DESIGN OF SECONDARY FUME CAPTURING HOOD FOR CASTING HALL OF A SiMn PRODUCTION PLANT

[1]Mehdi Kadkhodabeigi, [2]Dag Haaland

[1]R&D Department of Eramet Norway AS, PO Box 7465, Trondheim, Norway
[2]Eramet Norway Kvinesdal AS, PO Box 4484 Øyestranda, Norway
e-mails: [1]mehdi.kadkhodabeigi@erametgroup.com, [2]dag.haaland@erametgroup.com

ABSTRACT

Diffuse emissions which are generated during metal casting operation are known as important source of internal pollution in ferroalloys production plants. It has always been addressed as one of the main environmental issues in ferroalloys industries. In order to improve working environment in the casting area and hence the whole smelting hall, it is necessary to capture the generated fume in an efficient way at closet point to its source. Performance of a fume capturing hood for metal casting area can be influenced by several parameters such as the hood geometry, location of the hood regarding casting operational procedure, hall-wind effect, volume of diffuse emissions and off-gas suction capacity. Optimal design of a fume capturing system can be achieved through investigation of all the affecting parameters. In the present work a comprehensive study on all the above mentioned aspects for a fume capturing hood has been performed. Computational Fluid Dynamics (CFD) modeling as a powerful tool has been used as our scientific approach in the current study. As the first step a model which simulates generation and emission of the fume in the metal casting area of a plant has been developed. Temperature measurements both on walls and casting bed and velocity measurements of diffuse emissions and wind in the casting area have been used in the model. Results of the model are in good agreement with observations in the plant which prove validity of the model. Then in the next step using the validated model, performance of different designs for the hood, different emissions volume during casting operation, different tilting angles for the metal ladle, different off-gas capacities and different conditions for hall-wind in the casting area, have been evaluated. Based on the results obtained from the modeling work, an optimal design for the dust capturing hood has been obtained.

KEYWORDS: *Diffuse emission, fume capturing hood, metal casting operation, hall-wind, Computational Fluid Dynamics (CFD).*

1. INTRODUCTION

Metal casting operation is a main step in ferroalloys production process. Manganese has a relatively high vapor pressure, resulting in significant fume generation during tapping and casting of both ferromanganese (FeMn) and silicomanganese (SiMn) [2]. Therefore during casting of manganese alloys large volumes of secondary fumes are generated on the beds and released in the working environment. The secondary fumes are mostly composed of metallic oxides generated due to reaction between high temperature (i.e. 1500°C) molten metal and oxygen in the air. Fumes 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 [1].

Sustainable production in ferroalloys industry urges us to take care of environmental pollutions generated due to different operational processes. The trend is that legislation requires a minimum visible diffuse emissions and hence lowest opacity in the working environment. Continuous improvement of working environment in the plants as well as reducing diffuse emissions from different operations has always been considered as a goal in Eramet Norway AS. As a result of large investment on R&D activities focusing on the environmental issues in several years, a big improvement in the working environment of different plants in Eramet Norway AS has been obtained. The current work represents design of an efficient fume capturing hood over casting beds in SiMn production plant in Eramet Norway Kvinesdal AS (ENK).

In order to make an efficient hood design it is important to consider different physical aspects as well as operational restrictions into account. Experience from different industrial designs shows that traditional engineering design techniques are not adequate to allow novel designs to be developed and assessed. In contradiction to the traditional engineering design computational fluid dynamics (CFD) modeling can overcome the mentioned limitations. It can be used as a powerful tool to predict fluid flows and heat distribution in different parts of a system and with high accuracy for different geometries and operational conditions. Therefore CFD modeling has been employed in the present study.

2. SECONDARY FUME GENERATION DUE TO CASTING OPERATION

In ENK the SiMn metal which is tapped from furnaces is casted on a series of sand beds which are located in a casting hall. The outer surface of the each bed is covered with sand which is divided into several plates of rectangular shape. The molten metal is poured from ladle into a cylindrical vessel called 'Cola box'. The metal then flows into the mentioned plates over the sand bed. Figure 1 shows the casting hall of the SiMn production plant and distribution of diffuse emissions in the hall due to casting operation.

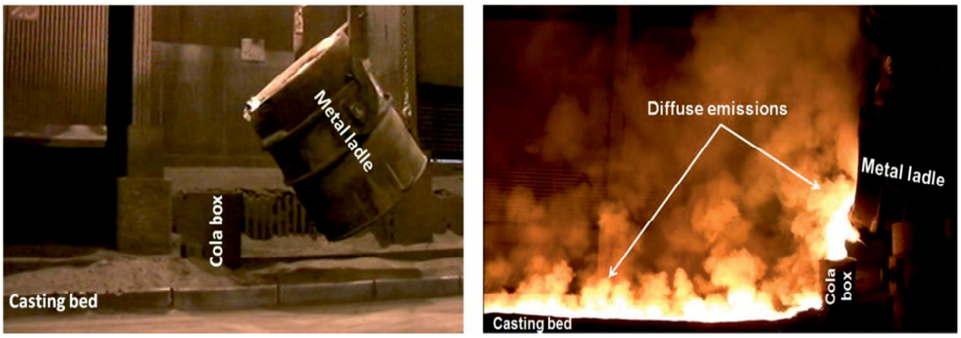

Figure 1: The secondary fume generation due to casting of metal in a SiMn production plant

3. DESIGN OF SECONDARY FUME CAPTURING HOOD

Results of several studies show that the secondary fume capturing will be more efficient if the extraction point is as close as possible to the fume generation source [2, 3, 4]. However operational procedure, for example need for having enough working space for picking up cooled metal plates by front head loaders in our case, can cause some restrictions regarding this aspect.

Another important aspect in design of an efficient secondary fume capturing hood is to minimize the volume of fresh air which is extracted by the hood [8]. Suctioning of large volume of fresh air into the hood may cause reduced fume capturing and hence reduced hood efficiency. It will then lead to need for having more extraction capacity to solve the problem. Plant observations show that during casting operation the largest amount of secondary fumes are generated from the melt surface in the ladle, open surface of the cola box and the first three plates of the sand bed. Since installation of a fume capturing hood over the whole sand bed area requires large extraction capacity to be provided by an intake fan, therefore we have just focused on capturing of the secondary fumes from the above mentioned zones.

4. DESCRIPTION OF THE CFD MODEL

In order to simulate the diffuse emissions in the casting area as well as evaluating different designs of the hood a CFD model of the considered system has been developed. The model is steady state, 3D and one phase but it includes different species for sake of fume simulation. The model has been made using ANSYS FLUENT 14.0 where model's governing mathematical equations including turbulent fluid flow, convective and radiative heat transfer, mass and species conservations are solved.

4.1. Geometry of the model

Geometry of the model has been built using ANSYS DesignModeler software and based on detailed industrial size of the system. Since during pouring of metal from the metal ladle into the cola box tilting angle of ladle, angle between the ladle axis and horizontal axis, varies between 0 to 90 degrees, therefore to different configuration for the ladle has been considered in the model. Figure 2 shows different geometrical configurations of the system used in the model.

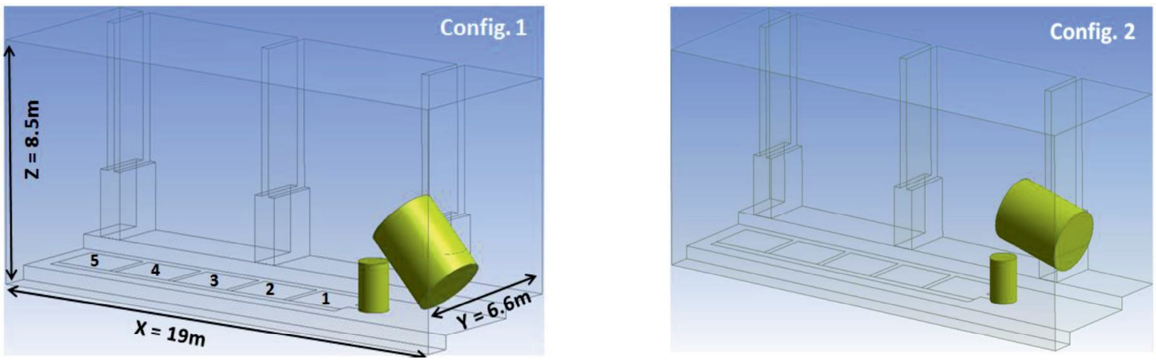

Figure 2: Geometry of the casting bed
and different ladle configurations during pouring of metal

Performance of different designs for the secondary fume capturing hood was investigated in form of several case studies using the model. The initial design was a hood which locates over the sand beds and it includes either two or three off-gas outlets as it has been shown in figure 3. It should be mentioned that the total surface area of the off-gas outlets are equal in both designs.

In order to study the impact of reducing the volume of fresh air which is sucked into the hood, closing one side of the hood using a vertical sidewall perpendicular to the sand bed, as it is shown in figure 4, was studied. Finally performance of a non-symmetrical hood geometry including the same vertical sidewall as the previous design was evaluated using the model. Geometry of this hood with having two and three off-gas outlets, with the same surface area as the previous designs, is given in figure 5.

4.2. Boundary conditions

In the CFD modeling approach setting up initial boundary conditions is required to start solving the involved mathematical equations involved in the model. The extraction capacity of the fume capturing hood in all case studies has been sat up to be 75000 Nm^3/hr.

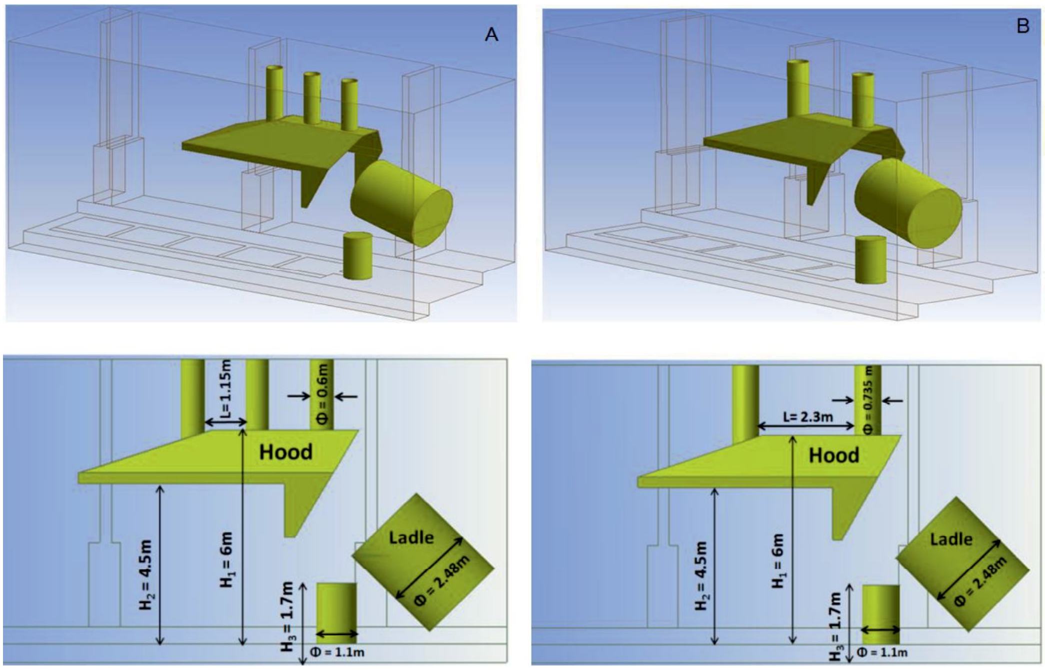

Figure 3: Detailed geometry of the hood with having two and three off-gas outlets

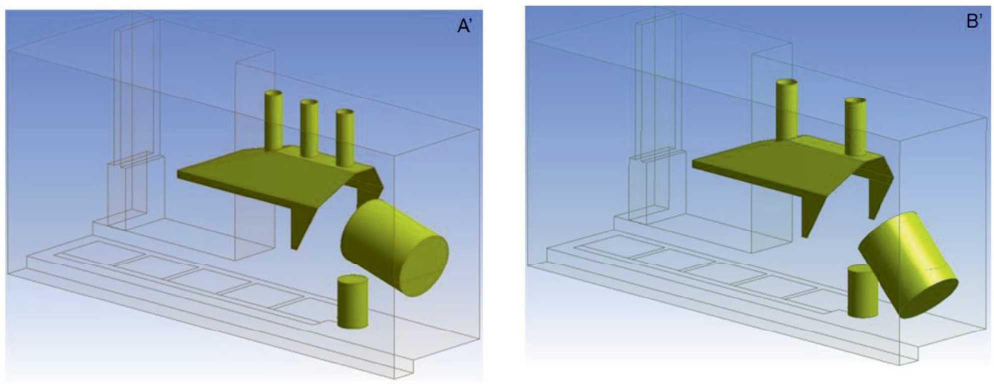

Figure 4: Geometry of the hood with having two and three off-gas outlets and closing one side of the hood using a vertical sidewall perpendicular to the sand bed

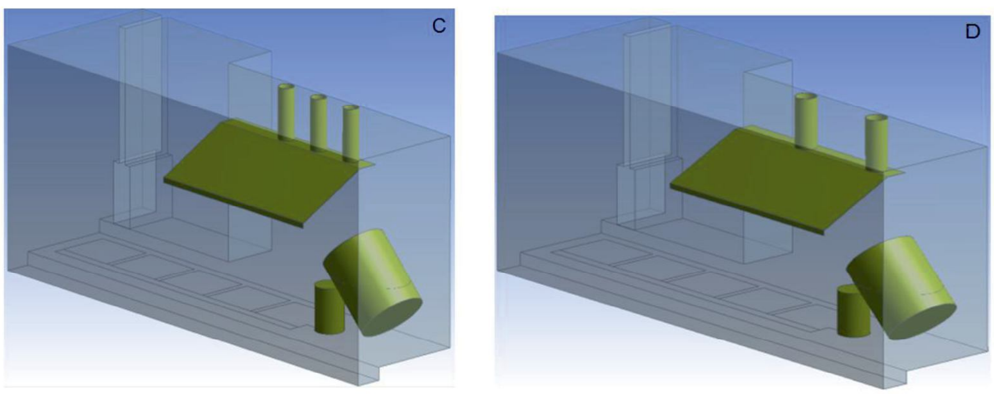

Figure 5: Geometry of the non-symmetrical hood with having two and three off-gas outlets and closing one side of the hood using a vertical sidewall perpendicular to the sand bed

The fume generation rate from different surfaces, especially from the ladle and the cola box, can vary depending on several parameters such as metal composition and its temperature. Therefore two different conditions, based on volume of the generated secondary fumes, with low emission (LE) and high emission (HE) have been simulated using the model. The secondary fume concentration from different surfaces has been considered to be 60 %. Since the plant measurements shows higher temperatures on the first two plates of the sand bed, therefore more fume generation from these two plates has been considered. The boundary conditions considered for fume velocity and temperature on different surfaces have been given in table 1.

Table 1: Temperature distribution and velocity of the secondary fumes generated from different surfaces in the model

Case study	**Melt surface**	**Cola box**	**Sand bed 1**	**Sand bed 2**	**Sand bed 3**	**Sand bed 4**	**Sand bed 5**	**Plates connection**
Velocity (m/s) Low Emission (LE)	1	0.5	0.3	0.3	0.2	0.2	0.2	0.3
Velocity (m/s) High emission (HE)	2	1	0.3	0.3	0.2	0.2	0.2	0.3
Temperature (°C)	1400	1400	1230	1230	1130	1130	1130	1230

4.3. Numerical solution

The governing equations for mass, momentum and energy [6] together with equations of turbulent flow (k-ω) [7], species transport and radiative heat transfer (DTRM) are solved in ANSYS FLUENT 14.0 simultaneously [5]. The governing equations are then discretized explicitly. Pressure and velocity are coupled using the SIMPLE technique. The mesh used for the CFD simulations was a hexahedral mesh comprising 1200,000 elements with smaller cells located close all walls in the system and around hot surfaces including the melt surface in the ladle, cola box and all five plates of sand bed.

5. RESULTS AND DISCUSSIONS

The first case study was to simulate the current condition of the secondary fumes emission in the hall due to casting operation. Figure 6 shows results of the model for the fumes concentration in the hall. As it is seen from this figure the fume has tendency to move towards the sidewall and hence the fume concentration is higher in this region. This result is in good agreement with observations in the plant which can prove validity of the model.

Comparison between results of the model and plant observations for the fumes in the casting area has been presented in figure 7.

In the next step the developed CFD model were used to simulate performance of different hood designs under different operational conditions. Pathlines of the secondary fumes using hood designs B and D have been shown in figure 8. It is seen from this figure that using these hood designs there are some fumes which cannot be captured by the hood. Part of the fumes which is generated from both plates number 4 and 5 is not captured by the hoods. The main reason is that the hood has not covered those plates. Using hood design B a portion of the fumes generated form the initial plates also is not captured. This is due to tendency of the fumes for flowing towards the sidewall. This problem is solved connecting the hood to the sidewall as it is seen from performance of the design D.

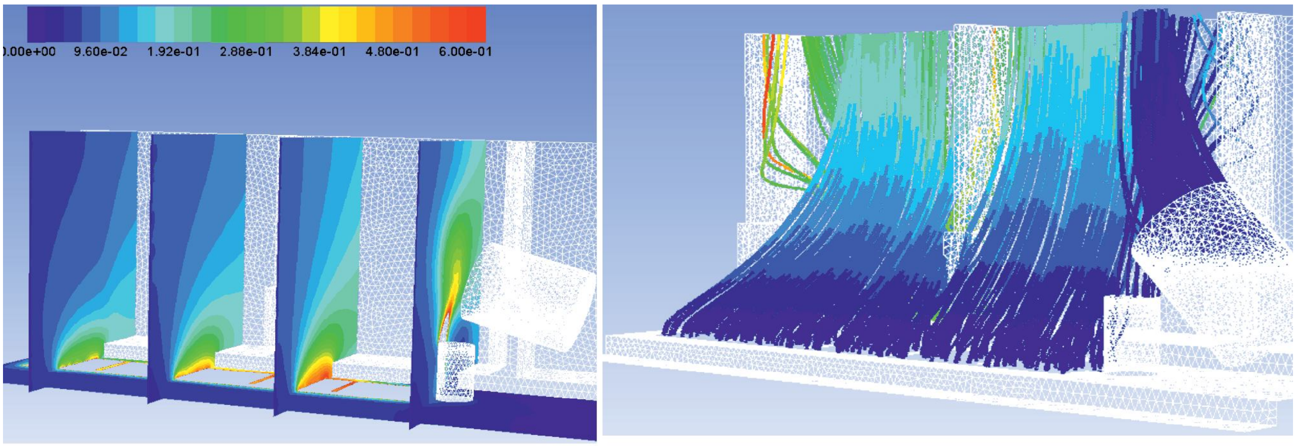

Figure 6: Fume concentration (left) and pathlines of diffuse emissions (right) in the hall showing the fumes tendency for flowing towards the sidewall

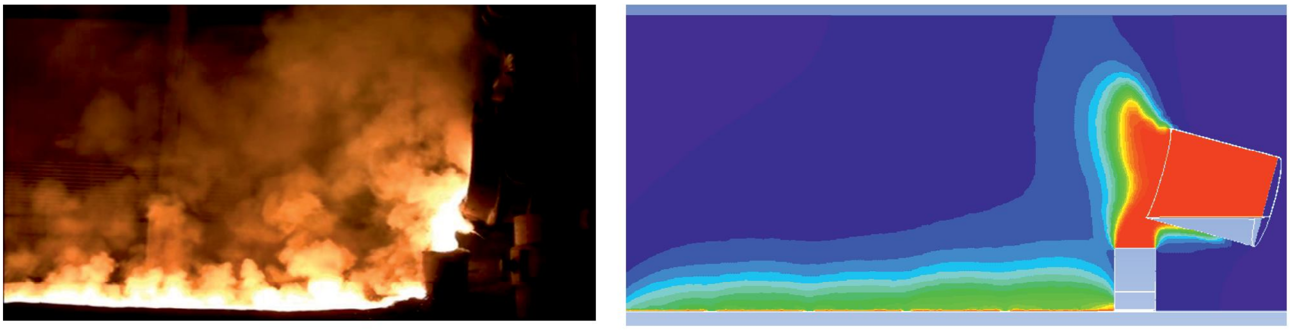

Figure 7: Comparison of diffuse emissions in hall with model's results during casting operation

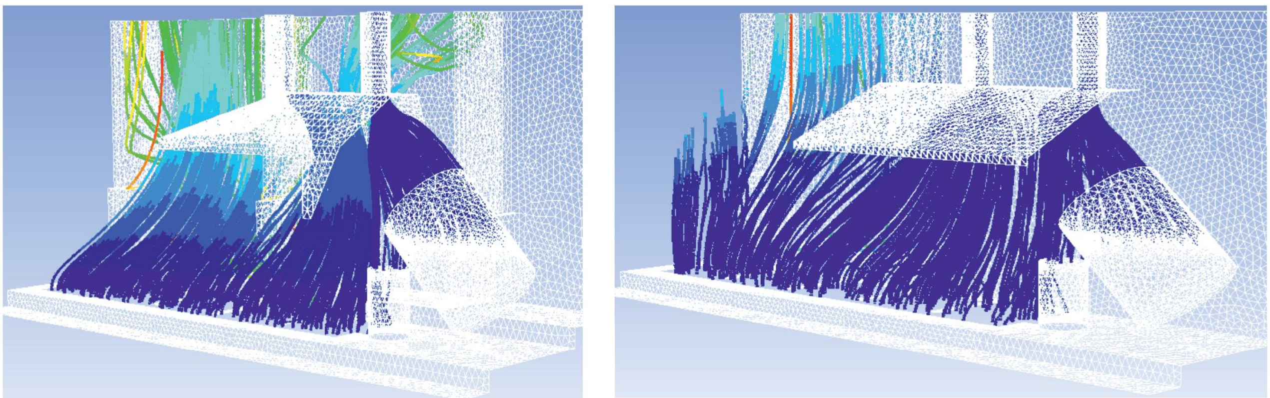

Figure 8: Pathlines of the secondary fumes in the hall using hood designs B (left) and D (right)

In the present work efficiency of the hood has been defined as ratio between the volume fraction of the fumes captured by the hood and the total volume of fumes released from different surfaces in the model. Efficiency of different hood designs has been summarized in table 2.

The results of the model show that among different designs the secondary fume capturing hood D has the highest efficiency. The results show that in the high emission condition the hood's efficiency is normally higher than the low emission condition. This can be explained through noticing to the vertical distance between the sand bed, as source of secondary fumes, and the hood.

In the high emission condition vertical component of the fume velocity is higher. Therefore the fumes move towards the hood and they are captured more efficiently.

Table 2: Efficiency (%) of different hood designs under different operational conditions

Case study	A	A'	B	B'	C	D
Config.1 - LE	69	72	69	72	75	82
Config.1 - HE	76	85	71	77	80	86
Config.2 - LE	69	71	69	71	74	80
Config.2 - HE	75	76	70	73	77	82

Temperature distribution due to radiative and convective heat transfer on both the sidewall and the hood body has been presented in figure 9.

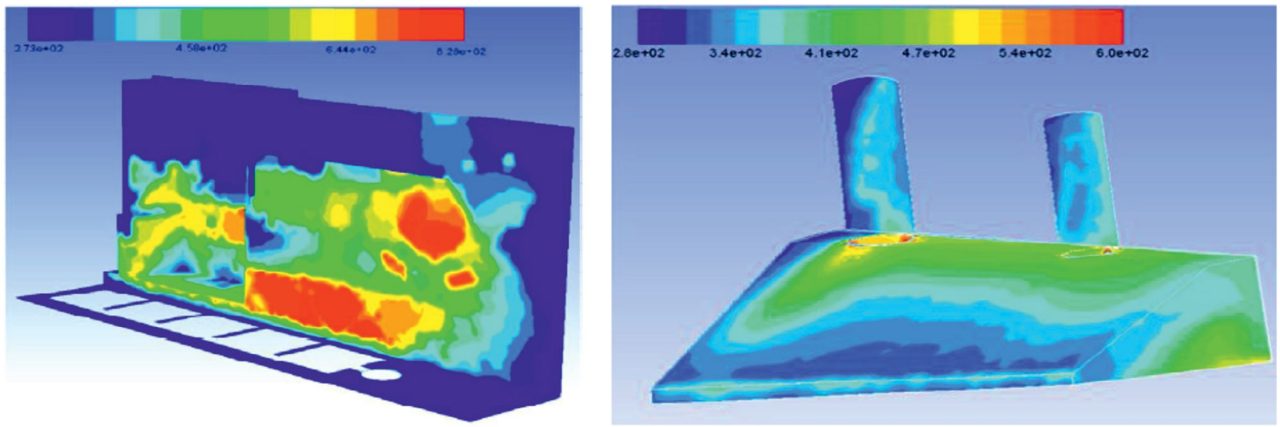

Figure 9: Temperature distribution both on the sidewall (left) and the hood body (right) using hood design D

As it was mentioned earlier the sand beds in ENK are located in the casting hall. Although the effect of wind on the fumes flow has been then to a large extent reduced however the well-known hall-wind effect has been studied in this work. In order to simulate this effect correctly, wind velocity from different directions in the hall and especially close to the sand beds was measured. The highest velocity was around 1.1 m/s in direction parallel to the sand beds. This was used then as a basis to investigate the hall-wind effect on the performance of the hood. It should be mentioned here that the effect of hall-wind only on performance of the hood design D was evaluated in this work. Figure 10 shows the fumes concentration on a vertical plane perpendicular to the sand bed with different wind directions parallel to the sand bed. Summary of the results have been given in table 3.

The results of model show that in the situation where the wind direction is from left to the right the hood will have a little higher efficiency comparing to the situation where there is no wind in the hall. As it can be seen from figure 10 in one hand some fumes from the melt surface in the ladle will not be captured by the hood but in the other hand a large volume of fumes from plates 4 and 5 are pushed towards the hood and hence are captured. Therefore the efficiency increases. The results also show that if wind blows from right to the left direction then it will reduce the hood's efficiency significantly since a large volume of fumes from the plates is blown out from the zones located under the hood and hence the hood's efficiency decreases.

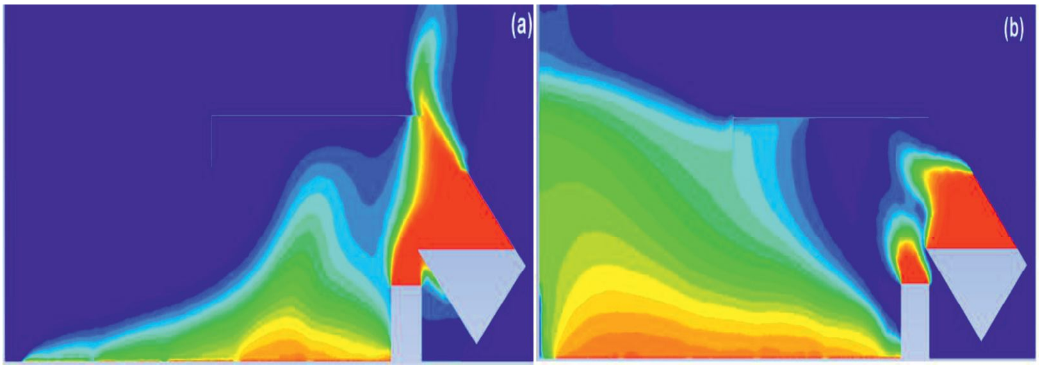

Figure 10: The effect of hall-wind on the secondary fumes flow in the condition where wind (a) blows from left to the right and (b) from right to the left

Table 3: The impact of hall-wind effect on the efficiency (%) of hood design D

Case Study	Efficiency of hood design D (%)
Config.1 – LE: wind direction from left to the right	89
Config.1 – HE: wind direction from left to the right	87
Config.1 – LE: wind direction from right to the left	56
Config.1 – HE: wind direction from right to the left	67

6. CONCLUSIONS

Performance of different hood designs for capturing the secondary fumes generated due to metal casting operation in a SiMn production plant has been evaluated using CFD modeling technique. First a model for current situation of the secondary fumes emission from the beds, without hood, has been developed. Good agreement between the modeling results and plant observations proves validity of the model. This model has then been further developed to investigate efficiency of different designs for the secondary fume capturing hood in different operational conditions in the casting area. As a result of the modeling process an optimal design for the hood has been obtained. In addition to this the model's results show that the hall-wind depending on its direction can decrease the hood's efficiency significantly.

7. REFERENCES

[1] Witt, P.J., Solnordal, C.B., Manzoori, A.R., Finn, S., Pluta, J., Namavari, H., Niknejad, E. and Davari, M., "Reducing Fume Emission from Smelting Vessels Using Combined Cfd and Engineering Modelling", Proceedings of Sustainable Developments in Metals Processing, 2005, 381-392.

[2] Els, L., Coetzee, C. and Vorster, O., "Design of tapping fume extraction system for ferroalloy furnaces", Proceedings of Twelfth International Ferroalloys Congress, Sustainable Future, 2010, 131-141.

[3] Kadkhodabeigi, M., Tveit, H. and Berget, K. J., "Silicon process – new hood design for tapping gas collection", Proceedings of Twelfth International Ferroalloys Congress, Sustainable Future, 2010, 109-119.

[4] Ravary, B. and Grådahl, S., “Improving environment in the tapping are of a ferromanganese furnace”, Proceedings of Twelfth International Ferroalloys Congress, Sustainable Future, 2010, 99-107.
[5] Fluent 6.3 user’s guide, 2003, 1077-1080.
[6] Patankar, S. V., “Numerical Heat Transfer and Fluid Flow”, Hemisphere Publishing Corporation, New York, 1980.
[7] Wilcox D. C., “Turbulence modeling for CFD, DCW Industries”, Inc., La Canada (1998), California, 1998.
[8] Goodfellow, H. and Tähti, E., “Industrial Ventilation Design Guidebook”, Academic Press, California, 2001.