Light Metals 2013 *Edited by: Barry Sadler*
TMS (The Minerals, Metals & Materials Society), 2013

Latest developments in potroom building ventilation CFD modelling

Nathalie Menet[1], Guillaume Girault[1], Nicolas Monnet[2], Catherine Turpin[3], Lionel Soulhac[4]
[1]Rio Tinto Alcan L.R.F., BP114, 73303 Saint Jean de Maurienne Cedex, France
[2]Rio Tinto Alcan Smelter Technology, Centr'Alp, BP 7, 38341 Voreppe Cedex, France
[3]Sillages Environnement, 64, chemin des Mouilles, 69134 Ecully Cedex, France
[4]Laboratoire de Mécanique des Fluides et d'Acoustique, Université de Lyon, CNRS, Ecole Centrale de Lyon, INSA Lyon, Université Claude Bernard Lyon I, Ecully, France

Keywords: CFD modelling, potroom building ventilation, cell design, heat and pollutant exposure, roof vent emissions monitoring

Abstract

In a context of production increases and proactive management of future health regulations, optimised workshop ventilation is necessary. Accurate predictions are required in order to achieve cost-efficient ventilation design including all specific features of the building, and this involves developing complex CFD models. The complexity of the models must, however, be well adjusted. They have to fulfil requirements as different as, for example, providing precise simulation of the impact of ventilation on cell cooling or determination of the positions for roof-vent monitoring equipment. A new approach is now considered, based on the development of several CFD tools of varying sophistication. Depending on project requirements, a combination of tools with the most appropriate level of complexity can be selected so that the trade-off between accuracy and computation time is optimised. This approach, validated through comparison with measurement campaigns, can now be used for cell development and new projects, regardless of building design and local conditions.

Introduction

It is becoming more and more critical to ensure efficient ventilation of potroom buildings in order to guarantee healthy working conditions. Greenfield smelters tend to be located in hot countries where ambient conditions are extremely detrimental to conditions inside the potrooms. This is made worse by increasing line amperage and therefore heat dissipation from the pots, which in turn increases the temperatures to which operators are exposed. In parallel, tighter regulations with respect to the exposure of workers to chemicals (mainly HF, dust and SO_2) are anticipated in the medium term.

But while the main concern when considering building ventilation is workers' health and comfort, new constraints are now emerging and must be taken into consideration. Examples include the impact of ventilation on cell heat dissipation and therefore performance, or on the reliability of critical equipment such as pot tending assemblies, which can be negatively affected by high temperature levels. Also, in order to identify the best positions for pollutant monitoring devices in potrooms, it is frequently necessary to determine the most representative locations according to the type of ventilation.

All these constraints have to be taken into account at a time when a lot of effort is being made to reduce the overall capital investment required both for new greenfield plants and amperage creep projects. The cost associated with passive ventilation equipment (claustra walls, floor gratings and roof vents) is significant – typically up to 6% of the potroom building cost.

In this context, tools are required to accurately predict the impact of proposed design changes on ventilation efficiency. The development of such tools has been a fairly recent subject of interest [1, 2, 3 ,4, 5]. These tools must, on one hand, be sufficiently complex to take full account of parameters that can significantly impact their prediction, and on the other hand they must also be capable of providing quick answers for the engineers in charge of the building design in a project environment. Lastly, they must also be versatile enough to be used to answer a wide range of questions, as discussed in the previous paragraphs.

This paper discusses a new approach based on Computational Fluid Dynamic (CFD) modelling to attempt to reconcile these apparently conflicting objectives.

The different modelling approaches

When building a new CFD simulation, the challenge is always to find the right balance between its level of sophistication and the time required to build and run it. The aim is therefore to design models of just sufficient complexity to address the questions at hand. J. Bos et al. [1] suggested some guidelines to help achieve this objective, which can be applied to our problem.

The first step consists in clearly defining what exactly is expected of the model, and with what corresponding accuracy. With this in mind, the following main questions will then have to be answered in order to characterize the model's level of complexity.

- What is the optimum scale of the model? A slice of a potroom, a full potroom or the complete site?
- Which physical phenomena should be included? Should meteorological conditions be taken into consideration?
- What is the optimum type and size of the model grid?
- With what level of detail should the main physical elements be represented (pot, claustra walls, roof vents, etc)?

Lastly, whenever possible, the model should be compared with existing or purposely collected data to validate its prediction capability.

In practice, to answer these questions, preliminary sensitivity tests are conducted prior to the development of the model itself in order to optimize these different parameters. This logic will be illustrated on the basis of the development of three different types of model, which correspond to three different levels of scale (complete site / intermediate / potroom slice), with a specific focus on the general 3D model.

It will then be seen how these three types can be combined to address numerous technical questions and two applications of these models will then be presented as illustrations of their potential uses.

General technical considerations

The following developments were performed using CFD modelling based on the commercial software program FLUENT 14.1, which uses a finite volume method to solve the three-dimensional Reynolds Averaged Navier-Stokes equations. The turbulence closure model was achieved through application of the two-equations k-ε model. Natural convection was modelled with the Boussinesq model (density function of temperature) and a body-force weighted pressure correction scheme was used in the calculations. The species transport model is also used. A logarithmic velocity profile, representative of an atmospheric boundary layer, is imposed at the domain frontier when outdoor flow is simulated.

"Slice" modelling approach

The first CFD model historically developed by Rio Tinto Alcan was a potroom slice representing one area between two pots [1]. Its main purpose was to evaluate ambient temperature in the working zone. This model, whose predictions have been validated by numerous measurement campaigns, has been improved and validated over the years to take advantage of improvements in hardware and software. Recent changes include for example the fact that real roof vent geometries can now be taken into consideration, as illustrated on **Figure 1 a**, whereas they used to be represented by a porous zone of equivalent pressure drop. Correct representation of the porous zone representing the claustra wall has been validated by comparison with a model based on the real wall geometry (**Figure 1 b).**

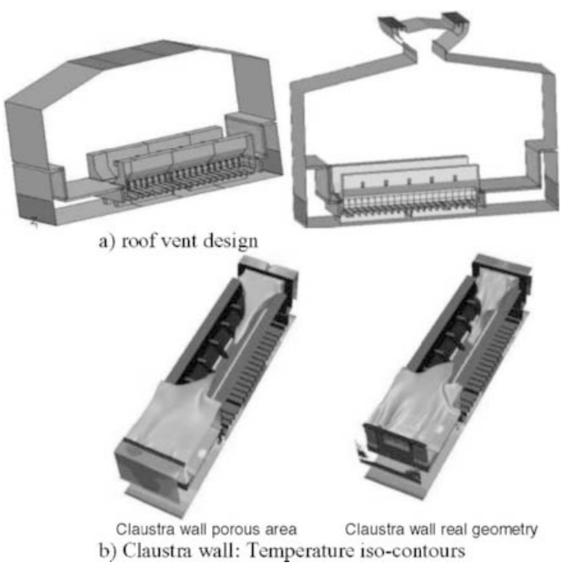

Figure 1 : Boundary conditions of the potroom slice model

This model provides highly targeted results concerning the ventilation and temperature between two pots, where operators will be standing. It is still useful in quickly evaluating the relative impact of proposed design changes (higher amperage, different roof vent, etc.). However, this model cannot take into account transversal flows such as those that would arise from wind blowing against the outside of building. In the case where such flows have to be represented, the model domain therefore needs to be extended.

General modelling approach

General models representing the entire industrial site including the potline as well as surrounding buildings and topography have been explored by others [1, 2, 4]. Such a model used as a pilot, representing an existing Rio Tinto Alcan smelter, was developed with the aim of estimating pollutant flux profiles leaving the potline roof vent depending on wind conditions. This could in turn be used to position environmental monitoring instruments more effectively, at more representative positions. Various tests were conducted to ensure the optimization and robustness of the model. Two such tests are presented as illustrations.

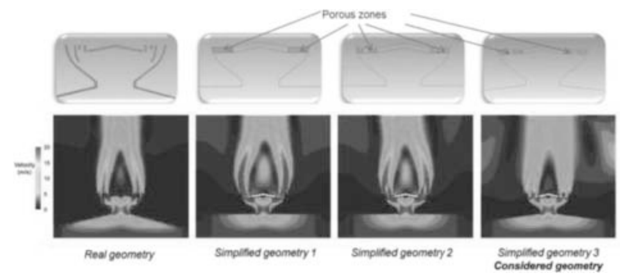

Figure 2 : Validation of roof vent geometry

The first test presented in this paper relates to the representation of the ventilation systems (roof vents, claustra walls and gratings). These devices are key elements in natural ventilation and must be represented as accurately as possible. In order to optimize the number of cells and the calculation time, different tests were conducted by simplifying the representation of the ventilation device more and more until the numerical results did not require further modification. The tests showed that, in a general configuration model, roof vent, claustra wall and floor grating geometries can be simplified using porous zones whose porosity is equivalent to that of the real system. **Figure 2** shows the results obtained during the vent geometry validation process. Comparison between flows through simplified roof vent geometries and reference flow obtained through the real geometry showed good agreement with the observations. **Figure 3** represents the validated model geometries for the different ventilation systems.

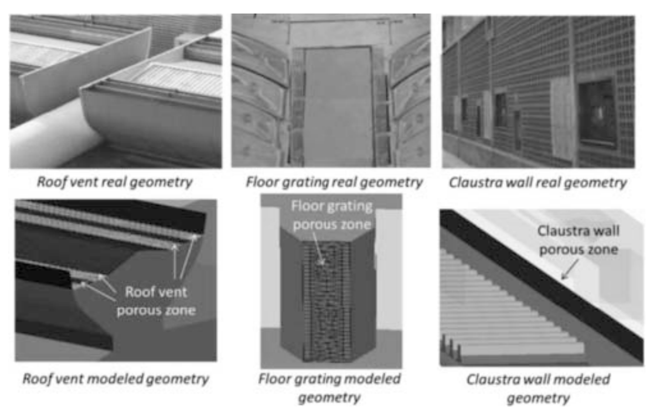

Figure 3: Real and modelled ventilation devices

Another series of tests dealt with the way the wind is taken into account through two possible options. In the "coupled approach", the outdoor and indoor airflows are modelled simultaneously within the same computational domain. Alternatively, the "decoupled approach" consists in modelling outdoor and indoor airflows separately. The outdoor flow field is simulated first to determine the airflow structure around the potroom buildings. Boundary conditions at the potroom surface are collected. These are later used as inlet conditions for the indoor airflow simulation.

These two approaches have been well discussed in several publications [6, 7]. It is generally accepted that the coupled approach leads to more accurate results but the geometry and meshing are complex and calculation time is long. On the other hand, the decoupled approach has some limitations. They include over-estimating the surface wind pressure [7], thereby omitting the inflow angle at the opening of the indoor domain and relying on a uniform inlet wind pressure. However the decoupled method allows two geometry scales with two different levels of meshing to be simulated, so that calculation time can be improved. This is particularly relevant for very large grids (>10 million cells).

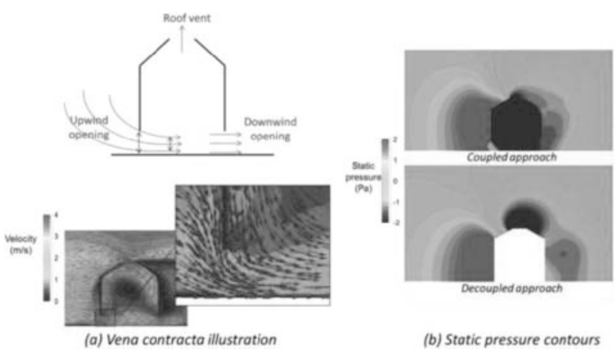

Figure 4 : Tests on a simplified potroom configuration

A sensitivity analysis was performed on a simplified potroom configuration in order to be able to choose between these two solutions. This revealed two main differences. With the coupled approach, a vena contracta type of behaviour, typical of flows through orifices, can be observed at the upwind opening. This phenomenon, illustrated on **Figure 4 (a)**, results in an additional pressure loss being generated. However, in the decoupled approach, the imposed pressure at the upwind opening is uniform and the vena contracta effect is therefore not taken into account. To compensate for this, a pressure coefficient needs to be introduced into the inlet condition of the indoor domain.

Secondly, differences in pressure contours around the potroom roof vent are observed between the two models tested, as shown in **Figure 4 (b)**. Further investigations demonstrated that imposing ejection velocity as roof vent boundary condition in the outdoor flow field simulation (in the decoupled approach) allowed produced pressure contours to be similar to those obtained with the coupled approach.

As expected, a better fit was obtained between the on-site measurements and the results from the coupled approach than those from the decoupled approach. The coupled approach is therefore the preferred method in this case. It should be noted that the critical points identified with the decoupled approach could theoretically be overcome by adjusting pressure loss coefficients at the domain openings and ejection velocity at the roof vent; however, these adjustments require a preliminary analysis to be conducted and additional computational time.

Figure 5 shows a comparison of the vertical velocity measured in the neck of the roof vent and those calculated by numerical simulations. The weather conditions (wind direction and velocity) applied for the simulation are representative of those during the measurement campaign. According to this figure, the numerical results fit well with the measured data.

This figure also highlights the significant variability of the vertical velocity along the roof vent. This variability is due to various factors which were studied independently. They include mainly the wind direction, the presence of surrounding buildings and the potroom doors.

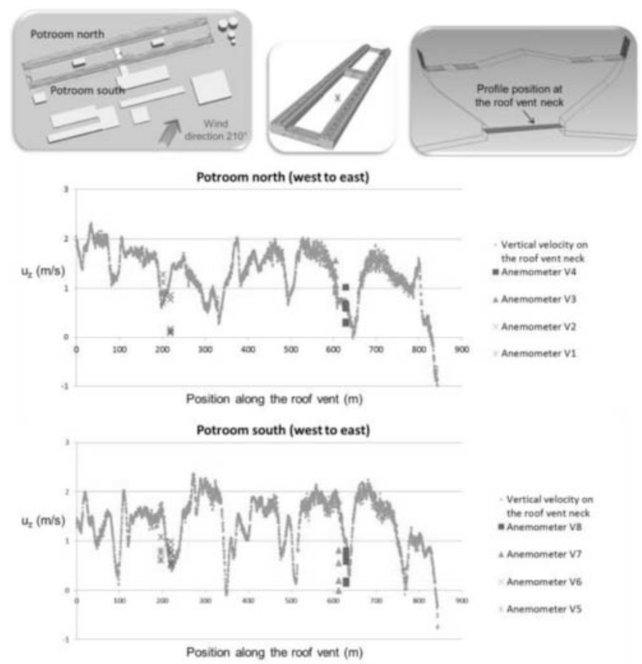

Figure 5 : Vertical velocity profile in the roof vent neck

Once all the optimization work described above was completed, it was concluded that this type of model is a good way of predicting general airflow patterns in and around buildings. Examples of use include the positioning of roof vent monitoring devices at representative positions. On the other hand, the weaknesses of these models include long development time. While they are good at providing large-scale representation of the physical phenomena at play, they do not provide a detailed representation of what is happening between pots.

<u>The intermediate model</u>

When considering the two extreme approaches discussed previously, it was thought that there was some opportunity for developing an intermediate methodology addressing the limitations of each.

This “intermediate” model, shown in **Figure 6,** represents a succession of slices (9 to 21 slices each representing one inter-pot) included in an outdoor domain. The model can be fully parameterized and takes into account external wind conditions and radiation of heat from cells. The central slice is defined with great accuracy, similar to the slice model; slices on both sides are progressively coarsened until accuracy similar to that of the general model is reached, meaning that the area of interest is limited to the centre. Two different roof vent configurations (low-profile or tulip) are considered.

The model offers the possibility of studying in particular the impact on ventilation by large doors for the passage of vehicles.

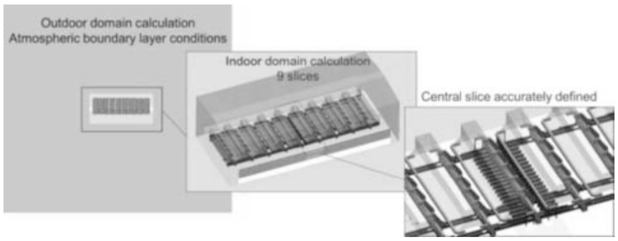

Figure 6 : Intermediate model geometry

The model was validated in two Rio Tinto Alcan plants in particular by observing flow paths using smoke devices and comparing them with the model results. The wind effect was taken into account for this comparison, as shown in **Figure 7**.

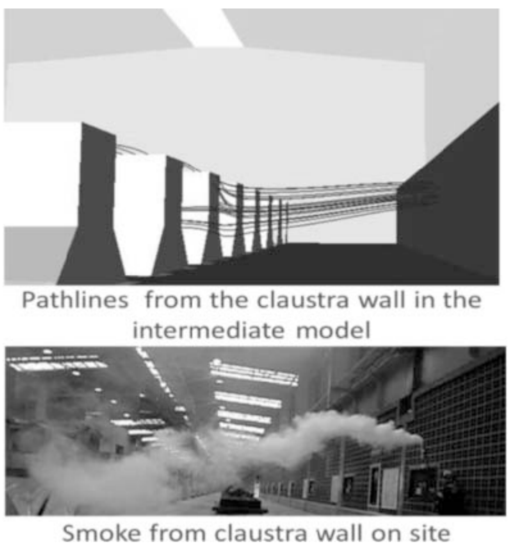

Figure 7 : Comparison between smoke test and modelling with the intermediate model

This intermediate model merges the major benefits from both the general and slice models. These two approaches were validated independently, in order to ensure the validity of the geometry, the grid and the parameterization of the central slice and outdoor domain.

This model is particularly useful when it comes to evaluating the absolute temperature levels operators will be exposed to, taking into account external conditions. An example will be discussed in the next section.

<u>Conclusion: the three-step approach</u>

The previous discussions illustrated the model development process **(Figure 8)**, that aims at optimizing the “accuracy to computation time” ratio.

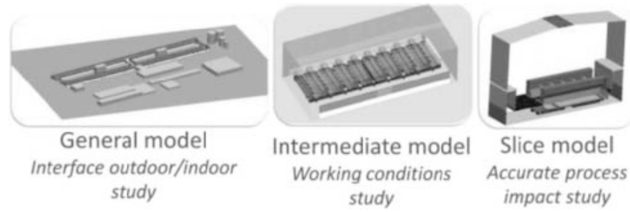

Figure 8 : Three-step approach

It is evident that there is no “one-size-fits-all” CFD model that is able to address all possible questions related to building ventilation. A CFD approach based on three levels of analysis is therefore proposed. This is summarised below in Table 1.

Depending of the question at hand, one of each model or a combination of them can be used. This approach will be illustrated by means of two practical cases that were recently studied.

	General Model	**Intermediate Model**	**Slice Model**
Pot design features	Considered	Considered	Considered
Species transport	Yes	Yes	Yes
Natural convection	Yes	Yes	Yes
Radiation	No	Yes	Yes
Weather conditions	Yes	Yes	No
Terrain topography	Yes	No	No
Potroom representation	Entire	Up to 21 inter-pots	1 inter-pot
Level of detail of the geometry	Simplified	Accurate	Very accurate
Mesh	Coarse	Fine	Very fine
Flow analysis type	General	Accurate	Targeted

Table 1 : Performance of each model

Practical examples of uses of the models

Impact of amperage creep projects on working conditions

A first example of use is the study carried out to support one Rio Tinto Alcan smelter amperage creep project. After conversion, working conditions, especially air temperature in working areas, have to be maintained or improved while heat emitted by the pots increases in the potrooms. This smelter (**Figure 9)** is subject to frequent high winds that have a strong impact on the natural ventilation of the potrooms, which therefore had to be taken into account. Each potroom comprises 160 end-to-end pots arranged in two lines and divided into 8 sections of 20 pots separated by passageways. Pot operations are performed in a typical layout with three tending aisles: a central aisle between the two lines of pots and two lateral aisles on each side of the building.

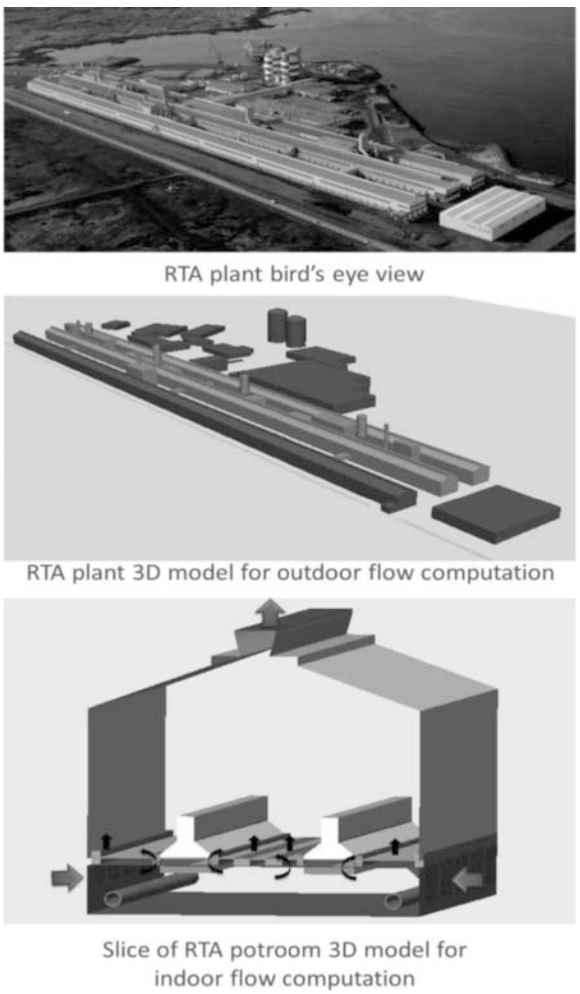

Figure 9 : Plant-wide & potroom 3D models

To limit calculation times, the decoupled approach was used. Compared to the pilot case presented above, this was made possible thanks to a different building configuration that is less sensitive to the vena contracta effect. Two models were subsequently developed:

- A plant-wide 3D model, on which were applied the two main prevailing wind conditions. Pressures on the building walls and roof vent were calculated through this model.
- A potroom 3D model, representing one section of 20 pots in detail and two half sections on each side less accurately.
- Pressures calculated through the plant model were applied to this potroom model as boundary conditions on louvers and roof vent.

A comprehensive measurement campaign took place in the potlines to validate this modelling approach. Comparison between measurements and calculations showed very similar figures. **Figure 10** represents an example of comparison between measured (Measurement curve) and calculated (Existing louvers curve) temperatures in a cross-section of potline 3. Several technical options to improve the building ventilation (corresponding to Solutions 1 to 3 on **Figure 10**) were investigated using this model, which allowed the optimum size and position of the louvers to be determined. This optimum solution, Solution 1, is now being implemented.

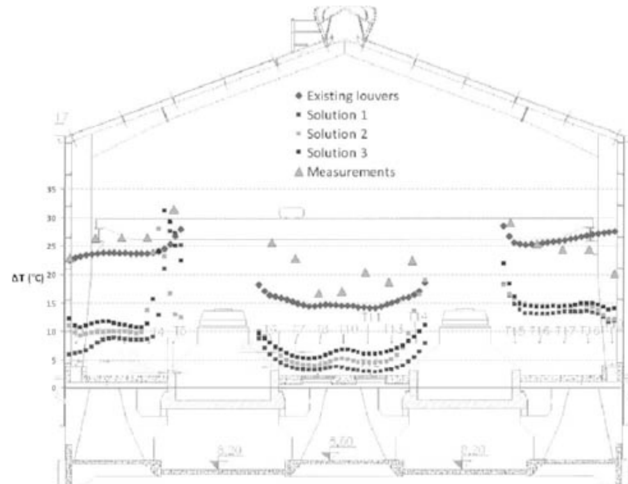

Figure 10 : Temperatures in potline 3 at 1.5m from working floor

Concentration of pollutants in covered roads

Another example of use is the study carried out to obtain the construction permit for the covered roads at another Rio Tinto Alcan smelter. Located in Canada, it has to face heavy snowfalls each winter season. The roads have to be covered in order to maintain continuous operations in the different workshops of the plant. To obtain a construction permit for these covered roads, the plant had to demonstrate that pollutant concentrations due to vehicles and their loads did not exceed regulatory exposure limits.

For this study, the intention was to use the coupled approach to develop only one general 3D model and save on run time. This was possible only if certain potroom details could be simplified enough to reduce the size of the model, i.e. the total number of meshes. The chosen option was to model pots as an equivalent continuous pot with gratings on each side. This simplified approach was validated by comparing the simplified model with a more complete one. The final model used is represented on **Figure 11.**

Once this simplification was validated, the resulting general 3D model could be run in a manageable time with a good level of detail for the others buildings. Air enters the building through louvers in pallets storage building and along the cover roads or through the basement of potrooms and leaves through potrooms and pallet storage building roof vents. Two pollutants (HF and SO_2) were considered. Sources of emissions include the pots, the anode butts from the pallet storage building and the vehicles.

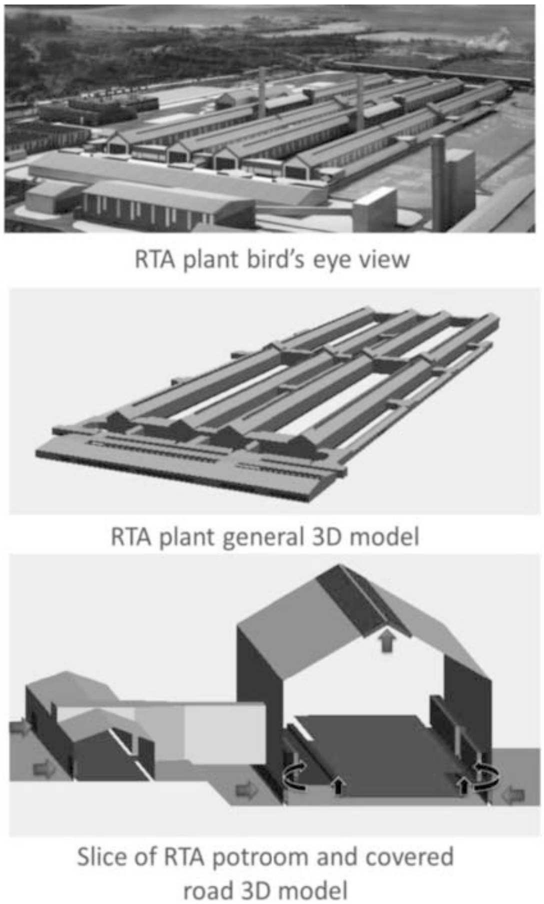

Figure 11 : Models used for the covered road study

Figure 12 illustrates the HF contribution from the anode pallet storage building in the neighbouring covered road for the prevailing wind conditions. It shows how HF will spread from the storage building and identifies the areas with the highest concentrations. With this approach, it is possible to differentiate the contribution from each source (storage, transport and pots) at any location. In this specific case study, both pollutant concentrations in the covered roads remained below exposure limits.

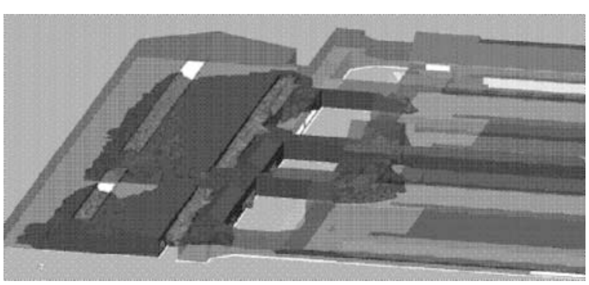

Figure 12: HF contribution from anode pallet storage building (isoconcentration 0.1ppm)

Conclusion

The modelling approach presented in this paper is based on three models of different complexity. Examples of use of each model were presented, illustrating their scope of application. Extensive sensitivity tests were conducted in order to optimize each model, whose predictions were confirmed by on-site measurements.

Combining these tools allows a large range of issues to be covered, including assessment of working conditions, qualification of passive ventilation devices, impact of potroom ventilation on the pot process, as well as the positioning of emission monitoring equipment. This list is not exhaustive and their use could be extended to many other environmental and engineering applications. Overall, they allow ventilation efficiency to be optimized for the lowest possible cost.

It is also worth mentioning that the direct link between these CFD models and the cell design models used to develop new or retrofitted cell technologies guarantees a robust general solution backed by extensive floor validation.

These tools are now operational and are used for both new greenfield and amperage creep projects. Efforts have been made to obtain as generic an approach as possible so that the time required for modelling is reduced, ensuring that project planning targets are met.

Acknowledgement

The authors wish to thank Rio Tinto Alcan smelters'staff for providing support for all on-site measurements.

References

[1] J. Bos et al., "Numerical simulation, tools to design and optimize smelting technology", Light Metals 1998, 393-401.

[2] E. Dernedde, "Ventilation rates of smelter buildings in northern climates", Light Metals 2004, 347-350.

[3] J.Berkoe, et al, "CFD Modeling of the Fjardaal smelter potroom ventilation", Light Metals 2005, 373-378.

[4] A.Vershenya, et al, "Modern Design of Potroom Ventilation", Light Metals 2011, 531-535.

[5] E. Manger, et al, "Pot Room Design and Ventilation in Desert Climates" Proc. 9th Australasian Smelting Technology Conference and Workshop, Nov. 5-9, Terrigal, Australia, 2007

[6] T. Kurabuchi, et al, "Domain decomposition technique applied for cross-ventilation of building" AWAS'08 2008, 1129-1140.

[7] T. Endo, et al, "Study on the numerical predictive accuracy of wind pressure distribution and air flow characteristics" International Conference "Passive and Low Energy Cooling for the Built Environment" 2005, 77-82.