Light Metals 2012 *Edited by: Carlos E. Suarez*
TMS (The Minerals, Metals & Materials Society), 2012

Optimization and CFD Simulation in the Ventilation of AP60 Reduction Cell Buildings

Siegmar U. Baltuch and Edmund G. Baltuch
Air-Therm Inc.
5801 Monkland Avenue,Suite 8,
Montreal, Quebec, Canada H4A 1G4

Key Words: Green Ventilation, Gravity Ventilation, Ridge Ventilation, Stack Effect, Steel Mill Design, Predictive Tools, Thermodynamic Analysis, Computational Fluid Dynamics

INTRODUCTION

Ventilation in process buildings is key to proper smelter function. Correctly engineered systems to evacuate surplus thermal energy from process areas and allow optimized rates of air change are essential, will assure the good functioning for the lifecycle of the process and allow the demonstration of conformity with occupational hygiene standards. Computational fluid dynamic simulations can demonstrate to end users and assist engineers, coupled with traditional calculations and physical modeling, in the optimization of mill ventilation reducing total installed costs while assuring that the proper solution is in place for the complete lifecycle of the process equipment.

Results are being implemented on an aluminium smelter application and we continue to study, optimize and innovate as new tools become available to provide the best possible engineered solutions for particular applications

DISCUSSION

The objective of this research project is the development and implementation of modern approaches and CFD simulation in the engineering of AP60 Reduction Cell Buildings in order to assure the predictive performance, improve the quality and constructability of industrial gravity ventilation equipment and the adjunct hot process buildings while lowering the overall project costs and planned schedules.

Our team was being asked by the world leader in aluminium process technology to provide their newest state of the art pilot facility with an engineered, optimized best in class natural ventilation solution. The project is the convergence of one of the world largest mining groups (RioTinto) by their aluminium division and our by far largest client (Alcan) and the world leader in aluminium electrolysis technology RTA's Pechiney division. All backed up by two giants in the fields of smelter design and construction SNC-Lavalin and Hatch and Associates. All possible effort was to be applied to ensuring our customer with our best possible result.

Much of this project remains confidential and therefore this paper related essentially to the process of Calculation, CFD utilization and the engineering methodology by which tools can be used in a repeatable manner to arrive at a predicted result. No part of the AP60 process, its ventilation requirements or other elements of this world leading technology is discussed in any manner.

For the hot process building of a steel mill to function, heat has to be extracted from the inside of buildings. This is done by having air inlets at the bottom of buildings and outlets at the apex of buildings. The reason for this is to maximize the potential chimney effect. When using stack (buoyancy) effect ventilation, the process heat becomes the driver and therefore increasing this distance (height) improves the ability of the heated airstream to utilize basic gravity for acceleration of the heat/air flow which allows a regular number of air changes per hour within the plant environments all while ensuring that the process environment remains weather resistant under process operating conditions.

This process, has been understood from the time King Charles I of England decreed in 1600 that ceilings should have 10 foot ceilings and that windows should be taller than wider to allow better smoke extraction

In 1914 the American Society of Heating and Ventilation Engineers (the predecessor of ASHRAE) published that 50 cubic meters of air per hour per human was a minimal standard. This amount was reduced in 1989 by ASHRAE/ANSI standard 621989[1] to 27 cubic meters of air per hour per human.

Natural Ventilation occurs by two different means: Wind driven natural ventilation, where the topography of the building is studied to create a building shape where either by pressure or suction, the warm air within the building is drawn to the exterior. This system is reliant on a particular constant minimal wind factor (Cw) which can generally not be assumed for most industrial building applications

Stack effect natural ventilation, where the differential of temperature and pressure between two bodies of air creates buoyancy in the air stream of the air body within the process building. This sort of system must work in all wind conditions including zero wind conditions.

Within the building there exists a neutral plane. Below this line air is drawn into the building, above this line air is exhausted the larger the distance between in the ingress and egress, the greater the chimney effect (therefore the velocity of air and therefore the total quantity of heat that can be extracted from the inside of the process building.

Gravity ventilation is a form of stack effect ventilation which requires no power source. A Process building is a building within which a heat producing transformation of materials is taking place, such as the smelting of non ferrous metals, hot rolling process of creating steel, or the production of glass products.

Within the buildings a very high quantity (as described in kilowatts or British thermal units) is produced and must be exhausted to conform to state and national building stands and in a number of cases state and national hygiene standards.

Although numerical calculation and computational dynamic modeling has been known and used since the the 1960s by the aerospace community and has been approached in the metal sector by a few trendsetting academics [2],[3]. The cost and complexity of the required tools (computers and software), the generation of proper mesh grid modeling and the engineering know how to properly implement and understand these technology meant although they were available, they were not practical and affordable for this particular application.

Instead mechanical engineering firms that engineer and supply the metal sector relied on three tool kits for the design and determination of air inlets, air outlets and the performance of these engineered systems: design calculation, physical modeling and field demonstration of performance.

The standard empirical design calculation for determining the performance of a particular ventilation application is:

$$Q_S = C_d\, A \sqrt{2\, g\, H_d\, \frac{T_I - T_O}{T_I}}$$

where:

Q_S = Stack vent airflow rate, m^3/s

A = cross-sectional area of opening, m^2 (assumes equal area for inlet and outlet)

C_d = Discharge coefficient for opening

g = gravitational acceleration, 9.807 m/s^2

H_d = Height from midpoint of lower opening to neutral pressure level (NPL), m

NPL = location/s in the building envelope with no pressure difference between inside and outside (ASHRAE 2001, p.26.11)

T_I = Average indoor temperature between the inlet and outlet, K

T_O = Outdoor temperature, K

In terms of physical modeling of green gravity ventilators

We have conformed to the **Air Movement and Control Association publication 511 in the design and evaluation of our models as seen in figure 1:**

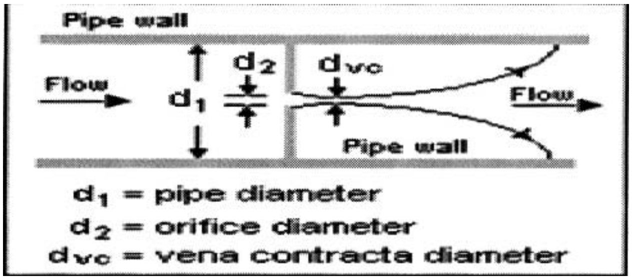


Figure 1

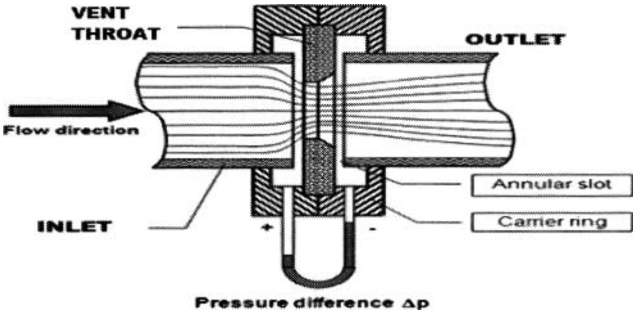


Figure 2. Air Movement and Control Association publication 511

As with any branch of industrial design, project performance in the field have influenced further evolutions in gravity ventilator. Gravity ventilators have evolved from being doghouses or penthouses, which were simply raised sections of the top parts of the process building's peaked roof, with the two side of the raised section left open to allow from heat and fume exhaust to a specialized engineering design feature which allows a calculated flow of air, and heat exhaust while insuring that the building remain weather resistant under positive operating conditions all while utilizing tradition and affordably available tools.

AVAILABLE AFFORDABLE CFD = RELEVANT AND COMPELLING INDUSTRIAL TOOLS

Innovation in an industrial engineering environment must occur due to two dynamics:

EXTERNAL FORCES of change; such as market pressure to design a better product or arrive at a better technical solution for safety or technological reasons or optimize design to compete in an increasingly globalised market place.

INTERNAL FORCES of change. Ventilator and Steel Mill engineering despite the challenges and subtleties are not comparable to designing the next stealth bomber and therefore the need for precision is generally balanced with other priorities such as budget and schedule.

For a new tool or innovation to be implemented this tool must immediate be relevant and compelling by improving the product, reducing the cost or allowing a better predictive tool for clients (both technical and non technical) to be assured that the right result is being applied lowering the stress and risk or project decision makers.

COMPUTATIONAL FLUID DYNAMIC tools that have been available in the past were expensive (within the budget parameters of most industrial projects), cumbersome (not user friendly) and took more time in general then deriving reasonably accurate results from empirical calculation, coupled with physical modeling and precedent field results. In the past although technical interesting and academically worthy, they were far from having an everyday place in an engineering tool box.

In the past 3 to 4 years due to increased computing capacity, a number of recent software packages available, the practicality and ease that these applications can be used to develop usable models, applied multiple field conditions and derive accessible understandable results, these tools have opened and entire horizons of opportunity to design, investigate and optimize mill designs and placement of ventilation inlet and outlets and optimize plant conditions for the manpower, steel mill equipment communities and lessen the environmental impact of the mill.

Computational fluid dynamics involves the discretization of the solution domain into finite volumes (control volumes) and the subsequent solving of the Navier-Stokes (conservation of momentum), conservation of mass, and conservation of energy equations. In addition, in this case, the buoyancy forces are also being considered. The commercial CFD software Airpak version 3.0.16 was used for the analysis. Airpak was originally developed by Fluent Inc as part of their inventory of specialty software and makes use of the Fluent solver to solve the system of equations. Fluent is a well known CFD solver that has been benchmarked for many types of problems. Based on the positive results utilizing this system we are currently upgrading to the next level of solver called CFX.

TESTING AND UNDERSTANDING CFD SIMULATION AS A RELEVANT TOOL

This research project has been carried out utilizing a quantitative research methodology. After an audit of available resources and the creation of a pilot project (one shot experimental case study), which would give rapid prognosis, and observing the strong potential for a successful result, a pretest-post test control group design for a complete project roll out was executed.
By assuming that all engineering technologists and engineers are equal we can declare that in equal groups the manpower required for the study were fully randomized. Due to the nature of the particular project, it was possible to have a control group working side by side with the group implementing the experimental system. It was immediately obvious to know who was part of the experimental group and who was not.

Basically one group used had access to internal; and external CFD resources and the other did not. Being in an engineering office together, both teams immediately recognized that management was implementing a test system. From an engineering standpoint this meant assembling the required information and specification, designing a system specific to the proposed application, implementing the system and then verifying the performance, using empirical data.

Due to the confidentiality agreement signed between Air-Therm, the client and a number of process equipment manufacturers involved on this project, the name and location of this project must remain confidential. This is typical in the field of steel mill design and construction where any technical advantage that might render the Steel Mill better performing allowing for the cheaper production of metal is closely guarded.

The findings of this research experiment are currently being repeated on other projects by this same sponsor therefore demonstrating that this integrated barcode driven construction management system will optimize project resources saving time and money. A major European Metal and Mining group is currently investigating the potential of implementing this system on a complete project (not simply the ventilation systems). Based on the findings on the succeeding pages, this same type of system (marginally customized for many different applications), could be repeated with a similar level of performance improvement found.

Most better modern CFD will allow designers to drop a 3 dimensional AutoCAD, Microstation or CATIA drawing directly into the solver therefore substantially reducing the engineering hours required to build and launch the simulation work. The more accurate and detailed that the model dropped into the solver the more discreet the results will be.

In this particular application, an Air-Therm VG-Series ventilator is positioned at the top the smelter application with a particular height of from ground to ventilator inlet (throat). To simplify the analysis, the model is a 6 metre deep section of the HRM facility with a precise heat load as designated by the client. The dimensions are identified in Figure 3 below. Figure 4 shows the details of the potential Air-Therm ventilators. In the CFD model, the ventilators were simplified in order to keep the mesh size reasonable, however, the important dimensions such as flow areas were maintained similar to those of the actual ventilators

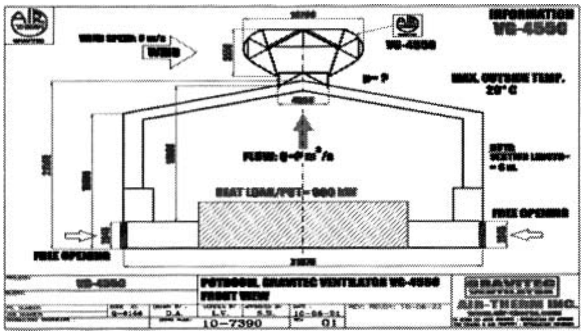


Figure 3a

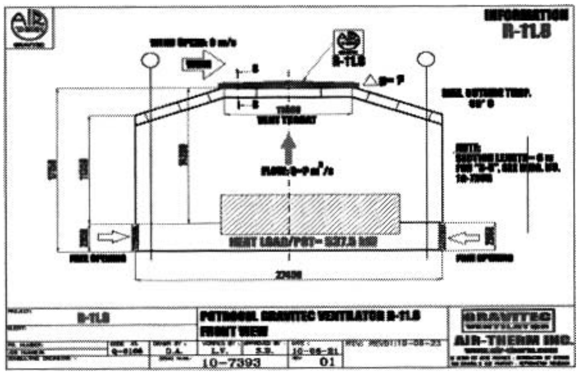


Figure 3b

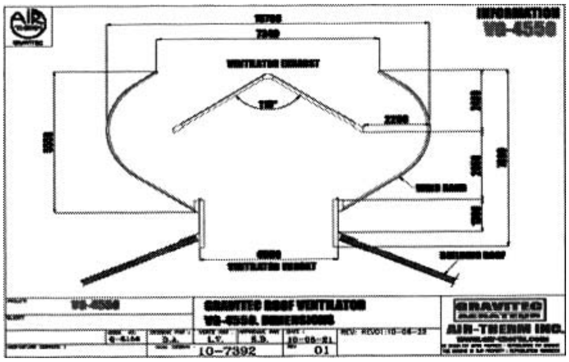


Figure 4a

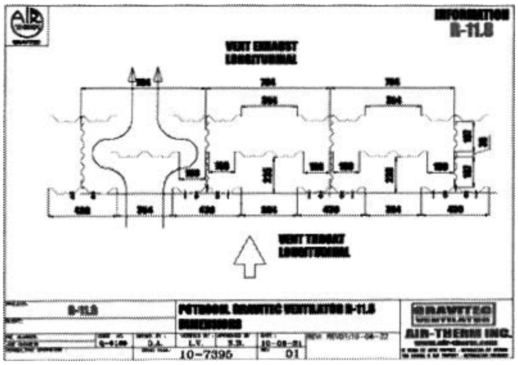


Figure 4b

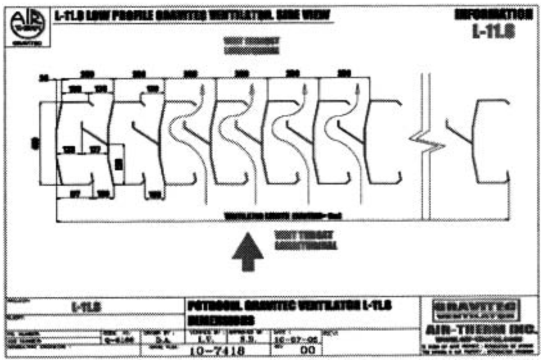
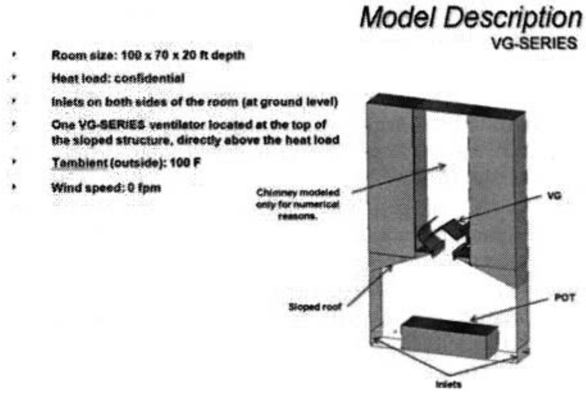


Figure 4c

The CFD model is shown in Figure 5 along with the boundary conditions used for the analysis. A scale is shown at the right of Figure 7 for reference.

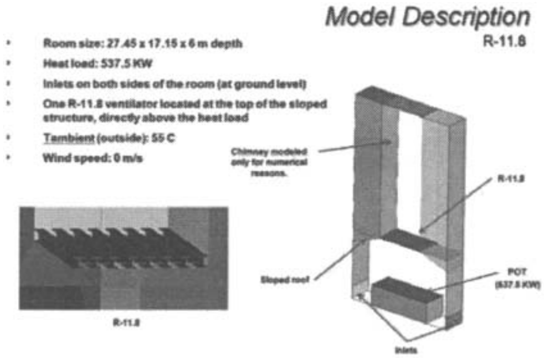


Model Description

VG-SERIES

- Room size: 100 x 70 x 20 ft depth
- Heat load: confidential
- Inlets on both sides of the room (at ground level)
- One VG-SERIES ventilator located at the top of the sloped structure, directly above the heat load
- Tambient (outside): 100 F
- Wind speed: 0 fpm

Figure 5a



Model Description

R-11.8

- Room size: 27.45 x 17.15 x 6 m depth
- Heat load: 537.5 KW
- Inlets on both sides of the room (at ground level)
- One R-11.8 ventilator located at the top of the sloped structure, directly above the heat load
- Tambient (outside): 55 C
- Wind speed: 0 m/s

Figure 5b

The analysis shows overall flow rates at the ventilators. Based on an open area of ventilator, a representational mean velocity of at the inlet (throat) of the ventilator is derived. Figure 6 shows particle traces of the flow field colored by temperature. It should be noted that for the purposes of this study, a chimney was defined above the ventilator in order to place the exit boundary conditions away from the actual ventilator.). Although this analysis was not carried out specifically to investigate the height of the plume, it is interesting to note that at the top of the chimney the air temperatures are still in the 40C range. The temperature difference would keep the hot air moving up, indicating that perhaps the plume could have a significant height before it mixes and its temperature is reduced to the point where buoyancy effects have been negated.

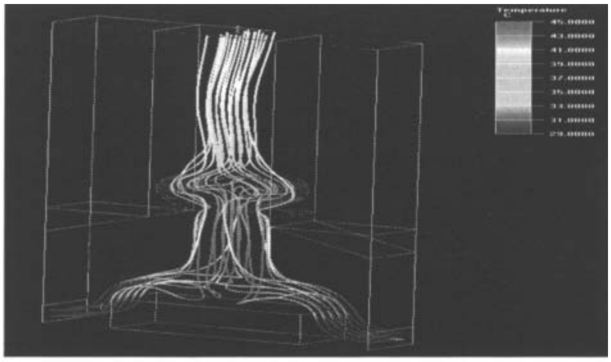


Figure 6a

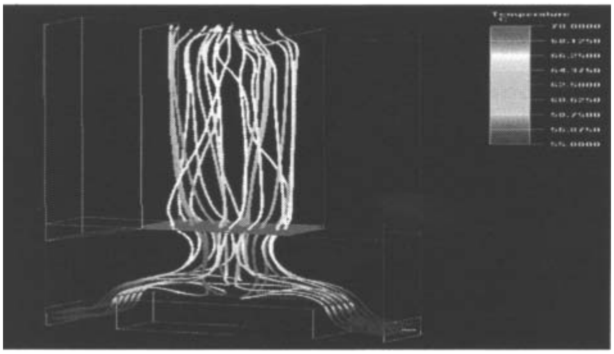


Figure 6b

The heat dissipated by the reduction cell creates buoyancy forces which allow the hot air to rise and exit through the ventilator. Figure 7 shows a mid-plane plot of the temperature distribution through the building. There is a clear stratification of the temperatures with a plume rising directly above the reduction cell. The temperatures near the inlets are very near ambient levels as the high flow rates entrain ambient air from the outside.

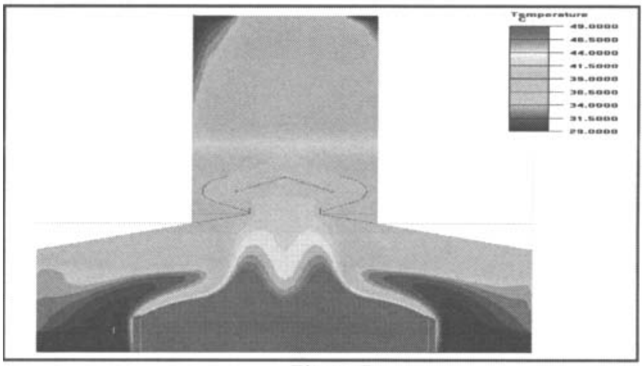


Figure 7a

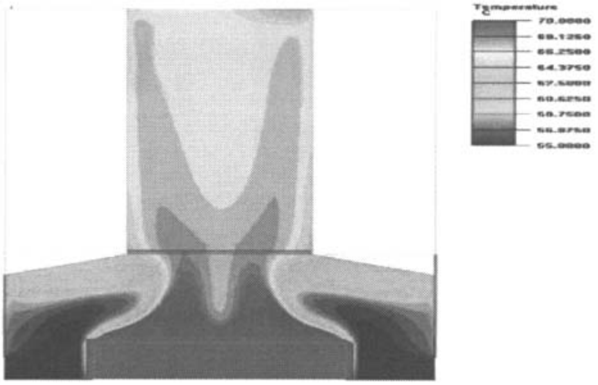


Figure 7b

Figure 7 shows air velocity vectors through the ventilator. Figure 7 shows the air velocity distribution at the inlet (throat) of the ventilator. The analysis shows overall flow rates at the ventilator of. Based on an open area, this designated a mean velocity at the inlet (throat) of the ventilator

Figure 8 shows a mid-plane plot of the pressure distribution through the building. Figure 8 shows the "0" pressure iso-surface in the building shows a close up view of the pressure distribution in the ventilator which we can utilize to modify the angle and location of ventilator components.. The pressure plot also show the negative pressure gradient as the air flows through the ventilator driven by the buoyancy forces.

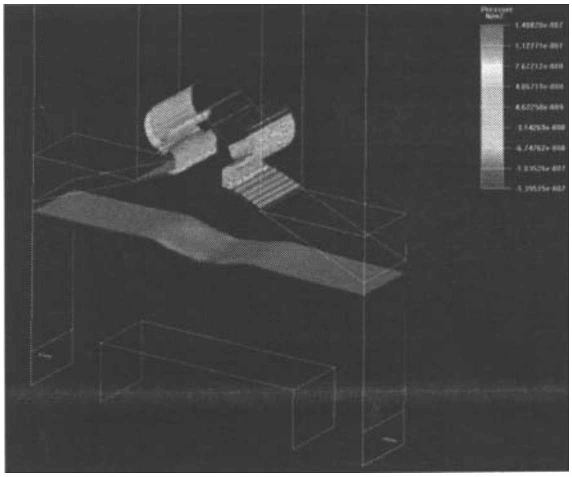


Figure 8

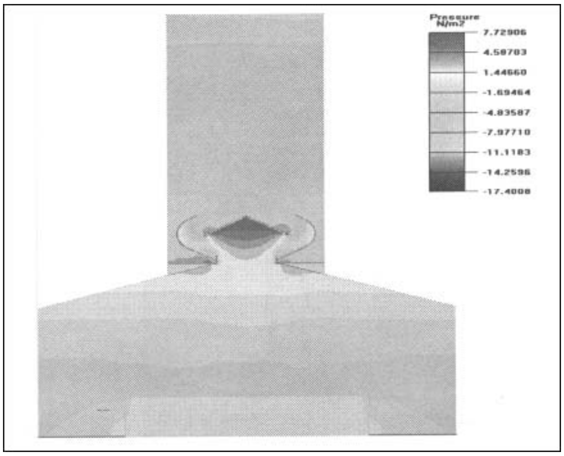


Figure 9

CONCLUSIONS

The team that designed with only empirical calculation and physical modeling arrived with an answer very similar to those that utilized the CFD options, however without the visual results or ability to rapidly change internal external conditions or replace equipment with other ventilation options.

With the advent and availability of affordable and practical CFD solutions, optimization in design is possible in the early stages of project planning that allow for both better understanding and back up of empirical calculation and physical modeling results. We have found that in its current state CFD technology can be considered as a "just in time" technology as explained in figure 10.

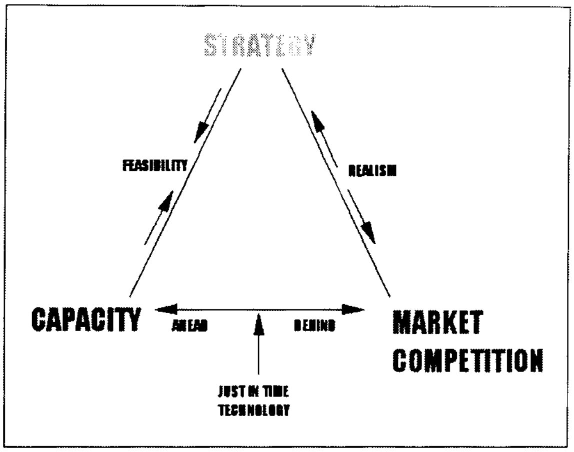


Figure 10

This study demonstrates that beyond the historical and correct use of empirical calculation and physical modeling, affordable and available technologies are improving our ability as engineers to deliver needed information and produce the best possible engineered result.

A design team can now request to see different sizing and types of ventilators to gauge the potential results, can adjust and move the location of air inlets and even potential adjust the location of process equipment to improve air guidance performance for man, machinery and the smelting Aluminium.

REFERENCES

1. American Society of Heating Refrigeration and Air Conditioning Engineers 1989. ASHRAE/ANSI Standard 621989 (Atlanta).

2. M.Dupuis, "Turbulence Modeling of Air Circulation in an Enclosure with Multiple Openings and Local Heat Sources", Proceedings of the 32nd Annual Conference of CIM, Computer Software Section, 1993 229-236.

3. H.B. Awbi "Application of Computational Fluid Dynamics in Room Ventilation", Building and Environment, 24 (1): 73-84 (1989).

4. L. Davidson; "Ventilation by Displacement in a Three Dimensional Room-A Numerical Study": Building and Environment, 24 (4): 363-372 (1989).