*Seventh International Conference on CFD in the Minerals and Process Industries*
*CSIRO, Melbourne, Australia*
*9-11 December 2009*

# INDUSTRIAL APPLICATION OF CFD – A REVIEW OF FEW EXAMPLES

**J. NASER**

Faculty of Engineering & Industrial Science Department
Swinburne University of Technology, Hawthorn 3122 , AUSTRALA

## ABSTRACT

The paper reviews six examples of CFD applications, useful for: minerals, power generation and oil & gas industry. The review starts with the multiphase modeling of gas injection into a liquid bath, where the CFD results were validated against experimental data. The example is very useful for metal extraction/refining industry. Simulation of a mineral concentrator bin is presented next, where multiphase modeling was used to find out the cause of blocking of the slurry pump. The 3rd example presents two alternative designs, evaluated for better extraction of dust from an underground mine crushing-plant. Multiphase modeling with boiling-heat transfer was used in the next example for investigating a low pressure water cooling system in slag fumers. The next example reviews full scale coal combustion modelling of a tangentially fired boiler with mill ducting. Finally a successful modeling of a gas liquid separator is presented. The CFD software FIRE (AVL 2009) was used in most of the investigations. Geometry dimensions and process data were suppressed as necessary for confidentiality reasons.

### Example-1. Gas injections into liquid baths using Top Submerged Lances (TSL)

There are various methods for gas injection into molten baths including Top Submerged Lance (TSL) technology, which uses a submerged vertical lance within an upright cylindrical furnace (Huda et al 2009, Floyd 1996). Through the lance, oxygen enriched air is injected with swirl into the molten bath which provides intense mixing of the bath and excellent contact between phases. Floyd (2005) has described the details of the Top Submerged Lance technology and its development since the 1970s. This paper reviews the fluid flow phenomena in a cylindrical bath stirred by Top Submerged Lance (TSL) gas injection, investigated by using CFD for an isothermal air-water system (Huda et al 2009). The multiphase flow simulation, based on the Euler-Euler approach, elucidated the effect of swirl and non-swirl flow inside the bath (Fig. 1). The effects of lance submergence level and air flow rate were also investigated in the study. The simulation results for velocity fields (Fig. 1(c)) and generation of turbulence in the bath were validated against existing experimental data from the previous model experimental study of Morsi et al. (2000).

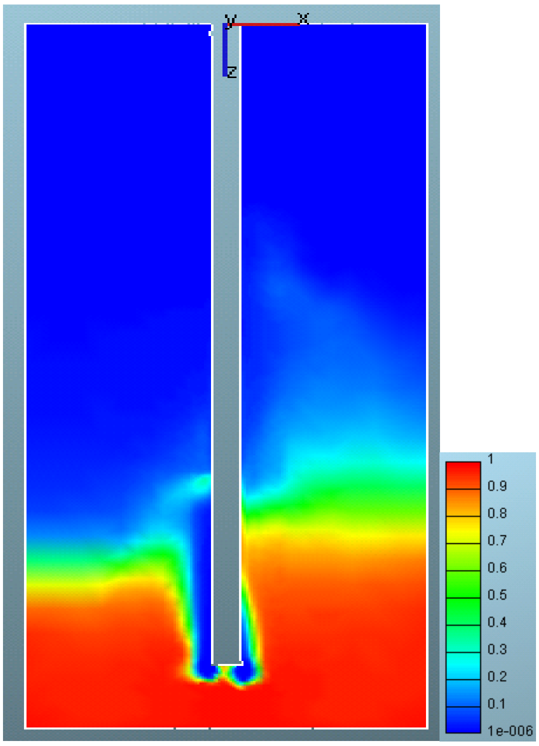



**Figure 1(a)** Volume fraction of liquid (TSL) (Huda et al 2009).

### Example-2. The mineral concentrator bin

The mineral concentrator bin reviewed here was reported to show problems of accumulation of solids on the upper slanting walls. Investigations were carried out to find out the possible cause of this unusual accumulation of solids. CFD simulations were carried out for the exact operating configuration and geometries modified to overcome the problem. A quarter of the geometry was used in the model due to structural beams inside. Figure 2(a) shows the grid and Figure2 (b) shows the accumulation of solids on the upper slanting walls, which will grow with time and eventually slump into the bottom, causing blockage problem for the slurry pump suction, located at the bottom of the bin.

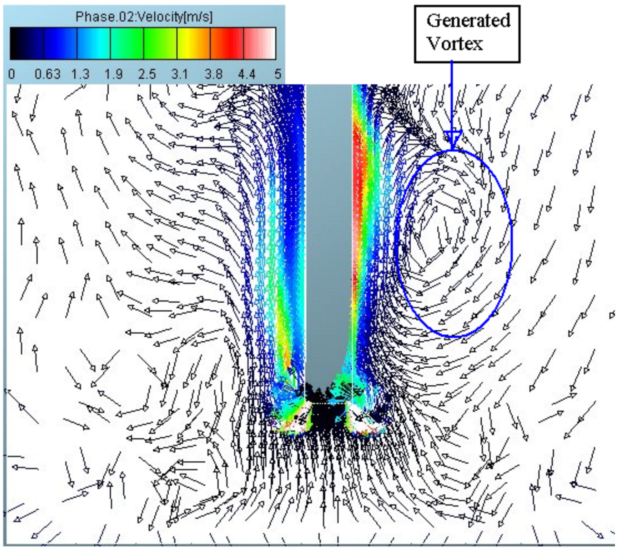



**Figure 1(b)** Velocity vectors for liquid phase (m/s) (TSL) (Huda et al 2009).

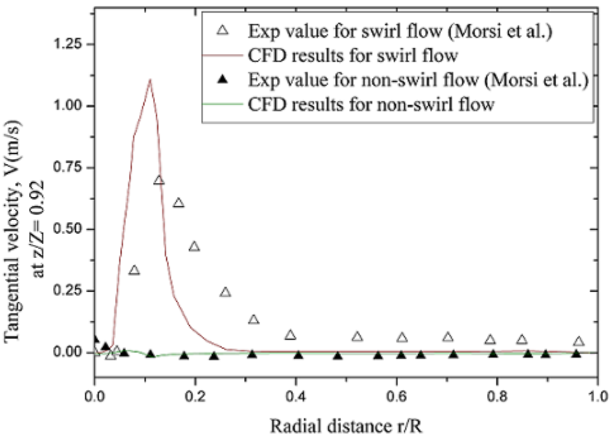



**Figure 1(c):** Mean tangential velocity comparison between swirl and non-swirl flow (TSL) (Huda et al 2009).

**Example-3. Dust collection systems in an underground mine-crushing plant**

Extraction of ore using block caving methods is common in underground mines. The ore extracted from the draw point is hauled to the underground crushing station using load haul dump (LHD) trucks and tipped or dropped into the run of mine (ROM) bin.

A typical LHD truck tipping into a ROM bin is shown in Fig.3(a). Dust is released during the tipping process; the released dust often causes visibility problem to the LHD drivers. The released dust is expected to go out along with the outgoing ventilation air. In the existing conventional approach, the outgoing dust laden air is collected through an exit in the ceiling of the chamber housing the ROM bin. The dispersion of dust is governed by local airflow controlled by the mechanical ventilation system and is further complicated by the localized turbulent air motion generated by large volumes of material undergoing drop feed (Naser et al. 2008).

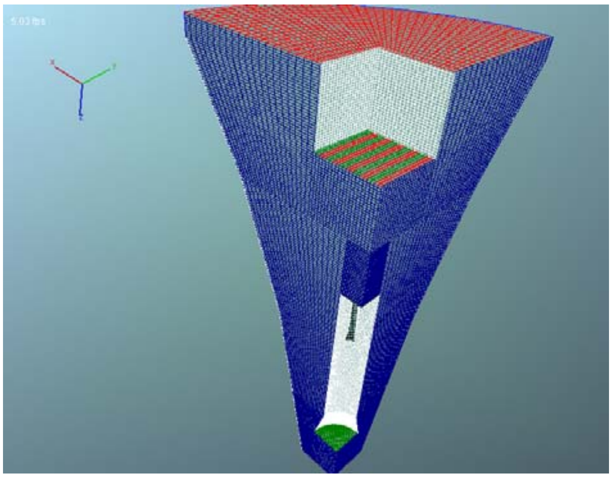


**Figure 2(a)** The computational grid used for concentrator bin.

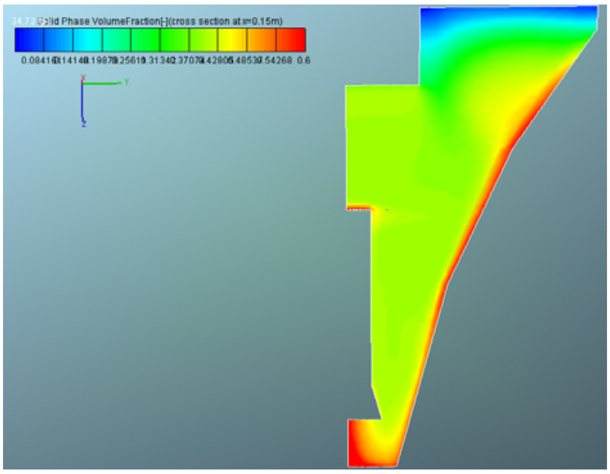


**Figure 2(b)** Solid-phase volume fraction, showing accumulation on the slanting wall. The colour key varies uniformly from 0.0 (blue) to 0.6 (red)

The piston effect of the LHDs returning after dropping the ore in the ROM bin causes the dust to be sucked into the tunnel and cause visibility problem. CFD modeling of two alternative dust collection systems in an underground mine-crushing plant was reported in Naser et al. 2008. A typical grid is shown in Fig.3(b). In the proposed system, dust was collected/absorbed through the holes along the top edge of the ROM bin. The dust collection performances of both systems (Fig. 3c & 3d) is summarized. Dust was found to be well contained within the crusher bin for the proposed system which stands out as a viable option.

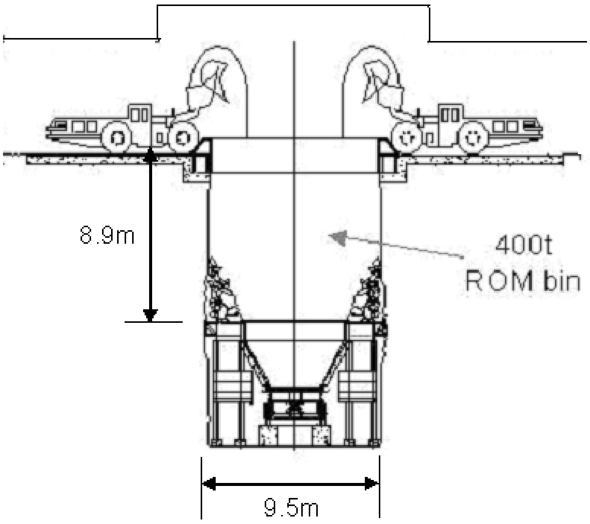



**Figure 3(a)** Typical LDH trucks tipping into a ROM bin (Naser et al. 2008).

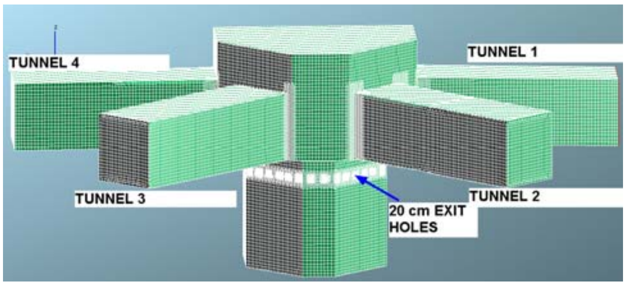



**Figure 3(b)** A typical computational grid used, exit through holes (Naser et al. 2008).

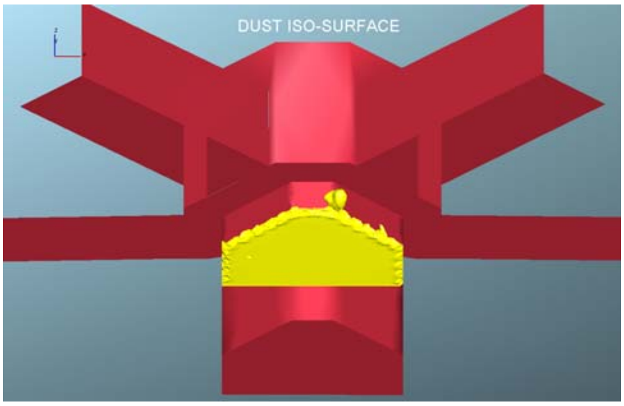



**Figure 3(c)** Dust iso-contours or iso-surface (100mg/m$^3$) exit through holes (proposed system) (Naser et al. 2008).

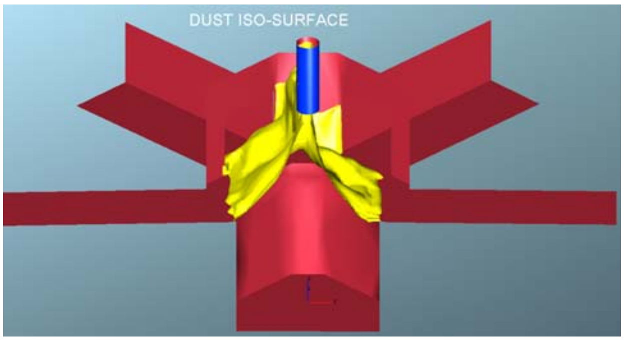



**Fig. 3(d).** Dust iso-contours or iso-surface (100mg/m$^3$), exit through top of tipple (existing system) (Naser et al. 2008).

**Example-4. Low pressure water cooling system in slag fumers**

Modelling was undertaken to evaluate the process performance of low pressure (LP) steam system (Fig4a). A typical set of jackets (Fig4b) was also modelled to determine the input conditions to the LP drum. Detail input conditions were developed from a combination of measured data, operating conditions and CFD modeling of above mentioned jacket. Full size solution geometry was developed from the drawings. The multiphase flow simulation, based on Euler-Euler approach was used. Boiling was modelled through interfacial mass and heat transfer.

Temperatures within the jackets were found to approach the boiling point until the flow reaches close to LP drum. However, localised boiling and trapped steam appears to be present in the jacket system (Fig.4c). Evidence of some steam entering the LP drum is also clear in Fig.4c. The modelled values of outlet temperature and velocity were well within the range of measured data.

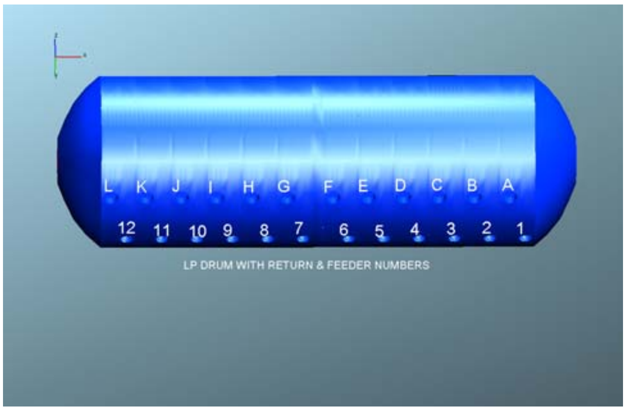



**Figure 4(a)** Geometry of LP Drum showing jacket returns & jacket feeders

Temperatures of returns I, J,1 & 2 (Fig.4a) flowing into the LP drum were above the boiling point and hence show boiling occuring from these returns (Fig.4d). The 10% by volume of steam released from the other returns (below the boiling point) appears to condense before it reaches the free surface. The average temperature of water in the LP drum was consistent with the infra red camera reading.

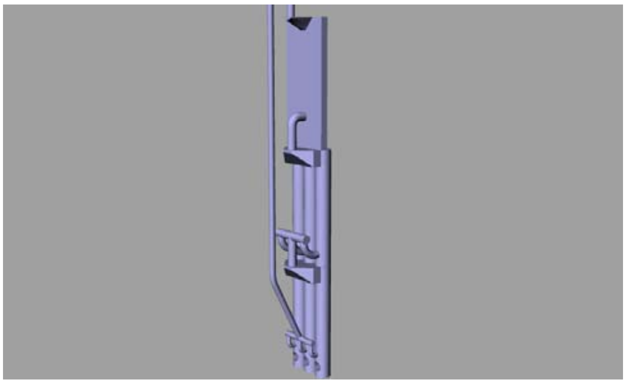


**Figure 4(b)** Geometry of typical cooling jacket set (zoomed)

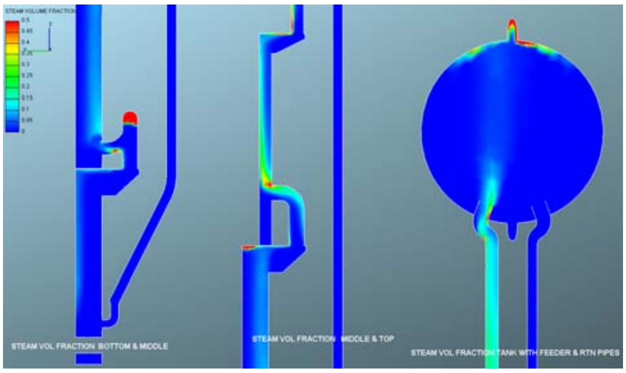



**Figure 4(c)** Steam volume fraction on a vertical plane through the middle of the jackets (showing side view). The colour key varies uniformly from 0.0 (blue) to 0.9 (red)

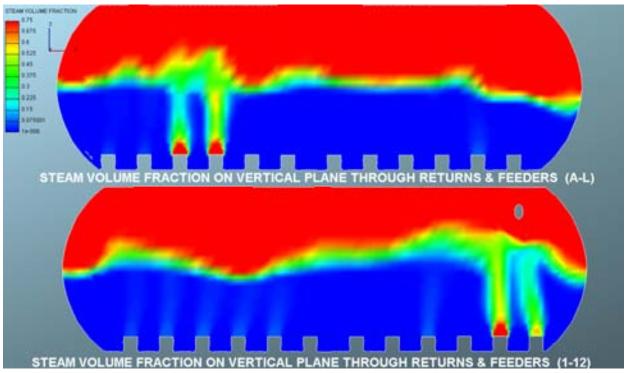



**Figure 4(d)** Steam volume fraction on a vertical plane through returns and feeders. The colour key varies uniformly from 0.0 (blue) to 0.9 (red)

**Example-5. Full scale coal combustion modelling of tangentially fired boiler with mill ducting**

A full scale combustion model incorporating upstream mill ducting of a large tangentially fired boiler with flue gas recirculation was examined numerically (Achim et al 2009). Fig5a shows the geometry of the boiler. Lagrangian particle tracking was used to determine the coal particle paths and the Eddy Dissipation Model for the analysis of the gas phase combustion. Volatiles and gaseous char products, given off by the coal particles were modelled by Arrhenius single phase reactions. Generally, the findings indicated reasonable agreement with observed qualitative and quantitative data of incident heat flux on the walls (Fig. 5d). Figure 5b shows the velocity vectors on Lower Boiler Plane. Figure 5c shows the temperature contours on the Lower Boiler Plane. The model developed here can be used for a range of applications in furnace design and optimisation of gas emissions of coal fired boiler plants.

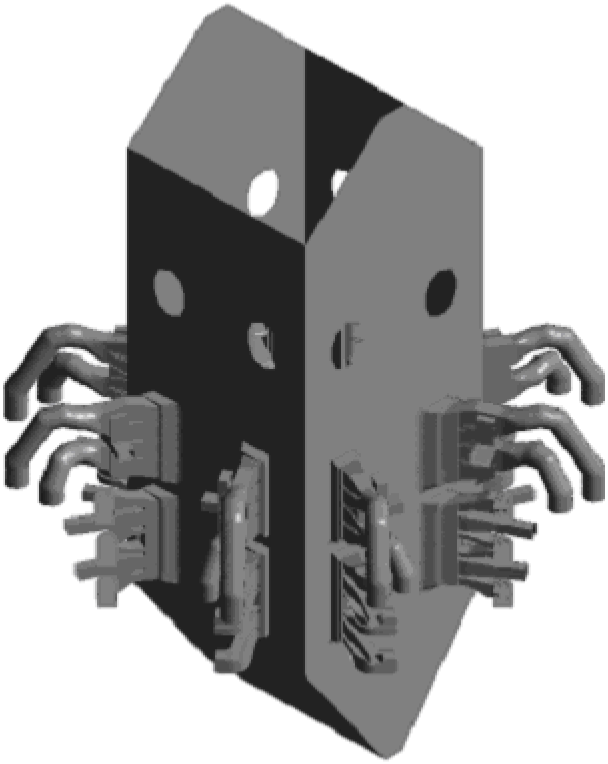


**Figure 5(a)** Full scale tangentially fired boiler with mill ducting (Achim et al 2009).

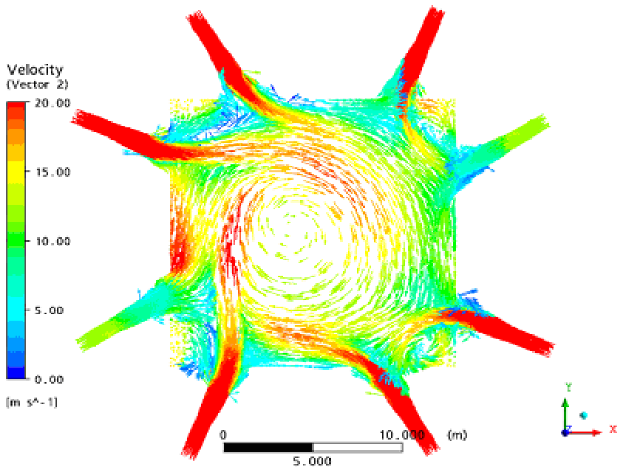



**Figure 5(b)** Velocity Vectors on Lower Boiler Plane (Achim et al 2009).

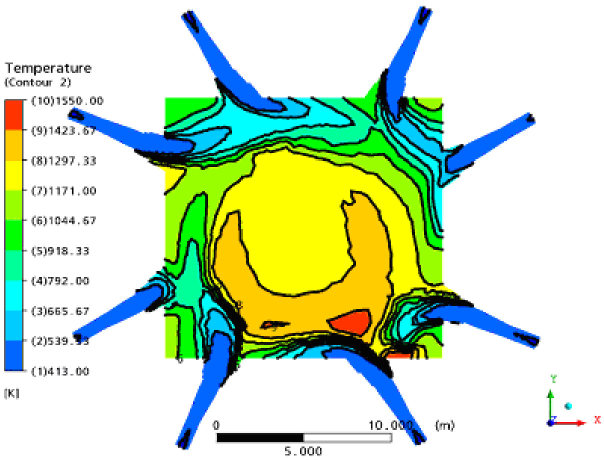



**Figure 5©** Temperature contours on Lower Boiler Plane (Achim et al 2009).

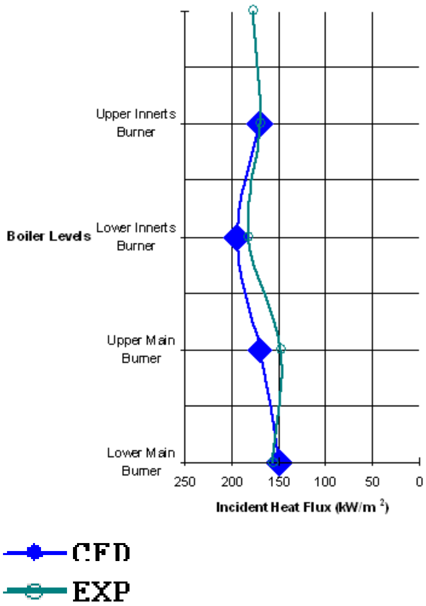
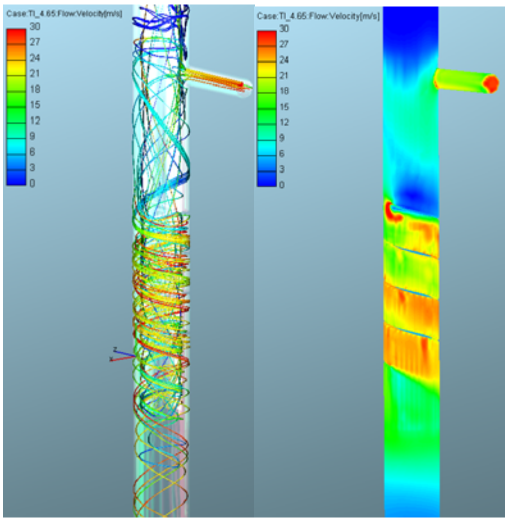



**Figure 5(d)** Comparison of measured incident heat flux on the wall with prediction (Achim et al 2009).

**Example-6. Sub-sea gas liquid separator**

Modelling of a laboratory scale gas-liquid separator designed for high gas content was carried out in (Ahmed et al. 2009). The separator consists of two concentric pipes with swirl tube in the annular space between the pipes (Fig. 6a & b). The gas-liquid mixture injected tangentially from the side inlet and the system works with a combination of gravity and centrifugal forces to achieve a high-efficient gas-liquid separation. Three dimensional transient gas flow was coupled with the modified Eulerian-Lagrangian spray and wall film models. Spray model involves multi-phase flow phenomena and requires the numerical solution of conservation equations for the gas and the liquid phase simultaneously. With respect to the liquids phase, discrete droplet method (DDM) was used. The droplet-gas momentum exchange, droplet coalesces and breaks-up, droplet-wall interaction with wall-film generation and entrainment of the water droplet back into the gas stream were taken into account in this investigation. The streamline pattern and near-wall gas velocity is shown in (Fig. 6a & b). The performance of the gas-liquid separator was visually established for a range of gas flow rates, with volume fraction VF=0.874-0.985, by observing the liquid carry over (LCO) regime where liquid was carried out in the gas stream. The liquid and gas flow rates at which the LCO was observed defines the upper operational range of the separator. Air-water mixture was used in the numerical simulations to keep consistent with the experiments.




(a) (b)

**Figure 6 (a)** Velocity streamlines showing the flow path of air and **(b)** velocity in the near wall regions (Ahmed et al. 2009)

The pressure between the inlet and exit was validated against the experiments for different air-water flow rate combinations (Fig. 6c)

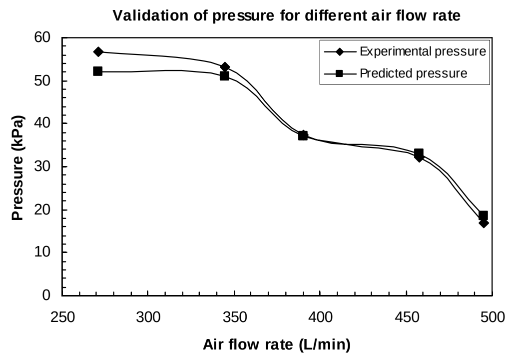
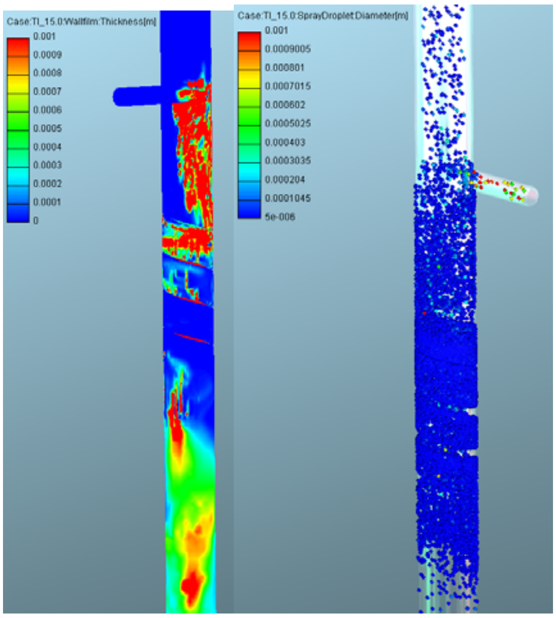



**Figure 6(c)** Validation of the predicted pressure between the inlet and air outlet for different air flow rates (Ahmed et al. 2009).

The predicted liquid wall film thickness and the liquid droplets for carry over situation are shown in Fig. 6d & 6e




(d) (e)

**Figure 6(d)** Liquid wall film thickness **(e)** Liquid droplet carry over (Ahmed et al. 2009)

## CONCLUSION

The review of six industrial applications showed that CFD is a mature and reliable tool for evaluating many industrial processes. However, appropriate knowledge and experience are very essential for obtaining reliable results.

## REFERENCES:


ACHIM, D., NASER, J., MORSI, Y., PASCOE, S. "Numerical investigation of full scale coal combustion modelof tangentially fired boiler with the effect of mill ducting" *Heat Mass Transfer, DOI 10.1007/s00231-009-0539-0*

S. AHMED, G. SOTO, J. NASER3 AND E. NAKAGAWA "A modified eulerian-lagrangian approach for solving multi-phase flow applied to a compact down-hole sub-sea gas-liquid sepatator", *Proceedings of the*

*International Conference on Mechanical Engineerin, 26- 28 December 2009, Dhaka,*

AVL FIRE http://www.avl.com

J. FLOYD, "The Third Decade of Top Submerged Lance Technology " *in Howard Worner International Symposium on Injection in Pyrometallurgy (1996), Melbourne, Australia, TMS, 417-428.*

J. FLOYD, "Converting an Idea into a Worldwide Business Commercializing Smelting Technology," *Metallurgical and Materials Transaction B, 36 (B) (2004), 557-575.*

HUDA, N., NASER, J., BROOKS, G., REUTER, MATUSEWICZ, R. W., "CFD modelling of gas injections in top submerged lance smelting", *Proceedings, TMS 2009 Annual Meeting and Exhibition, Moscone West Convention Center • San Francisco, California February 15-19, 2009,*

MORSI, Y., YANG, W., CLAYTON, B. R., AND GRAY, N. B., "Experimental Investigation of swirl and non swirl gas injections into liquid baths using submerged vertical lances," *Canadian Metallurgical Quarterly, 39 (1) (2000), 87-98.*

NASER, J., ALAM, F., KHAN, M., "Evaluation of a Proposed and an Existing Dust Ventilation/Collection System in an Underground Mine Crushing Plant", *WSEAS Transactions on Fluid Mechanics, issue 3, volume 3, July, 2008*