How Metals Manufacturers Are Meeting Internal and External Air Quality Requirements with Flow Modeling

Martin Kozlak, Alden

Introduction

Over the last ten years, metals manufacturing plants have found themselves under increased scrutiny for indoor air quality and emissions.

As an example, the aluminum manufacturing process involves the conversion of alumina (aluminum oxide) to elemental aluminum. This takes place in large cells (also called "pots") in which a high current induced by low voltage drives the high temperature reaction. The process releases Hydrogen Fluoride gas, which has been shown to cause asthmatic symptoms, as well as to induce respiratory disease in exposed plant workers[i]. Plant designers must ensure that workers are protected by proper ventilation in the potroom. Additionally, temperatures must be kept low enough to provide reasonable comfort and safety for plant staff. Similar requirements exist for other process rooms in aluminum and other metals manufacturing facilities. The challenge is to design ventilation systems (and possible design modifications driven by other requirements) that will sufficiently limit exposure with a reasonable assurance of success, without having to make structural modifications after construction or retrofit installation.

Regarding external emissions, metals manufacturing plants must install air pollution control (APC) equipment in exhaust systems in order to meet local emissions requirements. In the U.S., this is driven in the aluminum industry by the Environmental Protection Agency's 1997 MACT ruling (see sidebar, "Meeting the 1997 MACT Rule"). The vast majority of these systems rely on providing an even distribution of gas flow through the equipment, in order to maximize contact with the scrubbing agent or particulate capture technology. This is no simple task, particularly since APC units are often installed as a retrofit with a very limited footprint, requiring duct work that is far from ideal, aerodynamically.

Flow modeling can often be a solution to both of these challenges. This paper introduces the associated techniques and provides examples showing its usage in such applications.

Meeting the 1997 EPA Aluminum MACT Rule

In 1997, the USEPA issued a final rule to reduce emissions from air toxics from primary aluminum plants. The Clean Air Act Amendments of 1990 required the EPA to regulate sources of 188 pollutants, developing standards requiring the use of maximum achievable control technology (MACT) for pollutant reduction. Industries covered are those emitting 10 tons/year or more of a listed pollutant or 25 tons/year or more of a combination of pollutants[1].

Wet scrubbers are typically used to control gaseous and particulate fluorides as well as other particulates in aluminum processing plants. A fluoride absorption system is also sometimes used to control both gaseous and particulate fluorides. The latter consists of a fluidized alumina bed and a filtration system that passes the alumina dust back to the potlines. Additional control devices for particulates may include cyclone separators in series, electrostatic precipitators (wet or dry), or dry alumina scrubbers.

Flow Modeling

Flow modeling is an indispensable tool for reducing risk for industrial projects involving fluid flow. The field of flow modeling can be broken down into numerical modeling and experimental, or physical, flow modeling. Numerical models may be one-dimensional, two-dimensional, or fully three-dimensional. The latter is generally referred to as Computational Fluid Dynamics (CFD). Physical flow modeling entails the construction of a model, typically scaled so that the relevant geometry is smaller than the real-world prototype. Scaling laws are employed to make sure that the physical flow model behaves similarly to the full size prototype.

Computational Flow Modeling

CFD entails breaking down a digital representation of the geometric domain of interest into a large number of computational cells (a "grid"), then using a powerful computer (or series of computers) to solve the equations of motion for the fluid in each cell. Over the last fifteen years, commercial CFD software and computational speed have enhanced both ease-of-use and the feasibility of solving real-world problems using this method. Generally speaking, investigating numerous conditions and geometries is faster using a digital model than a physical one or a prototype.

While conventional wisdom puts the aerospace industry forward as being the early adopter of CFD technology, power generation and thermal applications were very early to the table for commercial software. One of the first applications of the original FLUENT solver (now sold by ANSYS, Inc.) was to simulate flow in a gas turbine combustor. During the 1990's, large room ventilation problems with complex geometry became tractable for CFD through improved grid meshing algorithms and high speed computers. Since that time, more examples of ventilation in manufacturing facilities have appeared.

Physical Flow Modeling

In a physical flow model, the geometry of interest is typically built out of plexiglass and/or plywood at a reduced scale. The matching of the velocity head (for inertia-dominated flows) or the internal Froude number (for buoyancy dominated flows) provides results that can be scaled back to the prototype. Matching the velocity head means that ½ ρV^2 is the same in the model and the field, where ρ represents the gas density and V represents the gas velocity. Matching the internal Froude number would entail keeping $Fr_i = V/\sqrt{\left(\frac{\Delta\rho}{\rho}\right) gL}$ the same, where g is the acceleration of gravity and $\Delta\rho$ represents a typical density difference in the flow field.

Although CFD is often less expensive for the majority of cases, many projects still entail the use of a physical model. The root reason comes down to trust. While CFD capabilities have made enormous strides, industry's CFD experience does not match the many years of history in using physical modeling to ensure desired behavior of gas flow systems. Additionally, there are still some complex flows where CFD has not been sufficiently validated (such as multiphase flows). Therefore, engineers often recommend physical modeling. Even when CFD has been properly validated for the physics involved in a particular project, physical modeling may be required based upon the perceived risk involved, or simply upon the comfort level of the project owner or reviewing agency. Verifying the construction of a physical model to the proposed conditions is very straightforward because of the tangible nature of the study. Stakeholders are not always willing to put the same trust in the digital geometry of a CFD model. Cost, schedule, and the availability of a practitioner who has the appropriate experience in the various modeling methods will also play a role in any decision to use physical modeling as opposed to CFD.

Case Study: Ensuring Continued Ventilation with Proposed Exhaust Duct Modifications

The intent of this study was to determine the thermal impact on an aluminum manufacturing plant's potroom interior as a result of modifications proposed to the existing vent system to address operational concerns. After simulating baseline conditions, a candidate modification to the vent system was evaluated and the results compared to the baseline. The goal was to show that there was no unacceptable temperature rise within the potrooms as a result of the modification. In the event of an unacceptable rise, the effort would have shifted to development and testing of mitigating design changes.

Modeling Approach

A full scale, three-dimensional CFD model of one bay of the potroom was developed. The model incorporated sufficient length to include four pots and enough exterior doors to represent the overall structure adequately. The CFD model of the potroom was then adapted to include the existing and proposed roof vent configurations. This model was encased within a larger volume representing the ambient surrounding air. Air flow inlet planes were defined with specified flow velocity at the side boundaries of the surround, to both sides of the potroom. The model outlet plane was defined with a constant pressure at the upper boundary of the surround. The pots were modeled as solid blocks, the walls of which were treated as surfaces with a constant temperature.

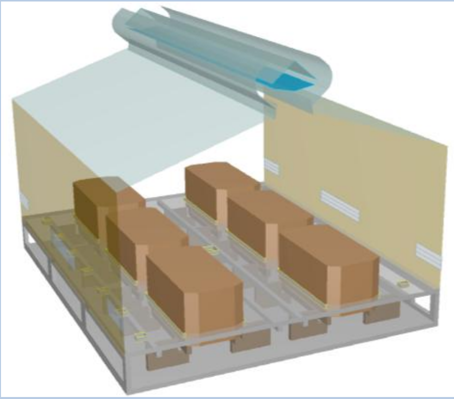

Figure 1: Digital model of a potroom bay and roof vent

The CFD code FLUENT was used for this study. A viscous, incompressible, turbulent, simulation was performed to calculate the velocity distribution through each of the systems. Convection, conduction and radiation effects were included.

a)

b)

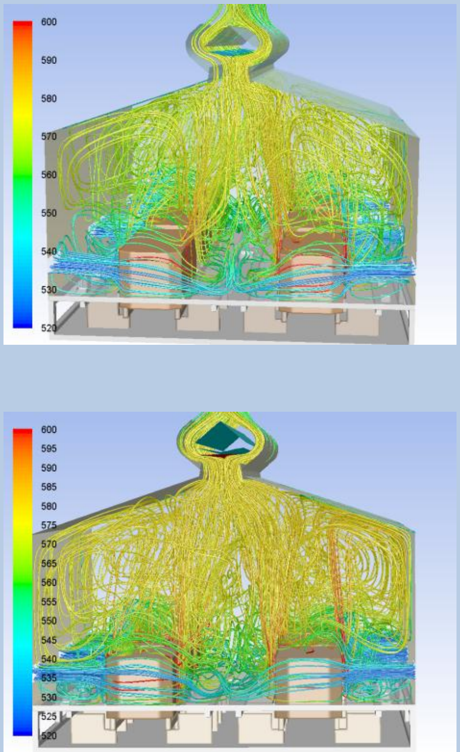

Figure 2: Flow pathlines colored by temperature for a) baseline geometry and, b) proposed venting system modification

Figure 1 depicts the digital geometry of the potroom bay and the roof vent. The pots are shown in brown and the roof vent is at the top. Louvers and side vents are included in the model. Flow pathlines colored by temperature are shown in Figure 2, showing that the flows are qualitatively similar for both cases, though the air residence time seems to be longer with the modification, with longer pathlines. This is corroborated by the average velocity at the vent outlet, which was 8.5% lower for the case with the modification. Correspondingly, the average temperature at the vent outlet was 7 ^{o}F hotter with the modification. It was concluded, however that because there was virtually no temperature change below a height of six feet, the modification would cause no decrease in comfort level for plant workers, and the plant moved forward with the modification.

Scrubber Performance Optimization

In addition to optimizing safety and manufacturing operations, flow modeling is also widely used to optimize air pollution control equipment, ensuring that purchased units will perform as required to meet local environmental regulations. As discussed in the Introduction above, performance of APC systems usually requires the existence of a flat velocity profile in the contacting region, so that gas is not "short-circuited" through a partial cross-section, reducing residence time. Often, a lack of available space for retrofitted air pollution control systems leads to upstream ducting that is non-ideal from a fluid dynamics perspective. Computational or scaled physical flow modeling allows an investigation of flow distribution in a unit before installation, as well as assistance in designing remedial flow control systems for improved distribution. Such a process minimizes project risk.

The example depicted here is for a proposed wet sulfur dioxide scrubber with four inlet ducts. Such a scrubber relies on a sorbent slurry spray being introduced inside a vertical tower such that the flue gas contacts the spray on its way upward through the tower. A mist eliminator above the spray headers removes most of the droplets that are "carried over" by forcing the droplets through serpentine channels, thereby forming a draining film on the channel walls.

The engineering firm tasked with construction and guarantees wanted the following:

- Minimized system pressure losses in the duct work
- Acceptable gas flow distributions at the booster fan inlets and absorber inlet
- Uniform gas flow distributions exiting the mist eliminator
- Acceptable pressure drop through the absorber
- Minimized liquid pull back into the inlet duct
- Effective liquid collector and drain designs for the absorber outlet duct and stack liner

In order to meet these requirements, the firm contracted a physical model study that would also provide the necessary design refinements. Because the ductwork both upstream and downstream of the booster fans was of interest, the physical model from the particulate control unit outlet to the absorber outlet was split into two sections. Symmetry allowed some cost reduction for the ductwork upstream of the fans, such that models were constructed for just two of the four ducts. The absorber model included ductwork for all four inlets downstream of the booster fans, and is shown in Figure 3.

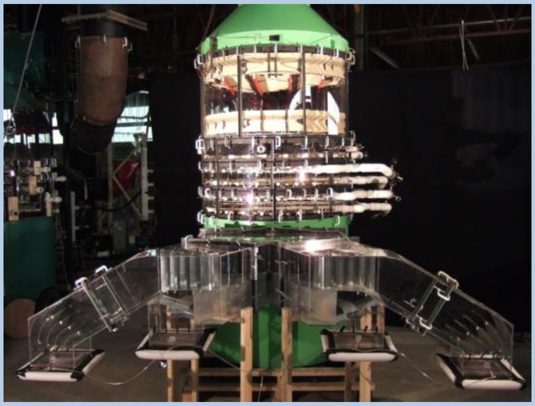
Figure 3: 1:12 scale physical model of a four inlet absorber

One of the major challenges in this case was creating a uniform velocity profile downstream of the duct junctions on each side of the absorber. In order to address this and to minimize pressure drop, some perforated plate sections in corners were added, as well as turning vanes and a false floor to guide the flow in one section, as shown in Figure 4. The modeled vanes and false floor can be seen as the aluminum sheets in the figure, while the perforated plate sections are visible in the upper right and upper left corners downstream of the turning vanes. All of these ducting flow controls resulted in a root mean square velocity variation in a cross-section downstream of the junction of just 13%, very acceptable for the entry into the absorber and considerably reduced from the 61% observed in the baseline design.

Figure 4: Flow controls in the absorber duct

Absorber testing for a scaled model is split into dry testing and wet testing. Pressure losses are evaluated through the system, and velocity profiles are measured at the mist eliminator outlet at the top of the scrubber. In this case, the mist eliminator pressure drop is simulated by an appropriately sized perforated plate. During wet testing, water spray is introduced and full sized mist eliminator plates replace the perforated plate at the top of the absorber tower (full size plates must be used because the droplet-film dynamics do not scale properly). Total pressure measurements are taken again during wet testing, and overall flow patterns are observed. In this case, the velocity distribution at the mist eliminator outlet was found to be acceptable, and there was minimal liquid splash-back (liquid film falling across the inlet duct) at full load.

Figure 5: Modeled liquid collection system downstream of the absorber (right) and into the stack breach (left). The angle brackets can be seen in black, attached to the acrylic walls.

Because the flow leaves the absorber tower at 100% humidity, condensation forms in the downstream duct and in the chimney (or "stack"). Any film which forms on the liner wall risks being entrained in the exhaust and causing corrosion on plant or other structures. In order to address this, a third model section was built, from the mist eliminator outlet to part way up the stack. Velocity heads were matched with plant flows of interest, and a water film was introduced throughout the duct surfaces. The resulting behavior was used to design liquid collectors that drive the film to drains at the bottom of the ducts and stack. For the most part, the liquid collectors are made of angle brackets attached to the duct walls, with the 90 degree angle facing the oncoming flow. The installations also tend to be aligned vertically such that gravity and the flow inertia can work together to drive the liquid downwards (Figure 5).

Overall, the study and the resulting design modifications provided significant value to the project by avoiding performance and operational pitfalls down the road.

Summary

Regulatory scrutiny on indoor air quality and emissions for metals manufacturing plants has been increasing steadily throughout the world. This white paper has focused on the aluminum manufacturing process, in particular, showing how computational and physical flow modeling can be used to reduce risk and ensure effective solutions for both concerns. Case studies were presented, the first showing how a potroom venting system could be modified while maintaining temperatures and circulation. The second case study showed how physical flow modeling could be used to ensure the proper performance of an exhaust sulfur dioxide scrubber.

Martin Kozlak is the Director of Gas Flow Systems Engineering at Alden Research Laboratory. The group uses physical and Computational Fluid Dynamic (CFD) models to solve industrial and utility power generation flow-related problems. In addition to participation in selected projects, he provides overall technical and fiscal management of gas flow studies, directing the team of engineers, technicians and craftspeople responsible for developing efficient and cost-effective solutions. His particular expertise is in the area of thermal and fluid dynamics. He has over 25 years of experience directing research and product development related to the process industries and environmental control systems. He has numerous patents relating to improved steam generator components, injection and firing systems, coal pulverization and flow measurement systems.

References

[i] Bas Sorgdrager, Aart J. A. de Looff, Jan G. R. de Monchy, Teake M. Pal, A. Ewoud J. Dubois and Bert Rijcken, 1998. "Occurrence of occupational asthma in aluminum potroom workers in relation to preventive measures," *Occ. & Env. Health*, **71**, N1, pp. 53-59.