Third International Conference on CFD in the Minerals and Process Industries
CSIRO, Melbourne, Australia
10-12 December 2003

REDUCTION IN FUME EMISSION FROM A SLAG FUMER CHARGE PORT USING CFD MODELLING

Peter WITT[1], Chris SOLNORDAL[1], Louis MITTONI[1], Stephen FINN[2] and John PLUTA[2]

[1] CSIRO Minerals, Clayton, Victoria 3169, AUSTRALIA
[2] Pasminco Limited, Melbourne, Victoria 3004, AUSTRALIA

ABSTRACT

Fume and hygiene hoods are widely used to prevent fugitive emissions from charge port, tap holes and many other openings in mineral processing and smelting vessels. The highly buoyant nature of the fume combined with often complex geometries make the design of these hoods difficult with traditional engineering tools. A computational fluid dynamics (CFD) model has been used to predict the fume capture and emission from a zinc slag fumer charge port. Model predictions show that increasing the draft flow rate by an order of magnitude would only give a marginal improvement in fume capture. Analysis of a novel new hood design shows that a significant improvement in fume capture is possible. Construction and installation of the hood has been performed and a 65% reduction in fume emission was achieved, thus eliminating a long-standing emission problem.

NOMENCLATURE

$\mathbf{g}$	gravity vector [m/s^2]
h	enthalpy [J/kg]
k	turbulent kinetic energy [m^2/s^2]
p	pressure [Pa]
T	temperature [K]
$\mathbf{u}$	mean velocity [m/s]
$\mathbf{u}'$	fluctuating velocity [m/s]
Y	mass fraction
ε	turbulent eddy dissipation [s^{-1}]
Γ	mass diffusivity [m/s]
λ	thermal conductivity [W/mK]
ρ	density [kg/m^3]
μ	dynamic viscosity [Pa-s]

INTRODUCTION

Hygiene or fume capture hoods are widely used in many industrial processes to capture fumes evolved from reaction vessels. Fume from many processes is often 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 Kulmala (1997) used computational fluid dynamics (CFD) modelling to predict the local flow field around a fume extraction hood for welding, however limited computing capabilities prevented buoyancy from being considered. More recently Kolesnikov *et al*. (2003) used CFD to design a laboratory fume hood and understand the complex interaction with the airflow in the room. For fume hood design, Walters (2001) notes 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. In the design of fume capture systems for mineral-processing applications, CFD has not been widely adopted with the exception of work by Berkoe *et al*. (1999) and Safe *et al*. (2002) for copper converters. Given the variety and large number of hoods used in minerals processing applications, there is potential to apply the technique to areas other than copper converters.

This paper reports on the application of CFD to the redesign of a fume capture hood for a slag fumer charge port.

BACKGROUND

Pasminco Port Pirie Smelter Pty. Ltd. (PPPS) 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.

The original slag fumer drafting circuit had deteriorated over a number of years and was found to have limitations in capturing fume emitted during charging. This resulted in a loss of product fume from the slag fumer.

Rather than simply refurbish the old system, which reportedly never performed adequately, Pasminco decided to undertake CFD modelling of the charge port hood combined with an engineering analysis of the pressure and flow characteristics of the ducting system. Details of the engineering analysis are not reported here but the novel application of CFD to the design of the new hood is discussed in this paper.

MODEL DESCRIPTION

Conservation Equations

To calculate the flow field, the model solves the Reynolds Average Navier-Stokes equations given below.

$$\nabla \cdot (\rho \mathbf{u}) = 0 \quad (1)$$

$$\nabla \cdot (\rho \mathbf{uu}) = -\nabla p + \rho \mathbf{g} + \nabla \cdot \mu \nabla \mathbf{u} - \nabla \cdot \left(\rho \overline{\mathbf{u}'\mathbf{u}'}\right) \quad (2)$$

Airflow in the area of the hood is strongly influenced by buoyancy of the fume leaving the charge port. To allow prediction of the temperature field, the enthalpy equation is included in the model.

$$\nabla \cdot (\rho \mathbf{u} h) = -\nabla \cdot \lambda \nabla T \quad (3)$$

An additional scalar equation is used to track the motion of fume emitted from the charge port so that fume transport and capture efficiency can be predicted.

$$\nabla \cdot (\rho \mathbf{u} Y) = \nabla \cdot \Gamma \nabla Y \quad (4)$$

The Reynolds stress terms on the right hand side of equation (2) mean that the above equations are not fully closed. To obtain values for the Reynolds stress terms and close the equation set a turbulence model is used.

Turbulence Model

The most widely used turbulence model is the k-ε model of Launder and Sharma, (1977), which is based on the Boussinesq approximation. In the k-ε model, the Reynolds stress terms are approximated by:

$$\rho \overline{\mathbf{u}'\mathbf{u}'} = \mu_T \nabla \mathbf{u} - \frac{2}{3} \rho k \delta \quad (5)$$

where μ_T is the turbulent or eddy viscosity obtained from:

$$\mu_T = C_\mu \rho \frac{k^2}{\varepsilon} \quad (6)$$

An effective viscosity can then be defined $(\mu_{eff} = \mu + \mu_T)$. The equations below are solved to obtain k and ε, which are the turbulent kinetic energy and dissipation rate respectively.

$$\nabla \cdot (\rho \mathbf{u} k) - \nabla \cdot \left(\mu + \frac{\mu_T}{\sigma_k} \right) \nabla k = P - \rho \varepsilon \quad (7)$$

$$\nabla \cdot (\rho \mathbf{u} \varepsilon) - \nabla \cdot \left(\mu + \frac{\mu_T}{\sigma_\varepsilon} \right) \nabla \varepsilon = C_1 \frac{\varepsilon}{k} P - C_2 \rho \frac{\varepsilon^2}{k} \quad (8)$$

Shear production is defined as:

$$P = \mu_{eff} \nabla \mathbf{u} \cdot \left(\nabla \mathbf{u} + (\nabla \mathbf{u})^T \right) - \frac{2}{3} \nabla \cdot \mathbf{u} \left(\mu_{eff} \nabla \cdot \mathbf{u} + \rho k \right) \quad (9)$$

Constants for the standard k-ε model are given in Launder and Sharma, (1977).

Numerical Scheme

Solution of the above equations by analytical techniques is not possible. To solve the equations and obtain the flow field, temperature distribution and concentration of fume the commercial CFD code, CFX4 by AEA Technology (1999) was used. CFX solves the equations given above using the finite volume method on a co-located body fitted grid. To avoid chequer-board oscillations in the pressure field, the Rhie and Chow (1983) interpolation procedure is used. Coupling between pressure and velocity is achieved using the SIMPLEC algorithm, which is a modified form of the SIMPLE algorithm and is described elsewhere (AEA Technology, 1999 and Patankar, 1983). 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 are given in AEA Technology (1999).

Model Geometry and Boundary Conditions

A three-dimensional CFD mesh 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.

A plot of the original design geometry is shown in Figures 1 and 2. The domain of the model extended from the floor to the roof. Vertical panels above the charge port shown in Figure 1 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 1.

The level of slag in the ladle changes during pouring, requiring the ladle to be rotated and moved closer to the charge port. It was not considered necessary to include this for the original design case so the ladle was positioned near the end of the pour. Pouring occurs over a time period of minutes and is treated as being at a steady state condition in the model. 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. Fume was assumed to be at the furnace temperature.

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 for the current drafting system was modelled, along with a high flow rate case based on a new fan and baghouse.

Figures 5 to 7 show the geometry of the new hood design. Details of the design are discussed below. For the new hood design seven different operating conditions were analysed with three different air flow rates through the ducts and two fume release rates from the furnace and for the typical case a simulation was performed without the ladle to assess the effect of the ladle. Table 1 provides a summary of the operating conditions and results for all the cases modelled.

RESULTS

Original Hood Design

The physical geometry of the original charge port is shown in Figures 1 and 2. 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 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. The results 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 the fume capture.

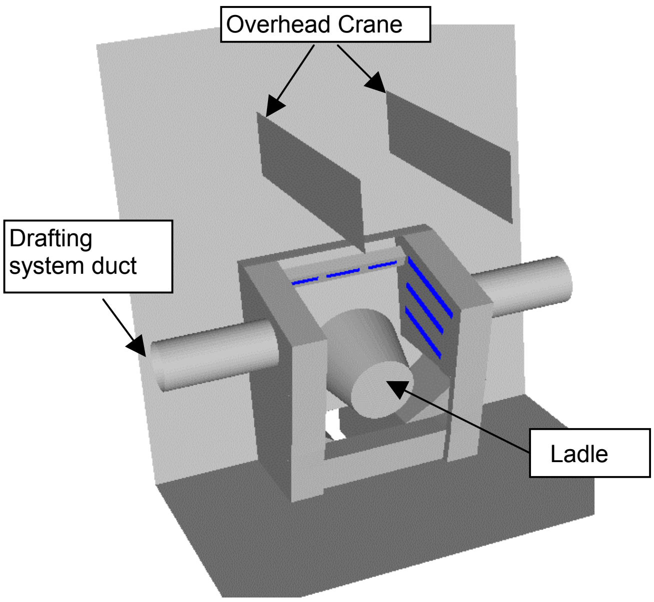

Figure 1: Three-dimensional CFD model geometry of the original charge port, drafting slots shown in blue.

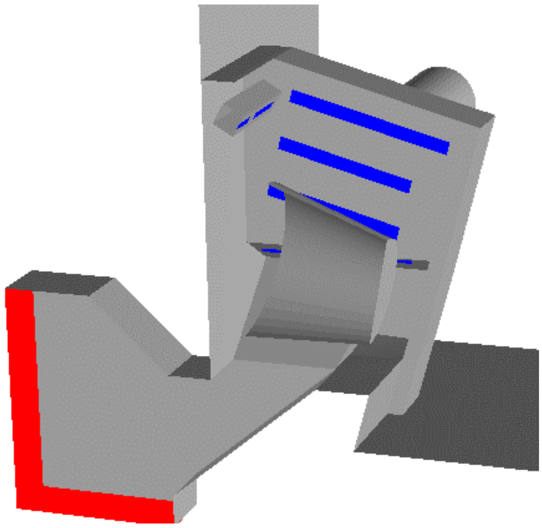

Figure 2: Three-dimensional CFD model of the original charge port design cut through the centre-line, blue areas indicate slots and the red area is the fume entry boundary condition in the furnace.

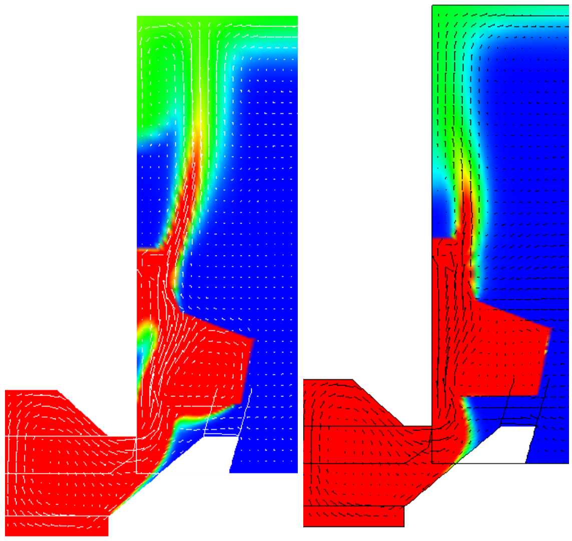

Figure 3: Side view of fume distribution for the original design at the low (left) and high (right) flow rate on a central plane.

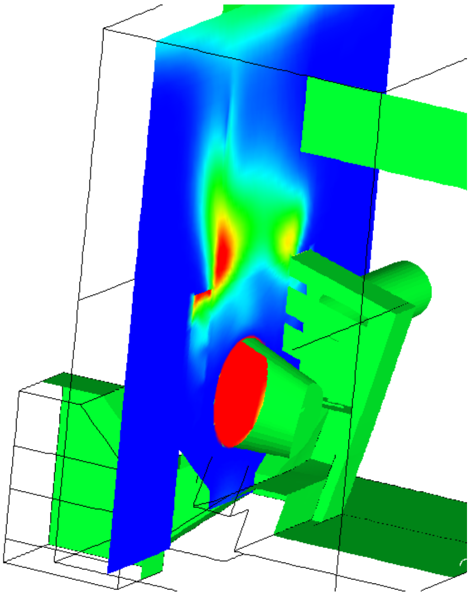

Figure 4: Fume distribution for the original design at the low flow rate case on a plane near the ladle opening.

New Hood Design

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.

CSIRO and Pasminco engineers worked jointly to develop a novel new hood design based on capturing the fume as it left the charge port and utilise 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.

Figure 5 shows the new charge port area without the ladle. 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, see Figure 6, 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 7. 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 8 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. The predicted fume distribution is shown in Figure 9. Results from this run indicated that the presence of the ladle produces a plume that is captured by slots in the top of the hood. It also shows that the ladle acts to shield the charge port slightly and more air is drawn into the charge port for the case without the ladle.

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% (Pasminco, 2002), effectively solving a long-standing environmental problem at Pasminco's Port Pirie smelter.

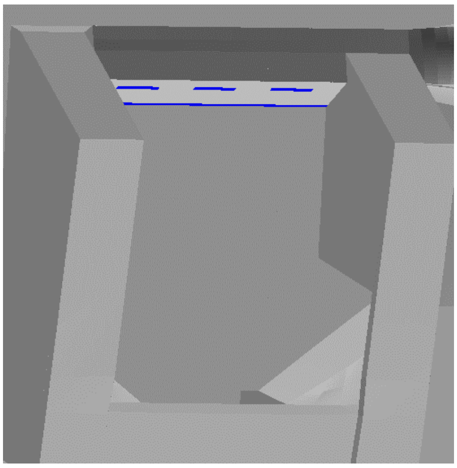

Figure 5: Three-dimensional geometry of the new charge port design without the ladle (drafting slots shown in blue)

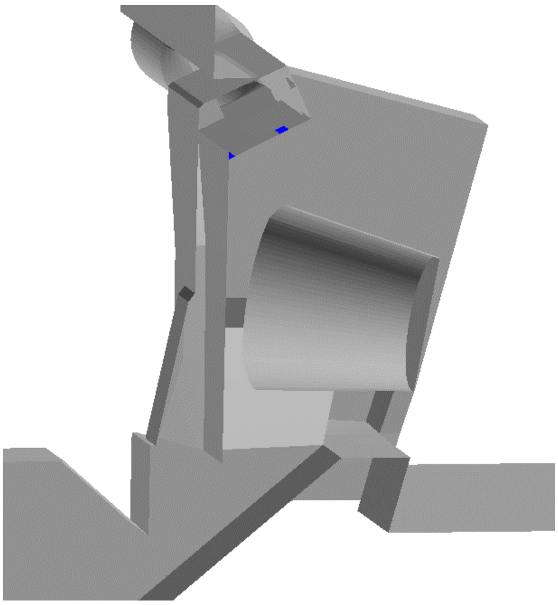

Figure 6: Three-dimensional geometry of the new charge port cut through centre line.

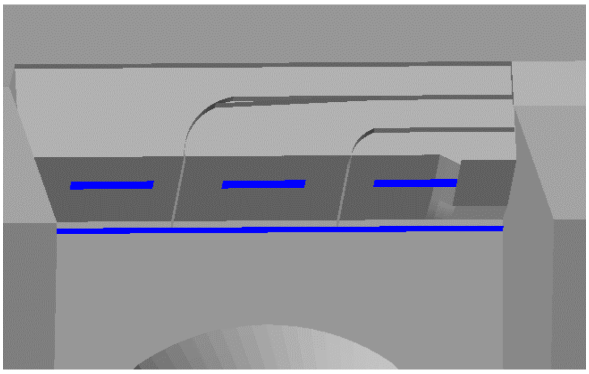

Figure 7: Detail of the slots and baffling in the top part of the new hood with the outer faces removed (blue regions are the slots)

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%

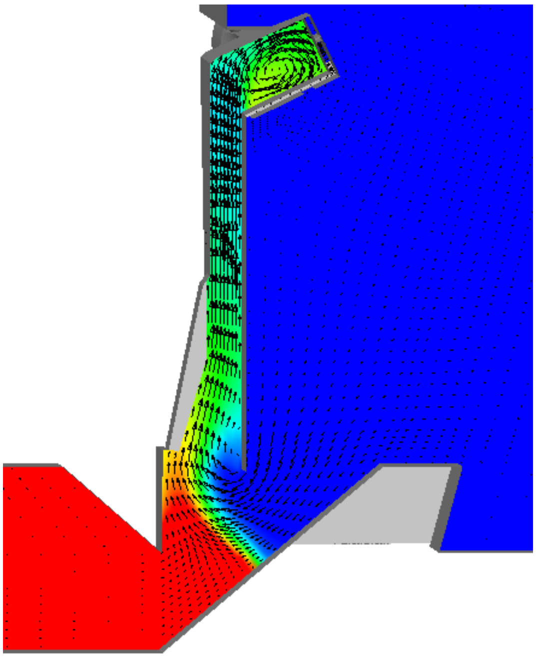

Figure 9: Fume distribution and velocity vectors 0.5m from the ladle centre-line for new hood design without the ladle at the medium flow rate case with high fume generation.

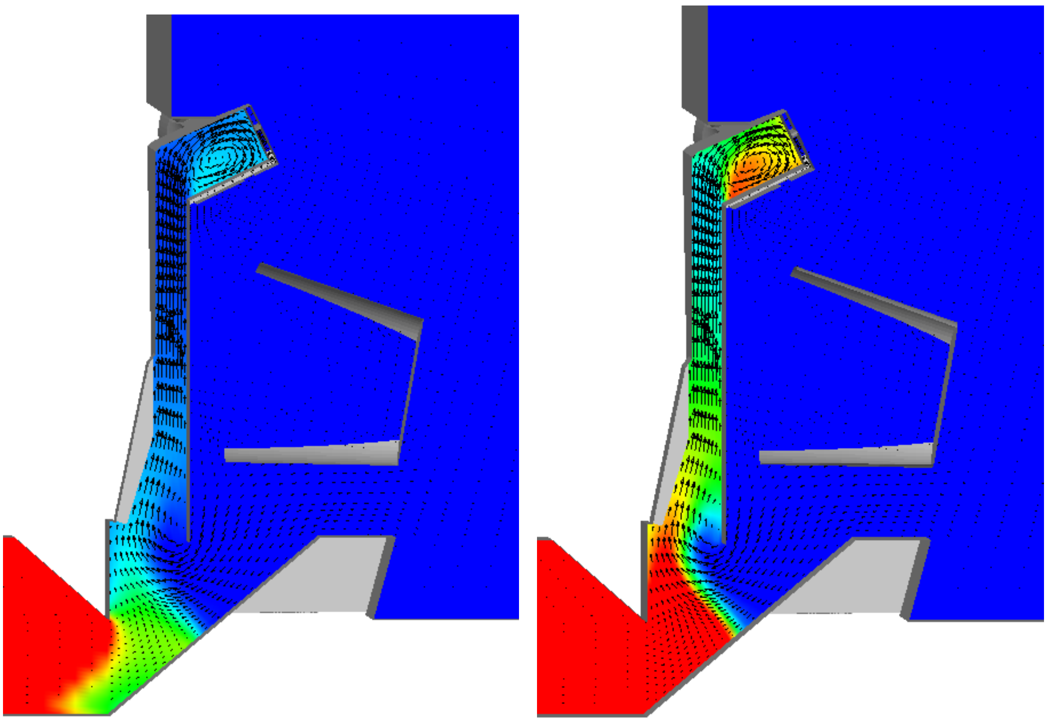

Figure 8: Fume distribution and velocity vectors 0.5m from the ladle centre-line for new hood design at the low flow rate case, for two fume generation cases 360 Nm^3/hr (left) and 1440 Nm^3/hr (right).

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

Run	*Slot Gas flows [Nm^3/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

CONCLUSION

A three-dimensional CFD model of the flow of fume and air around a slag fumer charge port and ladle has been developed. Model results show that hot fume from the charge port has substantial buoyancy and that slots in the side of the hood are not adequate to ensure capture even at high airflow rates.

Redesign of the hood has been undertaken with the aid of CFD modelling to ensure that the hood design would perform as expected. 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% (Pasminco, 2002).

REFERENCES

AEA TECHNOLOGY, (1999), “CFX-4.3: Solver Manual”, AEA Technology, Harwell Laboratory, Oxfordshire, UK.

BERKOE, J.M.; LANE, D.M.M. and ROSENDALL, B.M., (1999), “Computerized fluid dynamic (CFD) modeling, an important new engineering tool for design of smelting furnaces” *4th International Conference COPPER 99-COBRE 99,* **4**, pp. 53-66.

KOLESNIKOV, A, RYAN, R; and WALTERS, D.B., (2003), “Use of CFD to design containment systems for work with hazardous materials”, *Chemical Health and Safety*, **10(2)**, pp. 17-20.

KULMALA, I.K., (1997), “Air Flow Field Near a Welding Exhaust Hood”, *Applied Occupational and Environmental Hygiene*, **12(2)**, pp. 101-104.

LAUNDER, D.E. and SHARMA, B.T., (1977), “Application of the energy dissipation model of turbulence to the calculation of flow near a spinning disc”, *Lett. Heat and Mass Transfer*, **1**, 131-138.

PASMINCO, (2002), “Emissions Cut By New Hood Design”, *Pasminco News*, July 2002, p. 6.

PATANKAR, S.V., (1983), *Numerical heat transfer and fluid flow*, Hemisphere.

RHIE, C.M. and CHOW, W.L., (1983), "Numerical study of the turbulent flow past an airfoil with trailing edge separation", *AIAA J*, **21,** 1527-1532.

SAFE, P.; DEAKIN, J. and MATSON, S, (2002), “Effective design of converter hoods”, *TMS Annual Meeting Volume: 2002*, pp. 99.

WALTERS, D.B., (2001), “Laboratory hoods: quo vadis? past, present, and future” *Chemical Health and Safety*, **8(2)**, pp. 17-22.