

DEVELOPMENT OF A CFD MODEL FOR IMPROVEMENT OF AN INTERIOR ENVIRONMENT IN METALLURGICAL INDUSTRIES: MODELING¹Balram Panjwani, Helge Midtdal, ²Benjamin Ravary, ²Mehdi Kadkodabeigi, ²Roy Nordhagen¹SINTEF Material's and Chemistry, Norway; e-mail: balram.panjwani@sintef.no²Eramet Norway As, Norway; e-mail: Benjamin.ravary@erametgroup.com**ABSTRACT**

The compliance with existing regulations regarding the interior work environment in metallurgical industries is of major importance. Of major concern inside the smelters is the quality of the air such as temperature, humidity, dust contents and concentration of environmental and health-related adverse compounds. Ventilation is one of the best ways to improve the quality of the interior environment through more or less controlled exchange of air. The present study focuses on the role of natural ventilation system for improvement of the interior environment. In a smelter, open inlet areas are provided for fresh air supply and transportation purposes, and outlet areas on the roof are provided to discharge warm and polluted gases to the exterior environment. The quality of the air in the factory building can be improved by optimizing the location and size of the inlet and outlet areas. In the present study a RANS-CFD model of the factory building is developed, and the model is calibrated for the baseline condition with available measurement. The velocity and temperature profiles at the outlet compares well with performed on-site measurements. The calibrated model is then used for assessing the effect of closing the inlet/outlet areas on flow distribution inside the building and its consequences on interior environment. The study showed that reducing the inlet area reduces the total mass flow rate and increases local temperatures. Furthermore, closing one inlet opening leads to the increased velocity in another inlet opening even though the total mass flow is reduced. Closing part of the open roof areas leads to further reduction of the total mass flow and thereby a slight increase in temperature. The model developed to generate the presented results can now be utilized to optimize proportions of the inlet/outlet areas and their position to efficiently improve the interior environment.

1. INTRODUCTION

Natural ventilation is an exchange of fluid between the interior of the building and its exterior environment when the flow is produced by naturally occurring pressure differences. The purpose of ventilation is to remove air contaminated with excess heat, humidity, carbon dioxide, toxins and other unwanted substances, and to provide clean air at a comfortable temperature and humidity. In the design of most industrial and metallurgical buildings, a minimum amount of outdoor intake is generally required, also when wind is absent. This requires a proper design of openings and consideration of other parameters such as building thermal properties and outdoor thermal environment. The unwanted toxins, dust and humidity is the byproduct of different industrial processes in metallurgical plants. In metallurgical buildings there are many processes such as casting and tapping which are responsible for fume/dust generation. The working conditions are largely affected by the smoke and fine dust particle distribution inside the building. Understanding and controlling the distribution of the fume/dust is an important environmental challenge. Dust is formed locally at the location of the responsible processes, and thus affects the working environment of the workers assigned to these processes. In addition, a significant concentration of

fumes in the hall results in emission to the exterior environment, so called diffuse emissions. Stricter limitations for the diffuse emissions are expected in metallurgical industries.

Studies on natural ventilation have been conducted in the past. Hunt and Linden [4, 5] studied the effect of an opposing wind on the stratification and flow produced by a buoyant plume rising from a heat source on the floor of a ventilated enclosure. In the absence of wind, Linden et al. [11] show that warm air rises as a turbulent plume above the heat source and stratifies the interior air into two homogenous layers – a warm upper and cooler lower layer. The warm upper layer drives flow out through the high-level opening(s) which is replenished by air at ambient temperature entering the space at low level. CFD have been applied to predict passive airflows in buildings driven by internal heat sources with and without wind. Cook [3] carried out CFD studies of the wind assisted natural ventilation problem, they used an external flow domain/surrounding which enabled the airflow through the inlets and outlets to be modeled explicitly without the need for boundary conditions at the openings. High Reynolds number flow around an isolated building creates positive pressures on the windward side and negative pressures in the leeward. By suitably locating openings that connect the interior and exterior environments this wind-induced pressure difference ΔP may be harnessed to drive a ventilation flow through the building. External flow around building is highly complicated involving severe pressure gradients, swirls, separation and reattachment. Recently Allocca et al [2] carried out ventilation studies of a building and they addressed the importance of having the external flow domain/surrounding for predicting the correct wind assisted natural ventilation. Although wind is an important component which is best captured by including the exterior flow in the calculation domain, the exterior domain is often neglected. This is caused by the added computational cost of including an exterior domain, especially critical for buildings of complex geometry (i.e. many details) or for parametric studies where many CFD simulations are required. CFD models without an external flow domain/surrounding requires well posed boundaries at the inlet and outlet surfaces. A CFD study without an external domain has been carried out by Cook, Ji and Hunt [8]. The boundary conditions were applied directly at the openings, this approach considerably reduced the computational time. Results with this approach were very encouraging with only minor discrepancies between predictions and measurements. A similar approach is used in the present study, where pressure boundary conditions are applied on the openings of the building.

The present study focuses on the role of natural ventilation system for improvement of an interior environment of a given furnace building. This will establish an understanding between the operating condition of the furnace building (gates opening, roof location, sizes of the opening, initial air temperature etc) and the quality of the air and fume/dust distribution inside the building. Distribution of the finer particles inside the building depends on the local buoyancy force, velocity magnitude and velocity direction. Therefore it is important to understand these effects for a generalized geometrical configuration with different openings. Finding a generalized geometrical configuration, which is representative for all metallurgical plants, is a difficult task. Therefore we have used the Sauda factory building operated by Eramet Norway. Sauda is a municipality and industrial town in the south west Norway. Eramet Norway has a Manganese plant; they supply Manganese alloys for the international market. Eramet Norway and other Ferro industries are environment conscious companies. They are constantly working on improving the working environment inside and outside the factory buildings. The motivation of the present study is to address some of the issues towards a better working environment.

Modeling of the Sauda factory building will unveil the general principle of natural ventilation, dust and flow distribution, which will be applicable for other metallurgical buildings. These buildings are mainly naturally ventilated, and natural ventilation is governed by the inlet and outlet surface areas and locations. These buildings are also provided with local mechanical ventilations, which are not accounted for in the present study. Performing experiments to assess the effect of

building modifications is an option, but this is often expensive due to construction costs. Therefore a numerical simulation based on computational fluid dynamics (CFD) is a cost effective and powerful tool for such assessments.

2. GEOMETRY AND MODELLING

A CFD model of the air flow distribution in a given factory building was established based on the information provided by Eramet Norway. ANSYS Fluent v12.1 [1] was used in the present study. ANSYS Fluent is based on the Finite Volume Method (FVM). The steady state modeling approach solving for continuity, momentum, energy, and radiation equation was applied. The discretization scheme for momentum and energy equation is second order accurate. The flow in the building is turbulent. Based on the previous study [9, 10], we have chosen the $k-\omega$ model for turbulence modeling. The gas density was temperature dependent in order to include natural convection.

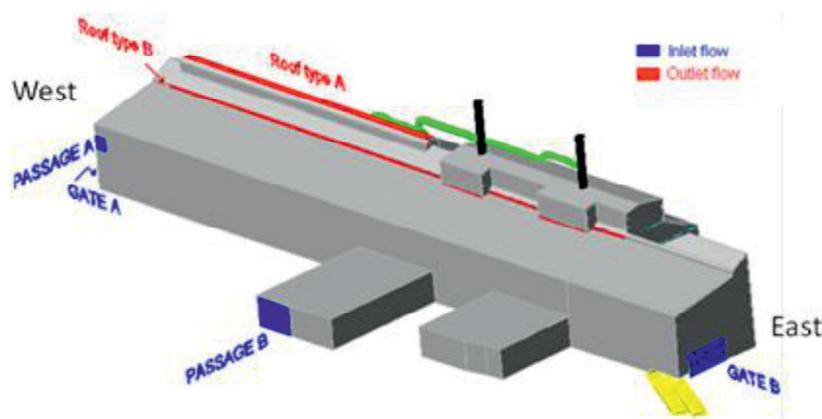


Figure 1: Schematic of the Sauda Building

Figure 1 shows the schematic of the building model, the length, height and width of the building are approximately 200 m, 25 m and 80 m respectively. Due to the large dimensions of the building, geometry simplifications have been made. Tapping areas, casting area, refining areas and slag areas have been replaced by simpler geometries. Also, influences of the mechanical ventilation are not accounted.

A CFD geometry model was generated using commercial code ANSYS Design Modeler. The CFD model geometry is flexible and it is possible to perform parametric studies on the model such as location of ladles, new inlet and outlet openings etc. In a baseline model, air enters from four inlet surfaces (2 Gates (A and B) and 2 passages (A and B)). The inlet surfaces surrounded by the open atmosphere are called gates and the inlet surfaces surrounded by other buildings are called passages. The flow at the gates is affected by the wind; however the flow at passages is not influenced by wind. Air discharges from the two outlet surfaces (Roof A and Roof B) as shown in figure 1. A computational mesh of the geometry was created using ANSYS Mesh. The model initial mesh was 6.5 million tetrahedral cells, which was converted into polyhedral mesh of 1.8 million cells. Natural convection is a dominating mechanism for the problem of interest; therefore a highly resolved grid is needed on the walls and especially around the ladles. The grid was refined close to almost all walls and all the ladles. In addition, a boundary layer mesh was generated. The Y^+ closed to the ladles is 85. The grid dependency study was not performed because of computational cost. As mentioned, the polyhedral mesh was generated from the tetrahedral mesh. Then it is also not

possible to use grid adaptation function of the ANSYS Fluent [1]. In the present study we have used the model without an exterior surrounding as shown in figure 1.

Inside the factory building the working fluid is mainly air but it also contains particles and the other component such as CO, CO₂ etc. In this investigation we have only considered air as the main component and the other components are neglected. The main heat sources are 6 ladles, refining area, casting area and the tapping floor. In the present study the ladle top surfaces, which are mainly liquid metal are maintained at 1400°C and the outer ladles wall temperature is maintained at 250°C. The temperatures on the other walls were taken from the IR pictures. The boundary conditions applied on all the walls and ladles are constant wall temperature, which is not appropriate in reality. The constant wall boundary condition does not allow any contribution from radiation. However, radiation is one of the major contributors for metallurgical buildings. For instance the walls which are facing towards the hot surface such as ladles etc, will be having higher temperature than the averaged temperature. Nevertheless, the main objective of the present study is to assess the flow pattern with different gate opening/closing.

3. RESULTS AND DISCUSSIONS

The first case (Case-0) is a baseline case as shown in figure 1, for which experimental data were available. The Case-0 is used for calibrating the CFD model with measurements. The boundary conditions applied in the calibration procedure were velocity inlets at the gates and pressure outlets at the roof openings as measured during the experiments. The CFD results (velocity and temperature) were compared with measurement data at the Roof B. Figure 2 shows the temperature distribution over the Roof B. The position "0 m" is at the west end of the building as seen in figure 1. It is observed that the computed temperature on the Roof B is lower all over the roof compared to the measurements. This could be caused by the approximated boundary conditions (constant wall temperature). One more CFD simulation was carried out where temperature on the ladles and building wall was increased. The surface temperatures at all interior walls inside the building were increased by 10°C, and on the temperature on ladles walls were increased by 100°C. This case is referred to as Case-1.

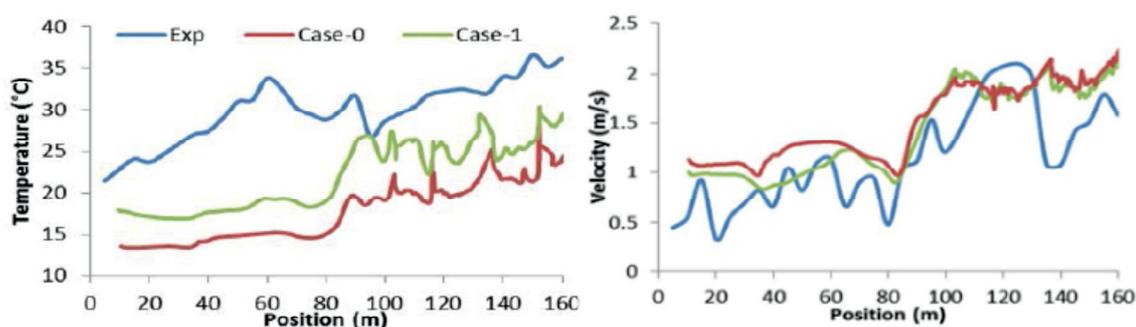
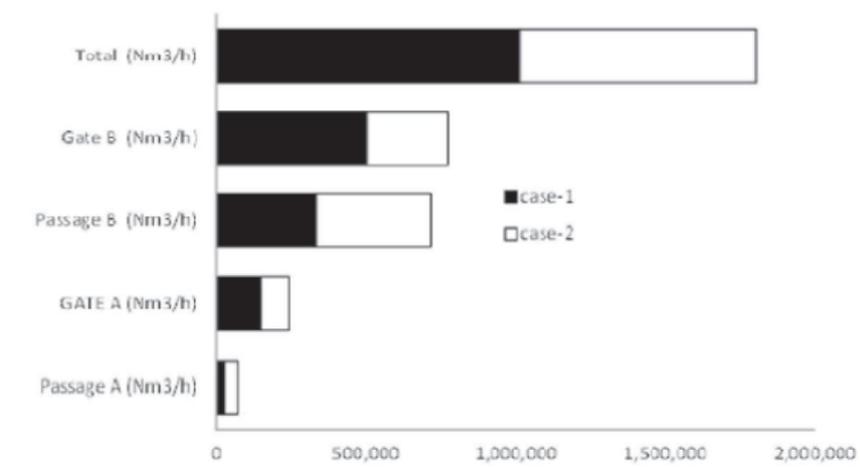


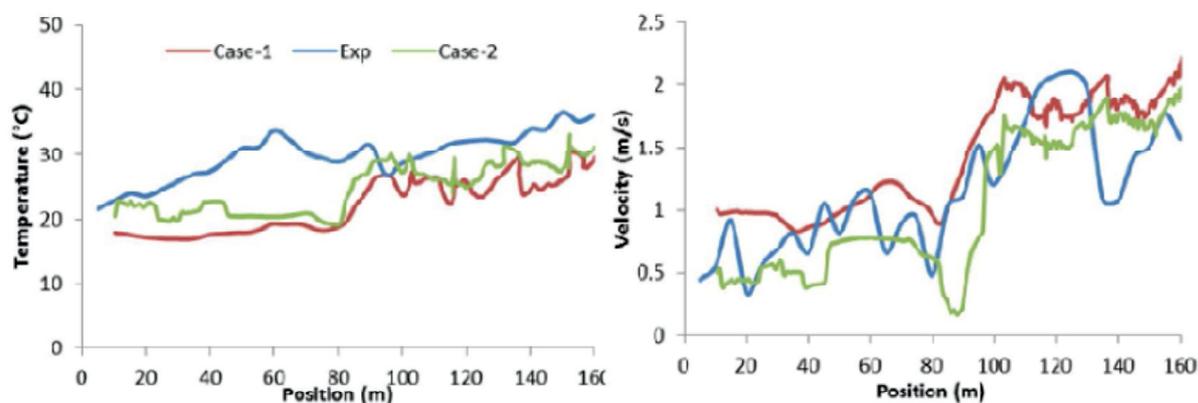
Figure 2: Temperature and velocity distribution at the Roof B

Figure 2 shows the temperature distribution over Roof B from CFD for Case-0 and Case-1 compared with experimental data. It can be seen that the temperature (Case-1) has increased all over the roof and it is now closer to the measured values. It is also observed from the figure 2 that the experimental value of the temperature is relatively high on the west side of the Roof B. However there is no heat sources in this part of the building responsible for generating this much heat. Figure 2 shows the velocity profile of the Case-0 and Case-1 compared with experiments at Roof B. It can be observed that computed velocity is consistent with the experiments. However

there is still some disagreement between CFD and measurements. One of the reasons could be approximated wall boundary conditions on interior and ladles walls. Nevertheless, it can be concluded that with increased temperature on the boundaries (Case-1), the predictions are better compared with experiments.



With this tuning exercise, a calibrated model is defined for assessment of flow conditions with different inlet and outlet configurations. Velocity inlets were used during the model calibration.



However, while studying the effect of closing and opening the gates, the velocities are unknown on the inlet surface. The opening/closing of the gates/roofs changes the velocities at the inlet surfaces which need to be measured. In absence of measurements, the best approach is to use wind velocity profile at an exterior/surrounding. The CFD simulations with an external flow domain enabled the airflow through the inlets and outlets to be modeled explicitly without the need for boundary conditions at these locations. However accounting the external domain is computational demanding. A compromise is to use pressure inlets directly at the gates. This does not account for wind, but will show the general effect of closing and opening the gates. The use of pressure boundary conditions at the inlet surface reduces computational time considerably.

Prior to using pressure boundary condition for parametric model study a validation test case was carried out. A CFD study same as Case-1 was performed except using velocity boundary condition, a constant pressure boundary condition was used. This case is referred as Case-2. The

pressure on the both gates and passages was set equal to atmospheric pressure. Figure 3 shows the flow rate at gates, passages, and total flow rate for Case-1 and Case-2.

With pressure boundary condition (Case-2), the mass flow rate from the Passage A and Passage B is increased by 14 % and 46 % respectively. That is because the passages are surrounded by large building where flow is not influenced by wind. However the mass flow from gates A and B is reduced by 36 % and 46 % respectively. Because the gates are surrounded by an open atmosphere where flow is mainly influenced by wind intensity and direction. The overall mass flow rate with pressure boundary condition is reduced by 22 %. This is because pressure boundary does not account for wind component. Figure 4 shows the temperature and velocity distribution along the Roof B. Temperature profile for Case-2 is nearly the same as for Case-1, which indicates that the boundary condition on inlet surfaces does not have much effect on the energy transfer. However there are some differences in velocity profile, the velocity is lower for Case-2 compared with Case-1. The overall mass flow reduction caused the velocity reduction on the roofs.

A main objective of the present study was to understand the effect of closing and/or opening gates and/or passages on the flow conditions in the furnace hall. When closing any of the gates or passages in the building, the velocity on other gates and passages also changes. In the present work a CFD study with closure of Gate B was carried out. This case is referred to as a Case-3. Case-3 has the same setup as Case-2, except Passage B was closed as shown in figure 5. The fume and air distribution also depends on the sizing of the outlet roofs. Therefore a case with reduced outlet area was investigated. This case is referred as Case-4 and shown in figure 6. Here Roof A is closed and Roof B was partially closed. Atmospheric pressure was set on the inlet and outlet surface. Table 1 shows an overview of all the cases that were simulated in the present study.

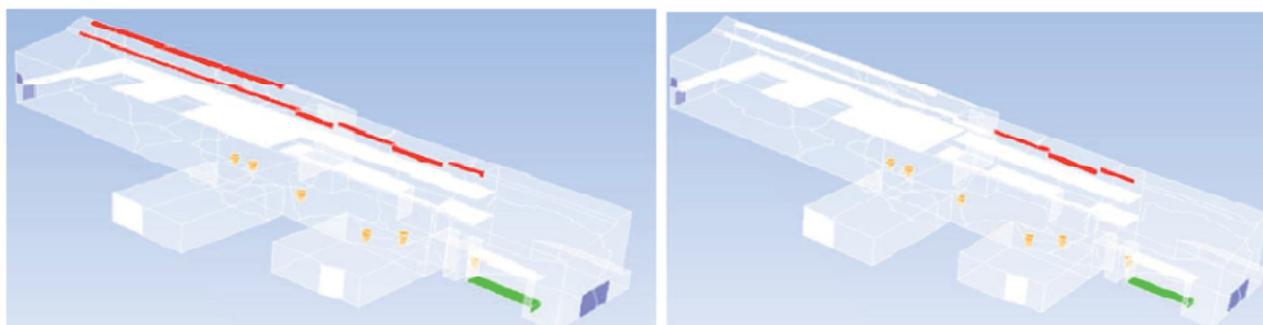


Table 1: Description of the cases

Cases	Description
Case-0	The base line case
Case-1	The baseline case with updated temperature boundary condition
Case-2	Same as Case-1, but with pressure boundary condition on the inlets and outlets
Case-3	Same as Case-2, but with reduced inlet area (Passage B is closed)
Case-4	Same as Case-3, but with reduced inlet and outlet areas (Passage B and Roof A is closed and partially Roof

Figure 7 shows the velocity and mass flow rate from the gates, passages and roofs for the Case-2, Case-3 and Case-4. It is observed that by closing the Passage B (Case-3), total flow rate is reduced by 13 %. It can also be seen that the mass flow rate at the Gate A and Gate B is increased by 89 % and 66 % respectively compared to Case-2. The mass flow rate from Passage A is increased by 45 % compared to Case-2.

One of the interesting results to be noticed here is the distribution of outflowing air between Roof A and Roof B. The simulations show that around 66 % of polluted air exits from Roof A and the remaining 34 % exits from the Roof B. When the factory was build, there were also furnaces in the west side of the building. It was therefore appropriate to have an outlet in the roof on this side. Now, when these furnaces are taken away, one could ask the question if these openings are necessary.

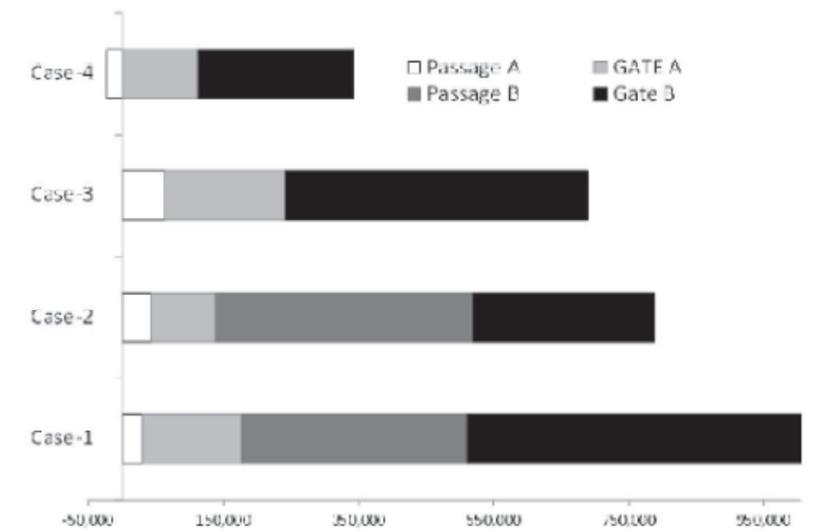
In Case-4 the mass flow rate from Passage A is negative, which indicates that a part of the air from Gate A is going out of Passage A. It is also observed that the mass flow from Gate A is increased by 19 % and from the Gate B it is reduced by 15 % compared to Case-2. The total mass flow rate for Case-2 is 788 200 Nm³/h, for Case-3 it is 689 300 Nm³/h, and for Case-4 it is 317 800 Nm³/h. That means for Case-3 the mass flow rate is decreased by 13 % compare to the Case-2, and for Case-4 the mass flow rate is decreased by 60 % compared to the Case-2.

The results for the CFD model is compared with a theoretical model [6, 7, 12]. According to the model the flow rate is governed by following type of theoretical relation.

$$q^3 = (C_d A^*)^2 2gh \cdot \frac{E}{\rho C_p}$$

$$A^* = \frac{A_{in} A_{out}}{\sqrt{A_{in}^2 + A_{out}^2}}$$

where, q is rate of flow through the building, C_d is discharge coefficient, h is height between inlet and outlet surfaces, E is released heat inside the building, A_{in} is total inlet area, A_{out} is total outlet area, C_p is specific heat capacity of the working fluid. As we see from the equation, the flow rate will decrease when the effective area (A*) is reduced while keeping other parameters fixed. For Case-3 the inlet area is reduced by 46 %.



Theoretically this gives a reduction in flow of 26 %, while the CFD calculation gives a reduction of 13 %. For Case-4 the inlet area is the same as in Case-3, but the outlet area is reduced by 68%. The reduction in flow is estimated to 46 % by the theory and 60 % by CFD. Theoretical and the numerical calculations are qualitatively consistent and predict the same trends. Figure 8 show the temperature and velocity profiles of the Case-2, Case-3 and Case-4. It is observed that the

difference in velocity and temperature is not significant between Case-2 and Case-3 in the east side of the building. In the west side, the velocity is reduced and this will result in a more wind sensitive opening. Cross wind can more easily penetrate into the building. The temperature slightly increases with decrease in mass flow rate. The velocity for the Case-4 is much higher than the Case-2 and Case-3. That is because the energy transfer is uniform in all cases; however velocity varies with the area and mass flow rate. In absence of wind, the effect of buoyancy dominates and for the cases shown, the buoyant layer is able to drive a flow out through the upper opening to the surroundings. Wind produces additional pressure differences and these may have quite different effects on the windward and the leeward sides of the building. Case-2, Case-3 and Case-4 fall under the category where the flows are driven by the hydrostatic pressure differences caused by density differences between the exterior and the interior fluid. They show some major changes in the flow conditions within the building as a result of reducing the inlet and outlet areas.

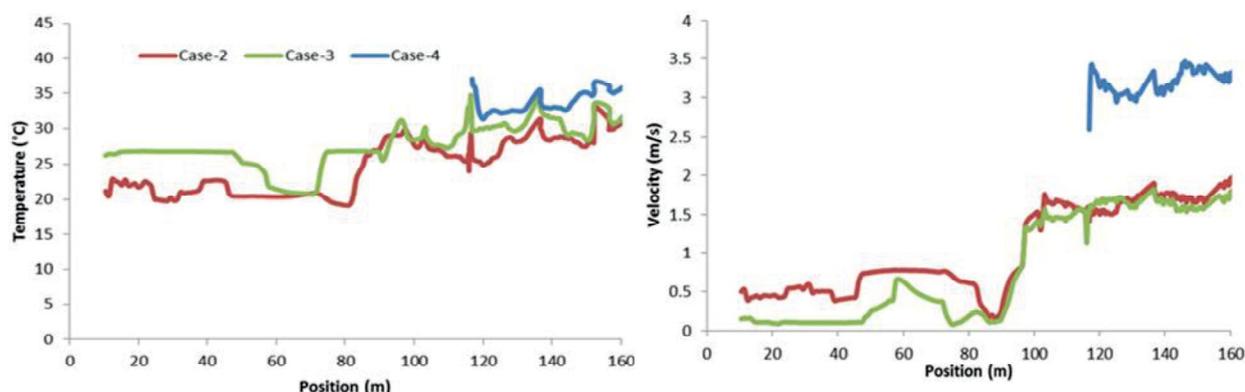


Figure 8: Temperature and velocity distribution at the Roof B

4. FLOW FIELD ANALYSIS

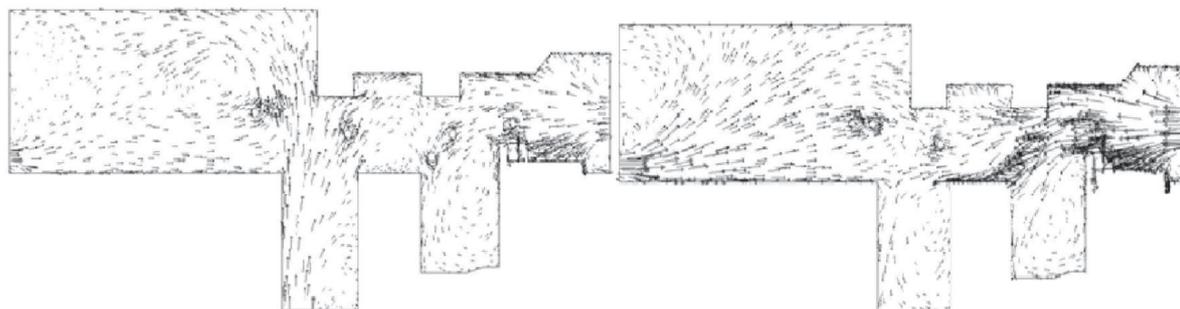


Figure 9: Velocity vectors at a plane located 2 m above floor level for Case-2. ($V_{max}=2\text{m/s}$)

Figure 10: Velocity vectors at a plane located 2 m above floor level for Case-3. ($V_{max}=2\text{m/s}$)

Dust distribution depends on the flow direction and magnitude. Flow field is presented to assess the influence on the flow distribution when opening and closing the gates. Figure 9 and figure 10 show the velocity vectors for Case-2 and Case-3 respectively. The flow is very complex and there are some local recirculation zones, which might trap the dust particles. The velocity at Gate B and Gate A increases with closing the Passage B as shown in figure 10. Furthermore, a local circulation zone is build at area closed to the Passage B with closing the Passage B. The streamlines released from the Gate B is plotted and shown in figure 11 and figure 12 for Case-2 and Case-3 respectively. When Passage-B is open the flow rate from the Gate B is reduced and does not penetrate deep inside the building, but by closing the Passage B the flow rate on the Gate B

increases. The increased flow penetrates deep inside the building area. With closed Passage B, the flow turns towards the casting bed area, as shown in figure 12.

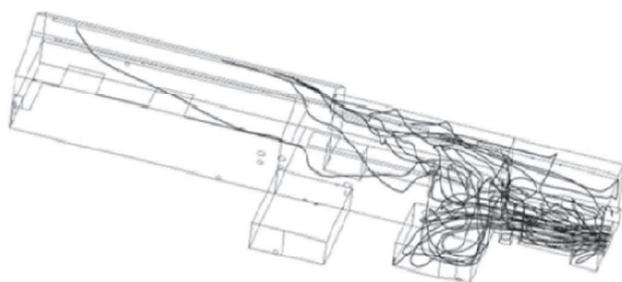


Figure 11 Streamlines released from the Gate B for the Case-2

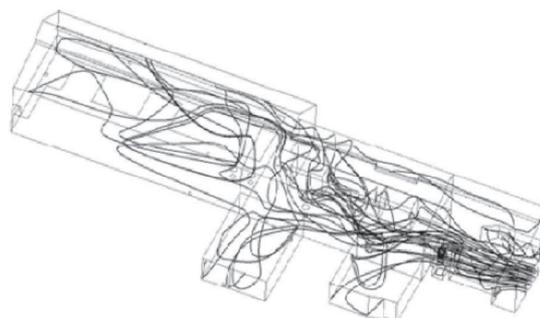


Figure 12 Streamlines released from the Gate B for the Case-3

5. CONCLUSIONS

In the present study CFD studies based on Reynolds Averaged Navier-Stokes (RANS) approach is applied to understand the flow behavior inside a furnace building at the Sauda plant operated by Eramet Norway. The CFD model was calibrated against experimental measurements. Since the main objective was to study the effect of closing and opening gates, passages and roof sections, wind was neglected. Thus pressure boundary conditions on the inlets and outlet surfaces were applied which simplifies the model and reduces the computational cost. Atmospheric pressure was set on inlet and outlet surfaces. With pressure conditions, the mass flow rate is 22 % lower than the measured rate. This is explained by the presence of wind during the experiments.

Closing an inlet or an outlet area will reduce the total mass flow through the building. Closure of an inlet (Passage B), caused a reduction in the total mass flow by 13 % (Case-2) compared to the baseline case. The total mass flow for a case with closed Passage B and closed Roof A and partially closed Roof B is reduced by 60 % (Case-3) compared to the baseline case. Closing Passage B will prevent cold air to affect the working environment at area closed to Passage B but will increase the velocity of the air that enters through Gate B, even though the total mass flow through the building is reduced. Increased velocity at Gate B can have negative effect on collection of dust in the east side of the building. Closing of inlet or outlet area will increase the overall temperature inside the building, as long as the heat sources are same as baseline case. Furthermore, the dust might accumulate on the west side of the building; this might be beneficial for reduction in diffuse emissions.

6. REFERENCES

- [1] Ansys Inc., ANSYS FLUENT 12.0 Theory Guide. 2009.
- [2] Camille Allocca, Qingyan Chen, Leon R.Glicksman, Design analysis of single-sided natural ventilation. *Energy and Buildings* 35 (2003); 785-95.
- [3] M.J. Cook, An evaluation of Computational Fluid Dynamics for Modelling Buoyancy-driven Displacement Ventilation. 1998.
- [4] G.R.Hunt, P.F.Linden, Steady-state flows in an enclosure ventilated by buoyancy forces assisted by wind. *Journal of Fluid Mechanics* 426 (2000); 355-86.

- [5] G.R.Hunt, P.F.Linden, Displacement and mixing ventilation driven by opposing wind and buoyancy. *Journal of Fluid Mechanics* 527 (2004); 27-55.
- [6] Karl Terpager Andersen, Theoretical considerations on natural ventilation by thermal buoyancy. *American Society of Heating, Refrigerating and Air-Conditioning Transactions*, 1995.
- [7] Karl Terpager Andersen, Theory for natural ventilation by thermal buoyancy in one zone with uniform temperature. *Building and Environment*, Volume 38, Issue 11, November 2003, Pages 1281-1289, ISSN 0360-1323, 2013.
- [8] Malcolm Cook, Yingchun Ji, Gary Hunt, CFD Modelling Of Buoyancy-Driven Natural Ventilation Opposed By Wind. *Ninth International IBPSA Conference*. vol. Montreal, Canada, 2005.
- [9] Nicolas Mercier, CFD model, sauda's furnace hall. *Eramet Norway Research and Development*, 2007.
- [10] Nicolas Mercier, Temperature Velocity and Pressure measurements in Sauda. *Eramet Norway Research and Development*, 2007.
- [11] P.F.Linden, G.F.Lane-Serff, D.A.Smeed, Emptying filling boxes : the fluid mechanics of natural ventilation. *Journal of Fluid Mechanics* 212 (1990); 309-35.
- [12] Yuguo Li, Angelo Delsante, Natural ventilation induced by combined wind and thermal forces. 2001.