John Floyd International Symposium on
SUSTAINABLE DEVELOPMENTS IN METALS PROCESSING
July 3 - 6, 2005, Melbourne, Australia.
Edited by M. Nilmani & W. J. Rankin — nilmani@vsnl.com

Reducing Fume Emission from Smelting Vessels Using Combined Cfd and Engineering Modelling

PJ Witt[1], CB Solnordal[1], A Manzoori[1], S Finn[2], J Pluta[2], H Namavari[3], E Niknejad[3] and M Davari[3]

[1]CSIRO Minerals, Box 312, Clayton, Victoria 3169, AUSTRALIA
[2]Zinifex Limited, Melbourne, Victoria 3004, AUSTRALIA
[3]NICICO, Sarcheshmeh Copper Complex, Rafsanjan, IRAN

Abstract

Capture of fugitive emissions from charge ports, tap holes and many other openings in mineral processing and smelting vessels poses a significant environmental and hygiene problem. While fume and hygiene hoods are often used the highly buoyant nature of the fume combined with complex geometries makes their design with traditional engineering tools difficult. A computational fluid dynamics (CFD) model has been used in conjunction with engineering modelling to predict fume capture and emissions from a zinc slag fumer charge port and five Pierce-Smith converters.

Redesign of the slag fumer hood was found to be necessary after modelling showed substantial buoyancy in the fume meant that slots in the side of the hood are not adequate to ensure capture even at high airflow rates. Installation of the new hoods has reduced fugitive emissions by 65%.

Incremental improvements to fume capture from the Pierce-Smith copper converters via changes to operating procedures and minor modification to the plate were identified by the modelling approach. To ensure 100% fume capture in most operating conditions a secondary hood was found to be necessary and a design has been proposed. Further investigations are currently underway, with implementation of a final solution expected shortly.

Introduction

Hygiene or fume capture hoods are widely used in many industrial processes to capture fumes evolved from reaction vessels. Fume from metal processing operations such as smelting are toxic and corrosive. Release of such fume poses both an environmental problem and a health and safety hazard for plant workers. Traditional engineering design techniques are not adequate to allow novel designs to be developed and assessed.

To overcome such limitation computational fluid dynamics (CFD) modelling has been used [1] to predict the local flow field around a fume extraction hood for welding, however limited computing capabilities prevented buoyancy from being considered. More recently CFD was used [2] to design a laboratory fume hood and understand the complex interaction with the airflow in the room. For fume hood design, it has been noted [3] that CFD is providing insights into the complex interactions between hoods and the surrounding environment to allow improved performance and control of fume capture systems. While some work [4,5] using CFD in the design of fume capture systems for metal-processing applications has been undertaken this approach is yet to be widely adopted. None of the works have considered the fume hood as part of the larger drafting system. This paper reports on the application of CFD combined with engineering modelling to redesign of fume capture hoods for two metal processing applications. Two case studies are used to demonstrate how the approach has been used to solve fume release from the charge port of Zinifex's slag fumer and to develop a solution to Pierce-Smith converters operated by NICICO.

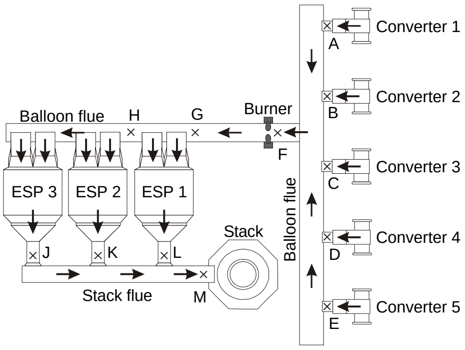
Modelling Approach

While a few authors [2,3,4,5] have used CFD models to assist in the design of fume hoods they have not accounted for the effect of the drafting system. Our approach has been to develop an engineering model of the drafting system and a CFD model of the fume hood. This has been found to be beneficial because the performance of the hood is directly related to the mass flowing though it, which is determined by the drafting system. Similarly the drafting system mass flow is influenced by the pressure drop across the hood and the amount of entrained air as this not only increases the mass flow but reduces the gas temperature.

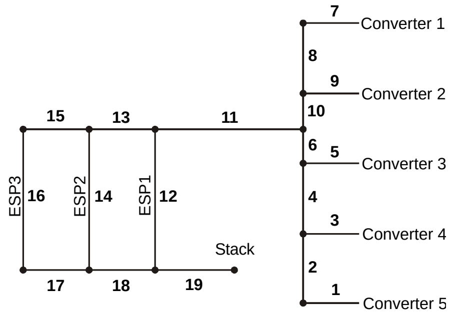
Engineering Model Description

The engineering model uses engineering correlations to predict flow rates, heat losses and pressure drops through the ducting system. Figure 1 shows the layout of the ducting system with five Pierce-Smith converters, two balloon flues, three electrostatic precipitators (ESPs), the stack flue and the stack itself for the NICICO Sarcheshmeh Copper Complex in Iran. Direction of the gas flow is shown with arrows. Also shown is the equivalent network of 19 pipes that the engineering model uses to simulate the flow domain.

The engineering model was integrated with a user-interactive computer program that enabled plant engineers and operators to gain a detailed understanding of the system. In conjunction with plant measurements, the model quantified the amount of in-leakage throughout the system, identifying it as a significant contributor to the poor extraction performance. The model gave unprecedented insight into the operation of a counter-intuitive buoyancy-driven extraction system. Further details of the engineering model are reported elsewhere [6,7].



(a) Plan view of converters and gas extraction system. Letters and X's represent measurement locations.



(b) Line diagram of EM pipe network. Numbers represent pipe sections.

Fig. 1: Physical and conceptual layout of the converters and associated gas extraction system for a copper smelter.

CFD Model Description

To understand gas flows in and around fume hoods a CFD model is used. In our work the commercial CFD code, CFX4 [8], has been used. Conservation equations for mass, momentum, energy and chemical species are solved to obtain the flow field, temperature distribution and fume concentration. The widely used k-e turbulence model [9] is used to account for turbulent effects. To capture buoyant effects the weakly compressible model is used for density, which assumes that density is a function of temperature and gas composition. Further details of the solution procedure can be found elsewhere [8].

Results

Slag Fumer Charge Port – Case 1

Zinifex Limited operate a slag fumer at their Port Pirie site. The slag fuming process utilises hot, ex-lead blast furnace slag and stockpiled granulated slag to produce zinc. Slag is supplied to the fumer in ladles at high temperature and poured through charge ports. During pouring it is thought that the fresh charge starts reacting and evolving fume as it enters the furnace. A consequence is that fume is emitted during pouring and drafting systems are required to capture and remove the fume to prevent its escape into the atmosphere.

A three-dimensional CFD domain of the slag fumer charge port area including details of the charge port, ladle and hood was generated using CFX-MESHBUILD. Two different geometries for the fume hood and extraction ducting were analysed: the original design and a new design. Block structure and mesh density in the models were selected and adjusted as necessary to achieve an optimal resolution and cell shape to minimise numerical problems. Slightly more than 436,000 cells were used in the model for the original design and 616,000 were needed to model the new design.

Figure 2 shows the charge port geometry and CFD model domain for the original design. The CFD domain extended from the floor to the roof, vertical panels above the charge port shown in Figure 2 represent the overhead crane that transports the ladle. Sides and front of the model domain were treated as pressure boundaries in the model and placed a reasonable distance from the hood to allow entrainment of surrounding air and a sink for fume. Round horizontal ducts attached to each side of the hood draw air through the hood and into the drafting system where it is cleaned before being vented to atmosphere. Internal chambers in the hood connect the ducts to slots in the hood. These slots are colored blue in Figure 2.

Pouring occurs over a time period of minutes and is treated as being at a steady state condition in the model with the ladle placed at a location near the end of the pour. Ladle surface temperature was set to that of the molten slag in the ladle. To get a reasonable flow profile for fume exiting the furnace through the charge port, a section of the furnace was included in the model. Inlet boundaries were applied to the open surfaces of the furnace, red areas shown in Figure 2, and fume added at a flow rate based on the amount of molten metal entering the furnace and an allowance for air entrained into the furnace with the molten metal.

Measurements of air velocities in the current drafting system duct indicated a velocity of 1.7m/s, which represents a total flow rate of 12000 Nm^3/hr through the two ducts. This was used as the lowest flow rate case. A medium flow rate case representing the maximum possible flow rate, calculated using an engineering model [7], for the current drafting system was modelled, along with a high flow rate case based on a new fan and baghouse.

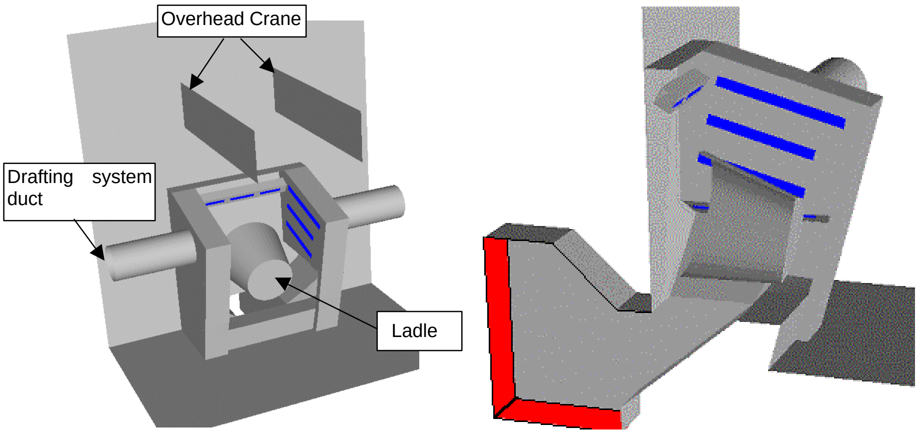


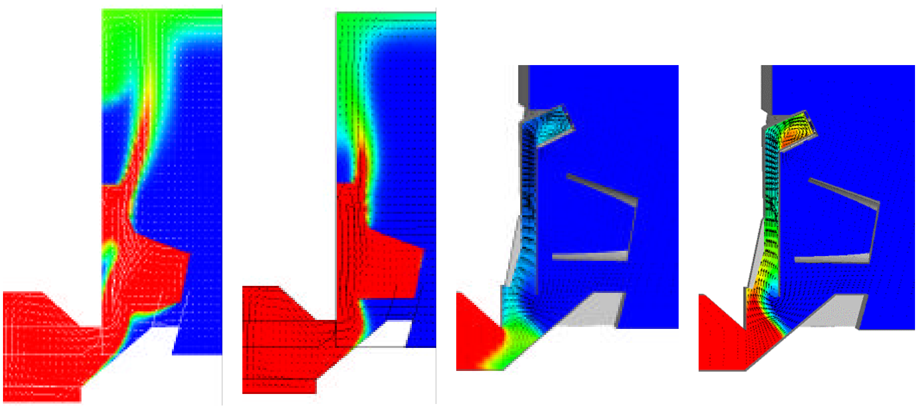
Fig. 2: Three-dimensional CFD model geometry of the original charge port, drafting slots shown in blue and the red area is the fume entry boundary condition in the furnace.

Plots of the fume concentration and air velocity for the low and high flow rate cases are presented in Figure 3. Figure 4 shows the fume distribution on a plane approximately across the mouth of the ladle for the low flow rate case. Table 1 summarises the results of fume capture showing the fraction of fume captured by the extraction system for the original design.

Experimental measurements of the flow in and near the charge port were beyond the scope of this project thus preventing quantitative validation of the model. Accuracy of the model was assessed qualitatively by comparing features predicted by the model, such as the plume shape and the escape of fume from the top slots as shown in Figure 4, with photographs and plant observations of the charging process. From these comparisons it was concluded that the model was capable of at least qualitatively capturing the flow around the charge port and could be used to assess potential new designs.

From these results it is clear that there is substantial buoyancy from hot fume in the furnace and that the temperature of the ladle drives the flow in an upward direction. Results in Table 1 show that increasing the airflow through the extraction system by almost an order of magnitude only increases fume capture from 10% to 50%. This analysis demonstrated that solely increasing the airflow through the drafting system would be insufficient to significantly improve fume capture and that a new hood design was needed.

CFD results of the original design showed that hot buoyant fume left the charge port at a low velocity and entrained air from the surroundings with the plume rising around the hot ladle. Closing in the area above the top of the ladle was not feasible because the ladle is suspended from an overhead crane. Significantly increasing airflow through slots located in the hoods to the side of the ladle was shown to be largely ineffective due to the highly buoyant nature of the hot plume and entrainment of surrounding air.



(a) (b) (c) (d)

Fig. 3: Side view of fume distribution on a central plane for the original design at the low (a) and high (b) flow rates. Fume distribution and velocity vectors for new hood design operating at the low flow rate case for two fume generation cases 360 Nm^3/hr (c) and 1440 Nm^3/hr (d).

CSIRO and Zinifex engineers worked jointly to develop a novel hood design based on capturing the fume as it left the charge port and utilising the buoyant nature of the plume to remove it into the ducting system before surrounding air could dilute it. To minimise cost minimal changes to the original hood were also a consideration.

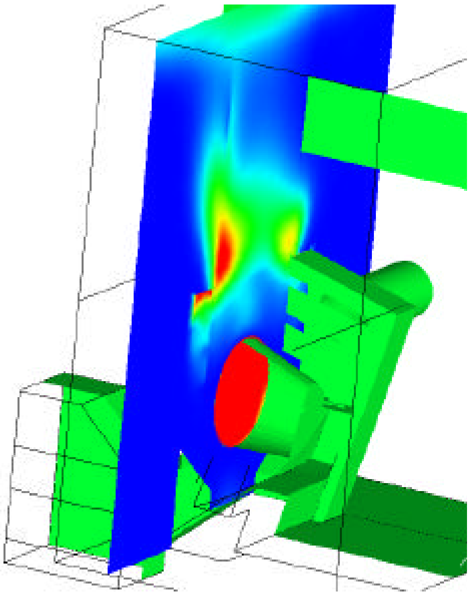


Fig. 4: Fume distribution for the original design at the low flow rate case on a plane near the ladle opening.

Figure 5 shows the design of the new charge port area. The side hoods are retained and a new top section and impact pad is installed. Internally the impact pad has a channel with an opening just above the charge port opening, which is connected to the drafting system. Slots along the face of the top section capture fume evolved from the ladle and are shown in Figure 5c. Internal ducts for each slot are used to ensure a uniform flow through each slot. The predicted flows through the slots are shown in Table 2.

Results for all the cases modelled cannot be included for space reasons but Figure 3 shows typical predicted fume distributions for the low drafting flow rate case with both low and high fume release rates. Predicted capture efficiencies for all the cases are presented in Table 1 and demonstrate that the new hood greatly improves fume capture. It was thought that the ladle position may influence the capture efficiency of the hood and a case with no ladle was run.

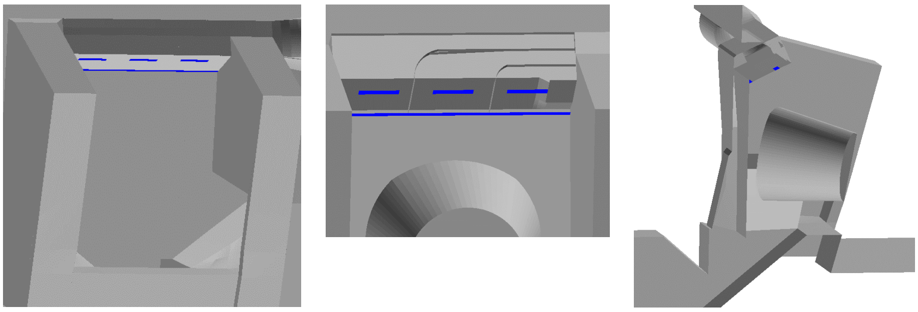
Once the modelling work was completed a detailed engineering design was performed and the hoods fabricated off site. During a planned plant shut down the new hoods were installed. When installed the new hoods were found to reduce fugitive fume emissions from the charge port by 65% [10], effectively solving a long-standing environmental problem at the Port Pirie smelter.

Table 1: Predicted Fume capture results.

Run	Drafting air Flow Rate [Nm³/hr]	Fume leaving the furnace [Nm³/hr]	Percentage of fume captured
Original a	12000	360	10%
Original b	75000	360	27%
Original c	100000	360	51%
New a	60000	360	100%
New b	60000	1440	100%
New c	75000	360	100%
New d	75000	1440	100%
New No Ladle	75000	1440	100%
New e	94000	360	100%
New f	94000	1440	100%

Table 2: Gas flow rates through slots in the New Hood.

Run	Slot Gas flows [Nm³/hr]		
	Left	Centre	Right
New a	2040	2007	1976
New b	2051	2016	1987
New c	2628	2621	2554
New d	2633	2623	2559
New No Ladle	2597	2656	2526
New e	3305	3347	3204
New f	3311	3353	3209



(a) (b) (c)

Fig. 5: Three-dimensional geometry of the new charge port design without the ladle (a) viewed through centre line (b) and detail of the slots and baffling in the top part of the new hood with the outer faces removed (blue regions are the slots).

Pierce-Smith converter fume capture – Case 2

The engineering model was used to model the drafting system of NICICO's five Pierce-Smith converters. Comparison of the predicted temperature and pressure at a number of locations in the system with plant measurements is shown in Table 3 for the case with two converters blowing. Good agreement with measured data gave confidence in the models predictions. Using the engineering model a number of changes to the system including the effects of hood gates, converters on line, hood dampers, flue insulation, electrostatic precipitator (ESP) operation, inclusion of a fan and inclusion of an acid plant were assessed. The effect of these different operating conditions on the hood pressure as well as the stack SO_2 concentration was calculated. These parameters are of interest because having a highly negative hood pressure will maximize SO_2 capture, while keeping a high stack SO_2 concentration will maximize the efficiency of an acid plant, which was under construction at the time of the work.

Conclusions drawn from the engineering model were: -

- Under normal blowing conditions approximately 200-220% infiltration air is drawn into the hood.
- With two converters blowing, approximately 30% of the total mass flow through the system is accounted for from in-leakage through the balloon and stack flues.
- Use of butterfly dampers to isolate converters that are on standby, combined with fully closing the hood gates on operating converters, will significantly increase draught within the operating converter hoods. Reduction of in-leakage will further improve the situation.
- The use of a fan to extract more gas through the system is inefficient since it will likely cause positive pressure within the balloon flues whilst reducing the temperature (and hence the pressure) at the base of the stack.

Table 3: Average plant measurements of temperature and pressure (meas) compared to engineering model predictions (EM).

	Temperature (°C)		Pressure (Pa g)	
Location	meas	EM	meas	EM
A	489		-12	
B	(496)[1]		(0)[1]	
C	395		-31	
D	-		-	
E	476		-7	
Ave	440	455	-17	-16
	2 converters blowing		*2 converters blowing*	
	meas	EM	meas	EM
F	183	180	-39	-41
G	213	199	-49	-48
H	200	195	-56	-48
J	(187)[1]	175	(-241)[1]	-357
K	(222)[1]	186	(-343)[1]	-381
L	-		-	
M	170	170	-417	-419

[1] Values in (brackets) are averages of at most two measurements. All other values in the table are averaged over more than two measurements.

The engineering model showed that some incremental improvements could be attained from modification of the drafting system but that changes to the converter hoods would be required to make a significant reduction in fume loss. Detailed CFD modelling of one converter was undertaken. Figure 6 shows the overall layout of the geometry that was modelled. Hot converter gases enter the flow domain from the bath surface (within the converter), while the surrounding air could be entrained from the front and sides of the flow domain around the converter and hood. Hot gases leave the flow domain through the duct to the balloon flue, while air and fugitive emissions can flow out from the hood mouth and hood gaps. The total number of grid cells was approximately 430,000.

Pierce-Smith converter operation has three main operating stages: Normal Blowing, Intermittent Blowing and Skimming/Pouring. Different gas flow rates for gas injected into the converter, different chemistry in the converter and converter mouth positions are used in each stage. Plant measurement and engineering model predictions indicated that the pressure at the duct connecting the converter to the balloon flue, see Figure 6, were different depending on the number of converters operating. A matrix of seven different operating conditions was developed to represent the typical operation of the converter with the CFD model run for each case. SO_2 capture efficiency for each case is presented in Table 4.

The intermittent blowing condition was a worst-case scenario with high blowing rates of 40000 Normal m^3/hr combined with a vessel rotated away from the hood and the hood gate completely open. Under these conditions the model predicts that only about 20-35% of the SO_2 is captured by the hood (see Table 4). The CFD results agree well with the plant observations, as shown in the photograph in Figure 7. By closing the hood gate, large amounts of the hot SO_2-rich gases would be captured. However, having the vessel on any angle facing away from the hood would direct it straight at the upper edge of the hood gate and thus provide conditions allowing SO_2 to escape. Also, the hood gate itself would get extremely hot under these conditions, possibly causing buckling of the gate and rendering it unusable.

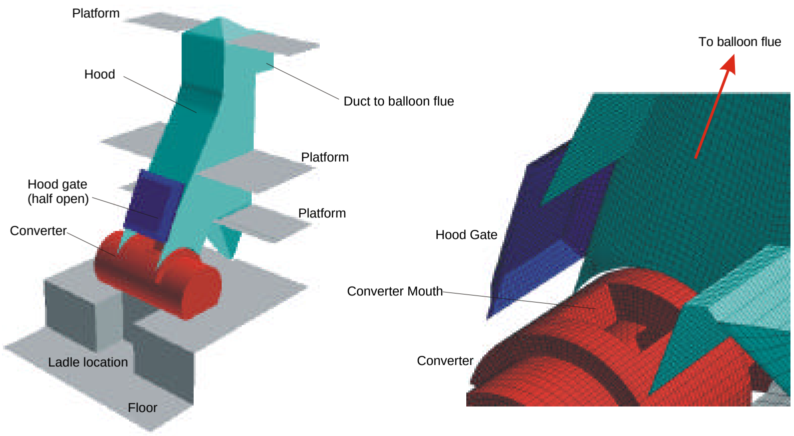


Fig. 6: Detailed geometry of the CFD flow domain, together with close-up of the surface grid in the converter mouth/hood gate region.

Table 4: SO_2 capture predicted by CFD modelling for each of the 7 initial modelling runs.

Flue Pressure	Normal Blowing	Intermittent blow	Skimming/ Pouring
0 Pa	98.5%	21.5%	
-15 Pa	100%	29.9%	97.5
-30 Pa	100%	35.5%	

Having performed modelling of the full ducting system and the converter hoods in detail, it was possible to identify specific methods of improving or eliminating the fume capture. The results showed that it might be possible to control fugitive emissions from the hood gate during normal blowing conditions by making minor modifications to the operation of the plant. However, the hoods themselves were only designed to capture fugitive emissions during normal blowing, not intermittent blows, skimming or pouring. Four modifications to operation and design were proposed.

- **Increased Hood Draught** to provide improved capture during intermittent blows, skimming and pouring. However, the engineering model had shown that significant increase in the flue draught is difficult to obtain.
- **Physical Blockage During Normal Blowing**. As the converter hood gates are often left partially (or entirely) open during normal blowing operations, a large gap exists between the hood and the top of the hood gate. Providing a blockage at the top of the hood gate could reduce leakage of SO_2 from this region.
- **Containment of Emissions using Blowers** to physically blow SO_2 back into the hood.
- **Secondary Hooding System** that extends over the existing hoods to capture what the existing hood misses, Figure 8.

The engineering model and CFD model were used to assess the viability of the various options. CFD modelling suggests that all SO_2 emissions can be captured under all operating conditions by using secondary hoods shown in Figure 8.

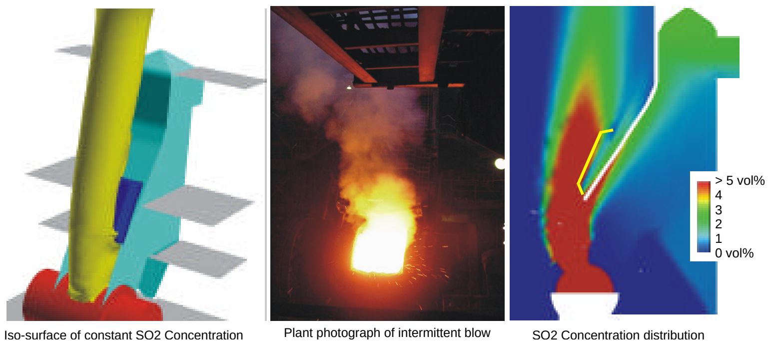


Fig. 7: SO_2 concentration distribution for Intermittent Blows at –30 Pa draught. The SO_2 iso-surface (shown in yellow) is at 1.0 vol% SO_2.

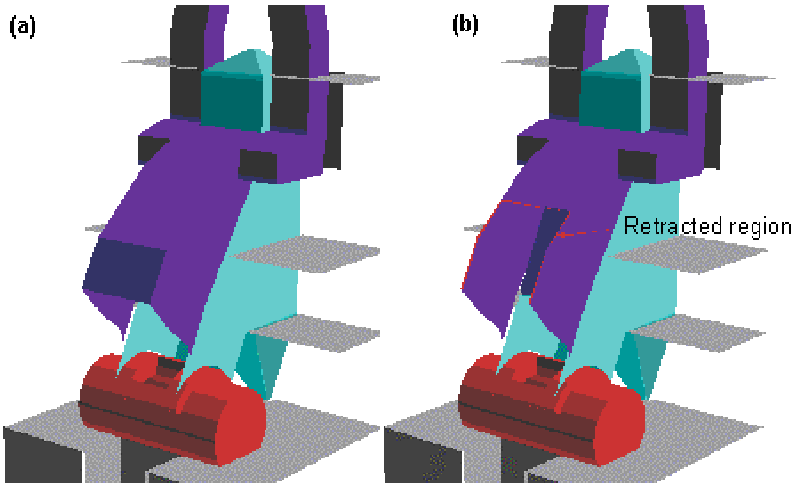


Fig. 8: Proposed design of secondary hood. (a) The full secondary hood, which would require a retractable upper section to allow access by the gantry crane. (b) The secondary hood in its retracted state where full crane access would be available.

Conclusion

A modelling approach based on combined engineering modelling of the ducting and drafting system and three-dimensional CFD modelling of the hood and fume generating process has been developed. The modelling approach has been used to design changes to the charge port of a zinc slag fumer and Pierce-Smith copper converters.

Redesign of the slag fumer hood was found to be necessary after modelling showed substantial buoyancy in the fume meant that slots in the side of the hood are not adequate to ensure capture even at high airflow rates. The new hood design is a novel departure from traditional hoods and the use of CFD was critical in development of the design. Installation of the new hoods has reduced fugitive emissions by 65% [10].

Incremental improvements to fume capture from the Pierce-Smith copper converters via changes to operating procedures and minor modification to the plant were identified by the modelling approach. Furthermore inclusion of a fan into the system was shown to not improve fume capture significantly and possibly be detrimental. To ensure 100% fume capture in most operating conditions a secondary hood was found to be necessary and a design has been proposed. Further investigations are currently underway, with implementation of a final solution expected shortly.

Acknowledgements

The authors gratefully acknowledge the help of Zinifex and NICICO's plant engineers and operators in assistance with the projects. The hard work of Dr Mittoni in initially setting up the projects is also gratefully acknowledged.

References

1. I.K. Kulmala, "Air Flow Field Near a Welding Exhaust Hood", *Applied Occupational and Environmental Hygiene*, **12(2)**, (1997), 101-104.
2. A. Kolesnikov, R. Ryan, and D.B. Walters, "Use of CFD to design containment systems for work with hazardous materials", *Chemical Health and Safety*, **10(2)**, (2003), 17-20.
3. D.B. Walters, "Laboratory hoods: quo vadis? past, present, and future" *Chemical Health and Safety*, **8(2)**, (2001), 17-22.
4. J.M. Berkoe, D.M.M. Lane, and B.M. Rosendall, "Computerized fluid dynamic (CFD) modeling, an important new engineering tool for design of smelting furnaces" *4th International Conference COPPER 99-COBRE 99*, **4** (1999), 53-66.
5. P. Safe, J. Deakin and S. Matson, "Effective design of converter hoods", *Proceedings of the 2002 TMS Annual Meeting*, February 17-21, 2002, Seattle, Washington, USA.
6. C. Solnordal, P.J. Witt, A. Manzoori, H. Namavari, E. Niknejad and M. Davari, "Whole-of-System Analysis Using CFD to Reduce Emissions of Fugitives Gases Inside a Copper Smelter Building", *METPLANT 2004: Metallurgical Plant Design & Operating Strategies*, Perth, WA, Australia, September 6-7, (2004) AusIMM.
7. P.J. Witt, C.B. Solnordal, L.J. Mittoni, S. Finn and J. Pluta, "Optimising the design of fume extraction hoods using a combination of engineering and CFD modelling", *Applied Mathematical Modelling* (submitted).
8. AEA Technology, *CFX-4.3: Solver Manual*, (AEA Technology, Harwell Laboratory, Oxfordshire, UK, 1999).
9. D.E. Launder and B.T. Sharma, "Application of the energy dissipation model of turbulence to the calculation of flow near a spinning disc", *Lett. Heat and Mass Transfer*, **1**, (1977), 131-138.
10. PASMINCO, "Emissions Cut By New Hood Design", *Pasminco News*, July 2002, 6.

* * *