and well-known analytical and empirical relationships:

- The energy dissipated by the cell can be estimated by a voltage breakdown. The amount of heat released by each cell component (collector bars, pot shell's side walls and end walls, pot covers and gas collecting ducts) can be obtained through a thermal blitz campaign;
- The internal heat generation on each busbar can be estimated through the Joule Heat Equation. Current distribution and busbars equivalent resistances can be obtained by means of voltage drop and temperature measurements.

Coupling with Cell Heat Balance and Pot-to-pot Busbars Thermo-Electrical Models

On the other hand, the CFD ventilation model can be further coupled with external cell heat balance and pot-to-pot busbars thermo-electrical (TE) analyses. Two kinds of coupling can be considered:

- One-Way Coupling: both the cell heat balance and the busbars TE analyses are performed considering loads and boundary conditions obtained independently from the CFD model. Their output heat release rates are then used as fixed heat sources for the ventilation analysis. This is generally what has been done;
- Two-Way Coupling: while the cell heat balance and the busbars TE analyses provide the heat sources for the ventilation model, they would in turn have their convective boundary conditions defined by the air velocities and temperatures obtained from the CFD code. This is an iterative procedure that repeats itself until convergence is achieved. This would be an improvement on the approach so far.

Case Studies

Over the years, CFD modeling has become an accepted practice which is used to improve control of contaminants, such as HF and SO_2 gases, and provide a method of controlling cooling air flow in potrooms. Through extensive experience in the aluminum industry, Hatch has developed a methodology that ensures proper modeling procedures to be used for analyzing potroom ventilation systems. To share the expertise on how to use CFD analysis properly, a list of case studies supporting the approach is presented.

Environment Canada – CFD Modeling of Potroom Ventilation (2010)

The objective of the Environment Canada study was to investigate the impact of wind speed and direction on the flow exiting the roof vents in an aluminum smelter. Specifically, the study was focused on wind effects on the velocity reported at the roof ventilator, which is used in the calculation of declared emissions by the industry. The plant discussed in this case study is Aluminerie Alouette located in Sept-Îles, Québec.

Firstly, the external model of the plant and surrounding terrain was used to simulate the flow patterns around the buildings for a series of particular weather conditions reported on site. External model geometry with applied boundary conditions is shown in Figure 1. The wind is treated as a turbulent, isothermal and incompressible flow with a velocity profile at the inlet specified to match the expected shape for an atmospheric boundary layer. Properties are assigned for air at atmospheric pressure, measured local ambient temperature and humidity. Upon convergence of the external model, the wind patterns around the potroom are determined for each scenario. The information extracted from the results of the external model simulation includes the velocity inlet and pressure outlet profiles that are used as boundary conditions for the internal model.

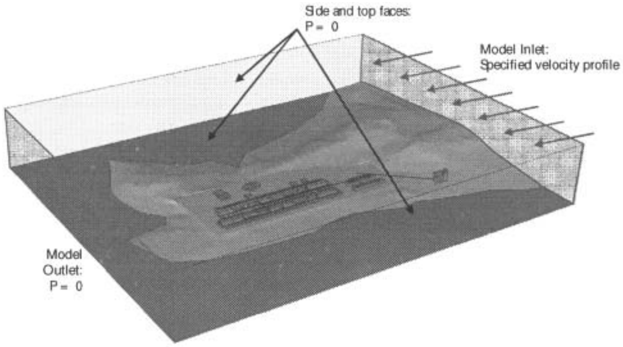

Figure 1. Boundary conditions for the external model geometry with the domain configured for a parallel wind.

The actual weather condition instances used to represent prevailing wind directions of interest were selected based on an extensive analysis correlating meteorological data from the onsite weather station (Pointe-Noire) to reported anemometer measurements. It was found that the maximum and minimum anemometer readings for a given period correlated well with certain wind directions.

Based on predictions obtained from the external model, an internal model of a slice of the potroom was used to simulate ventilation conditions corresponding to the identified scenarios. Since the velocity and pressure from the external model vary along the length of the potroom, the location of the profiles for application to the slice model is selected to simulate the worst case conditions. Generally, adverse wind conditions correspond to pressure at the roof vent being greater than pressure at the air inlet or pressure on one side of the potroom being significantly greater than on the other side, as can be seen in Figure 2.

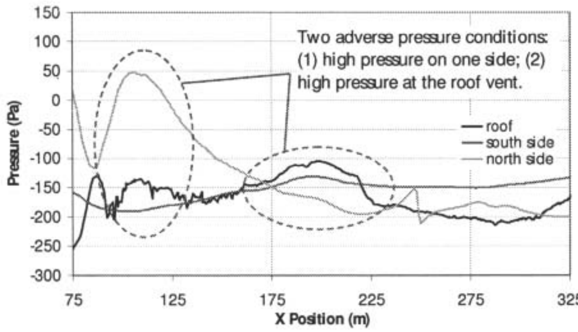

Figure 2. Predicted pressure profile along the north and south basement panels and the roof monitor of the potroom due to wind.

A more rigorous approach to the internal model setup, than described above, is implemented in this case in order to investigate the influence of wind conditions on the ventilation system performance. The internal model includes detailed geometry of the potroom as well as an additional domain outside of the building. Profiles obtained from the external model were applied to the domain of the internal model, thus capturing air velocity and pressure directly outside of the air ingresses and outlets. This approach more accurately predicts the division of 'over the pot' and 'under the pot' air flows. It also allows the model to capture re-entrainment of flow exiting the roof at the basement inlets when it occurs.

Based on whether wind direction of interest was parallel or perpendicular to the potrooms, the internal model setup had two different configurations. Figure 3 displays the model configuration when wind direction is perpendicular to the potroom. A specified wind velocity is applied on one side boundary with a pressure profile from the external model on the opposite side face, while the top face is set to atmospheric pressure to allow air to exit or enter the domain, as required. Due to the repetitive structure of the long potroom buildings, a slice model encompassing the two halves of the neighboring reduction cells with symmetry boundaries on the front and back faces of the potroom has been used. Plant anemometer data measuring the velocity at the roof vent has been used to validate the CFD models.

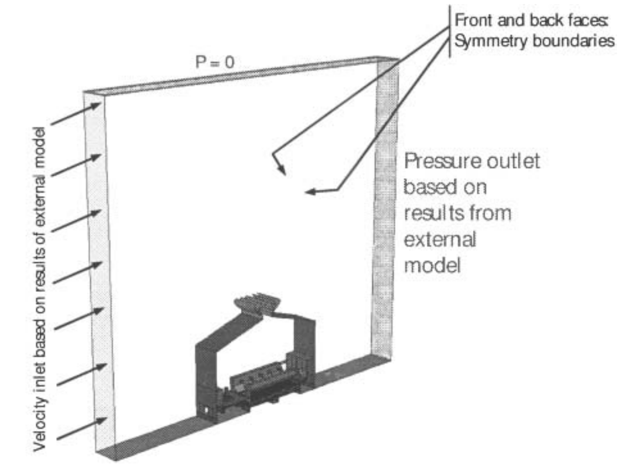

Figure 3. Boundary conditions for the internal model geometry with the domain configured for a wind approaching normal to the potroom.

In case of the wind direction being parallel to the potrooms, internal model has been setup as shown in Figure 4. A specified wind velocity is applied to the front face with a pressure profile applied to the back face, while the top and both side faces external to the potroom are set to the atmospheric pressure to allow air to exit or enter the domain, as required. For the front and back faces inside the potroom, periodic boundary conditions are used for the normal wind direction. Both of these configurations assume uniform internal conditions along the length of the potroom and neglect end effects, thus being representative of the potroom's central portion.

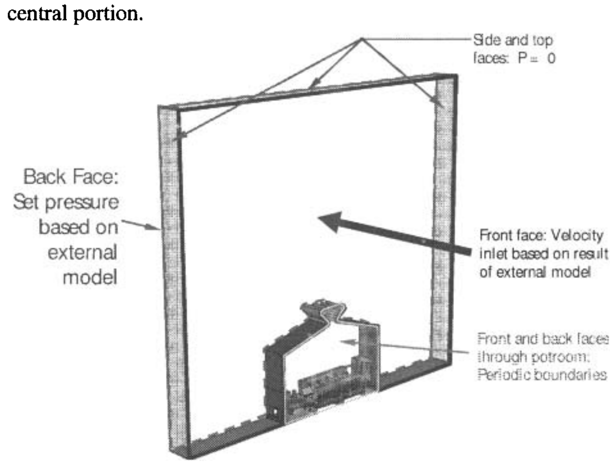

Figure 4. Boundary conditions for the internal model geometry with the domain configured for a wind approaching parallel to the potroom.

The flow inside the potroom is driven by the heat released from the pots. Therefore, the model must account for the energy transport. The flow is treated as a turbulent, non-isothermal, incompressible ideal gas. Hydrogen fluoride (HF) emissions are specified at the pot as a separate species with properties of HF. The air and HF species are free to mix and disperse together in the model.

As shown in Figure 5, emissions of HF were specified from the pot cover and anode butt surfaces. Heat release surfaces were the bottom and sidewalls of the pots as well as the superstructure with the total heat losses calculated using thermo-electric analysis.

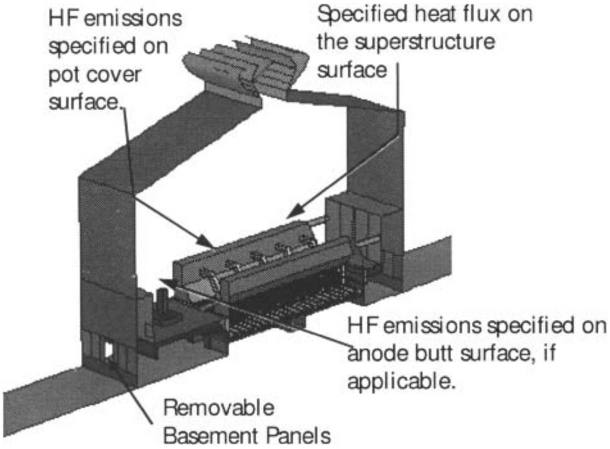

Figure 5. Boundary conditions on internal surfaces of the potroom slice model.

Additional sub-models have been run to calculate the loss coefficient for the basement grating, opening in the gas side and tapping side Claustra wall. They were treated as porous jumps in the potroom slice model, as the flow passes through these surfaces with specified loss coefficients.

Model predictions for each scenario have been validated using the anemometer data collected at the roof vents. Summary of the results for all cases is presented in Figure 6. Although the CFD model generally under-predicted the roof velocity, as compared to the anemometer data, it was able to successfully capture the trend in changes of the roof velocity due to variations in wind speed and direction.

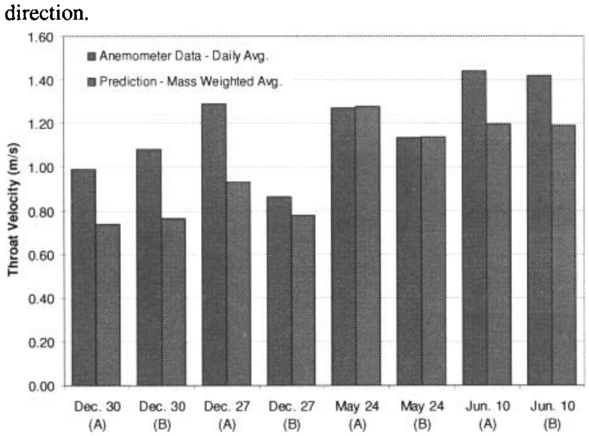

Figure 6. Comparison between predicted roof vent throat velocity and the recorded plant anemometer data.

The simulation results showed that, using the outlined methodology with reliable boundary conditions, CFD models are capable of capturing the effects of wind direction of the ventilation flow and gas velocity at the roof vents.

Alcoa – CFD Modeling of the Proposed Potline Ventilation Design (2010)

The purpose of the study commissioned by Alcoa was to determine if a new potline, given its close proximity to the neighbouring casthouse and maintenance buildings, would cause a fatal flaw with respect to meeting the pot and bus cooling, and industrial hygiene criteria. Alcoa smelter discussed in this study is located in Baie Comeau, Québec.

An external CFD model of the smelter was first run to determine the ambient air pressures and temperatures adjacent to the new potline. It provided pressure and temperature values at the potroom air intakes and outlet boundaries. An internal CFD model of a section of the potroom was run using the results from the external model simulation. Other boundary conditions were set based on known process conditions (pot heat release, emission rates, etc). The potroom slice model predicted the ventilation flow rate through the potroom, the heat stress and emissions level in the worker area, pot and busbar temperatures.

Based on the climatological data for Baie Comeau from 2003 to 2009 and corresponding wind rose for each season, several wind conditions were selected to use as inputs to the external model. The simulation results were analyzed to identify the pressure/temperature conditions at the potroom air intakes (basement panels) and outlet (roof monitor) that could produce the worst ventilation performance. The selected pressure/temperature conditions were applied at the two air ingresses and a roof ventilation outlet as shown in Figure 7. The internal model included a slice of the potroom from the basement panels up to the roof gravity ventilator.

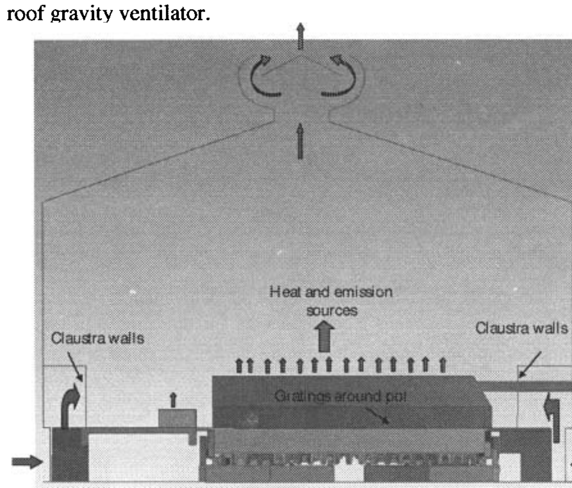

Figure 7. Boundary conditions for the internal model geometry.

Due to the small scale of the gratings, the internal CFD model did not include their actual geometry; however, in order to maintain the accuracy of the results, detailed pot geometry reflected the exact open grating area. The pressure drop caused by the gratings was accounted through the use of an appropriate loss coefficient value. A separate CFD sub-model, which included complete geometry of the gratings around the pot, was created to determine the loss coefficient. Similarly, a loss coefficient was calculated for the Claustra walls and implemented in the potroom slice model.

The air flow within the potroom was treated as an ideal, incompressible, non-isothermal, turbulent gas with natural ventilation occurring due to thermal buoyancy. For the winter operation of the potroom, all 6 basement panels, shown in Figure 8, were kept closed. For the summer operating scenario, all 6 basement panels were open, thus the corresponding walls were removed from the model setup. It is important to establish the exact area of air inlets as the reduction in ventilation air flow in winter has a significant effect on the amount of heat removed from the pot surfaces and results in an increase of the concentration of HF, SO_2 and dust in the work zone.

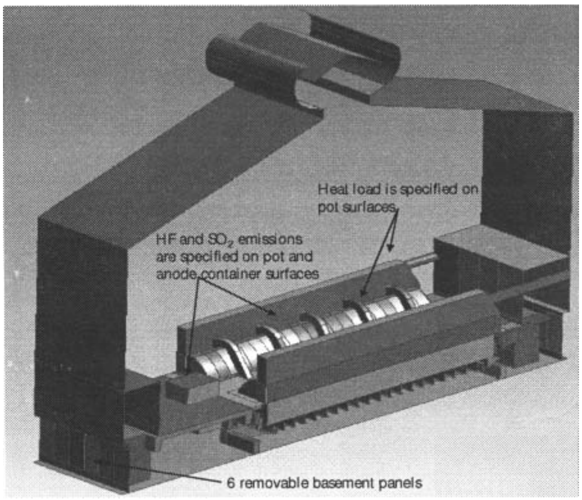

Figure 8. Internal potroom geometry for summer and winter configurations.

Sources of heat loss present in the model are the busbar, superstructure, side and bottom of the pot. The mass flow rate of HF, SO_2 and dust particulates emitted into the potroom were specified over surfaces within the internal CFD model. Emission points, as displayed in Figure 9, include: (1) the end gap between the cover and side wall; (2) collar around the anode stems; (3) the gap above the doors; (4) the openings around the anode stems in the pen butt container. The distribution of HF and SO_2 were calculated as a result of ventilation air flows within the potroom.

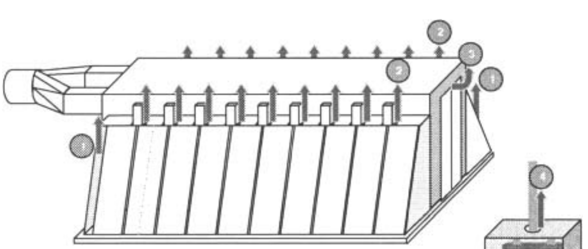

Figure 9. Location of modeled emission points on the pot structure.

A total of 6 cases were modeled to encompass a variety of wind conditions in both summer and winter seasons. Each case predicted air flow throughout the potroom as well as concentration of all emissions at various heights above the operating floor. CFD modeling results were analyzed to determine whether potroom conditions for each case met the criteria for the heat stress and contaminant threshold limits. Predicted values of wet bulb globe temperature (WBGT) in the working zone have been compared against set heat stress levels to ensure that both temperature and humidity levels were acceptable for the smelter personnel. The working zone was identified as the area up to 2 m above the operating floor between the pots and in the traffic aisle in front of the pot for tapping and pedestrian movement. Additionally, HF, SO_2 and dust concentrations throughout the entire potroom have been evaluated to guarantee that the predicted range does not exceed specified threshold levels.

Completed CFD modeling work provided a comprehensive analysis of the proposed ventilation system for multiple operating scenarios. Various combinations of wind direction, speed and ambient temperature have been used to assess performance of the ventilation system and ensure that the efficient pot and bus cooling is observed while meeting industrial hygiene criteria.

Conclusions

CFD is frequently used to assess current operational conditions at aluminum smelters and furthermore predicts the effects of the proposed geometrical or process changes. Due to the complex interaction between known boundary conditions and performance of the ventilation system inside the potroom, it is critical to establish the correct methodology for CFD analysis. The presented case studies outline the factors that need to be taken into consideration in order to ensure proper modeling predictions are obtained. The approach has been validated using measured site data, which allows for the potroom models to be used with confidence to predict air flow in the potroom, while taking into account outdoor ambient conditions and plant terrain unique for each smelter.

Acknowledgements

The authors would like to acknowledge the support and input from the staff of Environment Canada, Aluminerie Alouette Inc. and Alcoa. Daniel Richard is gratefully acknowledged for the valuable inputs and comments.

References

[1] A. Van Maarschalkerwaard, "The Use of CFD Simulations to Optimize Ventilation of Potrooms", Light Metals 2010.

[2] J. Berkoe, et al, "CFD Modeling of the Fjardaál Smelter Potroom Ventilation", Light Metals 2005, 373-378.

[3] J. Woloshyn, "CFD Modeling of Potroom Roof Ventilation - Impact of Wind", Hatch report 334517-RPT-CA01-10002 for Environment Canada, October 2010.