Computer Simulation Helps Solve SO2 Scrubber Problem in Primary Aluminum Smelter Plants

By Chang-LI Hsieh, Koch-Glitsch, Inc., Wilmington, Massachusetts
Frank Hartgers, Hoogovens Technical Services, IJmuiden, The Netherlands
Peter Mihalik, Hoogovens Technical Services, Burlington, Ontario
Greg Boles, Koch-Otto York, Inc., Wichita, Kansas

Figure 1: An image of the scrubber studied

Computer simulation has helped to solve a challenging problem in scrubbers used to remove sulphur dioxide from gas in the production of aluminum at the Reynolds Metal Company, St. Lawrence Reduction Plant, Massena, N.Y. Tests showed that under conditions requiring high gas flow rates, the mist eliminators did not completely remove fine mist and liquor droplets in the gas stream. This resulted in potentially acidic moisture droplets being emitted to the atmosphere. Engineers simulated the operation of the scrubbers using computational fluid dynamics (CFD) and discovered a significant gas maldistribution in the absorber. This was due to the geometry of the tower and inlet ducting. In further CFD work, they evaluated several alternate tower and ducting designs, with involvement of Hoogovens and Koch engineers, and selected one that solved the maldistribution problem at minimum expense.

Engineered and installed by Hoogovens Technical Services, the gas desulphurization scrubbers used at the aluminum smelter consist of two 51 foot, 6 inch diameter stainless steel vertical absorber towers that spray a sodium sulphate/sodium carbonate solution countercurrent to the gas flow. After the gas has contacted the solution, it passes through a mist eliminator to remove any fine mist and liquor droplets remaining in the gas stream. The FLEXICHEVRON® mist eliminators used in this application, produced by the Koch-Otto York division of Koch-Glitsch, consist of a series of turning vanes that force the gas to change direction so that inertia causes droplets to impinge on the blades where they are removed from the gas stream. These mist eliminators collect essentially 99+% of all particles between 8 and 40 microns in diameter, depending on design parameters.

Moisture carryover experienced

The normal gas flow rate specified through the mist eliminators is 1,099,600 actual cubic feet per minute (ACFM) through each of the two absorbers. During periods when one of the absorbers is not in operation, a single absorber is specified to handle 2,034,870 ACFM. During air/water trial operations, moisture carryover was visually witnessed at a flow rate of 1,300,000 ACFM, well below the specified design conditions for single unit operation. At the request of Hoogovens, the mist eliminator manufacturer was brought in to help solve the problem. It's important to note that mist eliminators are capable of effectively removing droplets only when velocity is maintained at or below their design limit, which in this case was 17-18 feet per second. While the scrubbers had been designed so that average velocity was well within these limits, both Hoogovens and Koch-Glitsch engineers were concerned that geometrical irregularities might be causing the FLEXICHEVRON® mist eliminator velocity limits to be exceeded in certain areas of the scrubber.

This problem would have been very difficult to resolve using conventional physical testing methods. First of all, it would have been very difficult to equip the scrubber with enough sensors to determine whether and where velocity limits for the mist eliminators were being exceeded. Secondly, assuming that measurements showed that excessive velocity was the problem, the testing results would provide little or no guidance to engineers in determining the cause. Finally, designing and testing a solution would be very expensive and time-consuming using the physical testing approach. It would be necessary to modify the scrubber and perform another series of tests for each hypothesized solution to the problem. For all of these reasons, CFD simulation was adopted to investigate the problem and examine proposed solutions.

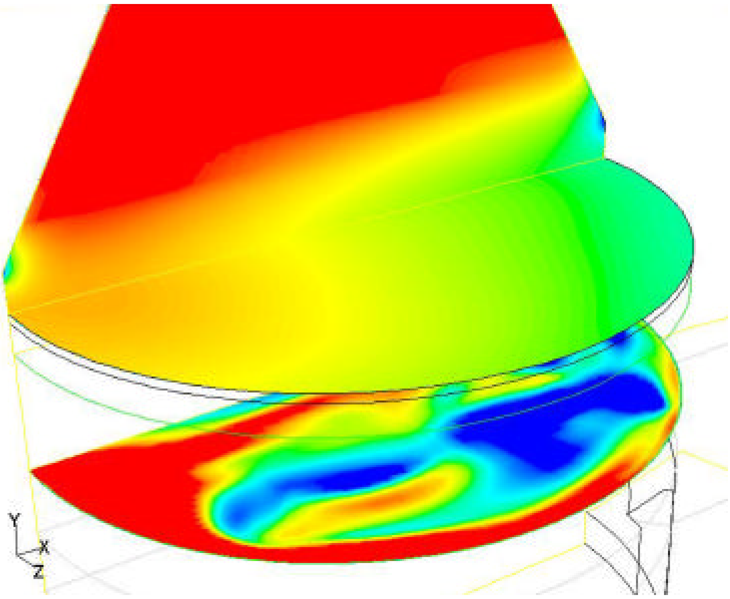

Figure 2: Analysis of the original design showed vapor velocities exceeding the maximum capacity of the mist eliminator

CFD advances

In recent years, advances in CFD simulation have made it possible for engineers to generate a relatively fast and inexpensive computer model of flow within a scrubber. CFD involves the solution of the governing equations for fluid flow, heat transfer and chemistry at several thousand discrete points on a computational grid in a defined flow domain. The use of CFD enables engineers to obtain solutions for problems with complex geometry and boundary conditions. A CFD analysis yields values for fluid velocity and temperature throughout the solution domain. Based on the analysis, a designer or engineer is able to optimize fluid flow patterns or temperature distribution by adjusting either the geometry of the system or the boundary conditions, such as inlet velocity/temperature and wall heat flux.

Koch-Glitsch engineers selected FLUENT CFD software, from Fluent Incorporated, Lebanon, New Hampshire to perform the analysis. FLUENT was selected because it has a powerful range of choices for modeling the turbulent flows seen in scrubbers and other process equipment.

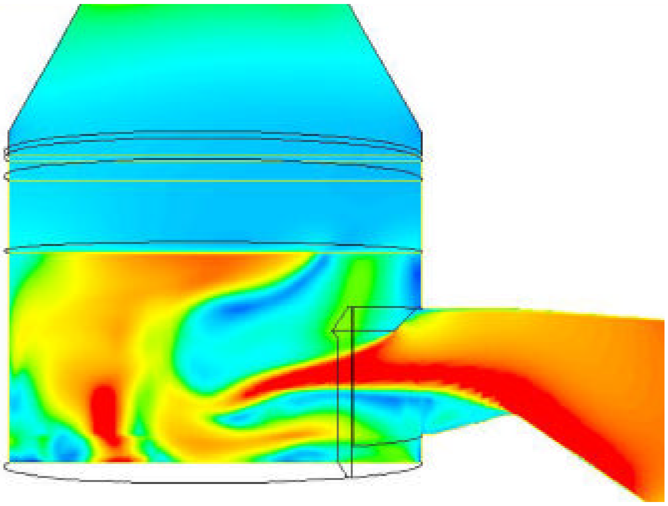

Figure 3: Overall view of airflow in the scrubber helped engineers understand the problem

Simulation matches experiments

The simulation of the scrubber performed by Koch-Glitsch engineers showed, as was suspected, that there was a significant vapor maldistribution in the absorber due to the geometry of the tower and inlet ducting. The countercurrent liquor spray in itself did not provide enough pressure drop across the tower to equalize vapor flow, and stabilize the subsequent vapor velocity through the mist eliminator. The result was that vapor velocities were above 20 feet per second just prior to entering the mist eliminator. The analysis showed the high velocity flow concentrated in the area directly opposite the inlet duct. These results match the evidence that Hoogovens and Koch-Glitsch engineers obtained, which showed moisture coming out of the stack in that area.

The CFD results further confirmed and quantified various design features, which the team suspected to be causing the flow maldistribution. In particular, they examined a two-dimensional plot with velocity vector contours on the symmetry plane of the scrubber and ductwork. This plot provided an excellent birds-eye view of the flow through the scrubber and helped to almost immediately gain an intuitive understanding of what was causing the problem. The plot showed that a "rain hood" around the scrubber inlet helped to focus the gas flowing through the duct into a high-speed flow path that shot across the width of the scrubber until it hit the opposing wall. Once it hit the wall, the flow was forced upward directly into the mist eliminator. Understanding the flow within the scrubber and both inlet and outlet ducting helped to focus the teams' attention on the inlet ductwork and inlet transition to the scrubber. Hoogovens engineers realized that modifying the gas flow within the inlet duct itself and at the junction of the inlet duct and scrubber would greatly impact the performance of the Koch-Otto York mist eliminator.

Iterating to a solution

With the problem now well understood, iterating to a solution was a relatively straightforward process. Since the rain hood was clearly identified as a major contributor to the velocity maldistribution, one of the first things tried was removing the hood from the model and re-running the analysis. This change provided a major improvement in velocity distribution and reduced the maximum velocity to just below the limit for the mist eliminator. Hoogovens engineers also suggested smoothing the bottom of the transition duct by dropping it to the maximum open position allowable from a construction point of view. Engineers tried a variety of other changes in an effort to further improve the velocity distribution and to gain a larger margin of safety. They tried a variety of different baffles in the inlet ducting in order to diffuse the flow entering the scrubber. The most obvious position for the baffle was right on the inlet plane, and the analysis showed this geometry to be very effective in reducing maximum velocity. However, Hoogovens engineers ruled this out because it would be difficult to install and support the baffle plates.

Koch-Glitsch engineers then tried a range of other baffle positions based on Hoogovens engineers' suggestions with the goal of finding a low-cost, low-maintenance solution. They finally found a position in the ductwork, well upstream of the inlet that provided a very even velocity distribution. Hoogovens engineers agreed that it would be easy to install a baffle in this position and that virtually any type of maintenance work could be performed without removing it. The analysis of this new configuration showed that the streamwise velocity was 16 feet per second or less at all areas of the cross-section of the scrubber at the inlet to the mist eliminator. The results were accepted and Hoogovens proceeded with the necessary modifications in a very timely manner.

What would have otherwise required a long and expensive build and test process was resolved in about two weeks. Using the traditional approach, engineers would have wasted months trying to solve the problem by changing the geometry in the area where the maldistribution occurred. Instead, using the simulation results as a guide, Hoogovens and Koch-Glitsch focused immediately on the inlet duct area that the analysis results indicated was the root cause of the problem. In addition to obtaining a quick and inexpensive solution, the customer gained a new level of respect for the analytical capabilities of the two suppliers involved in the project.

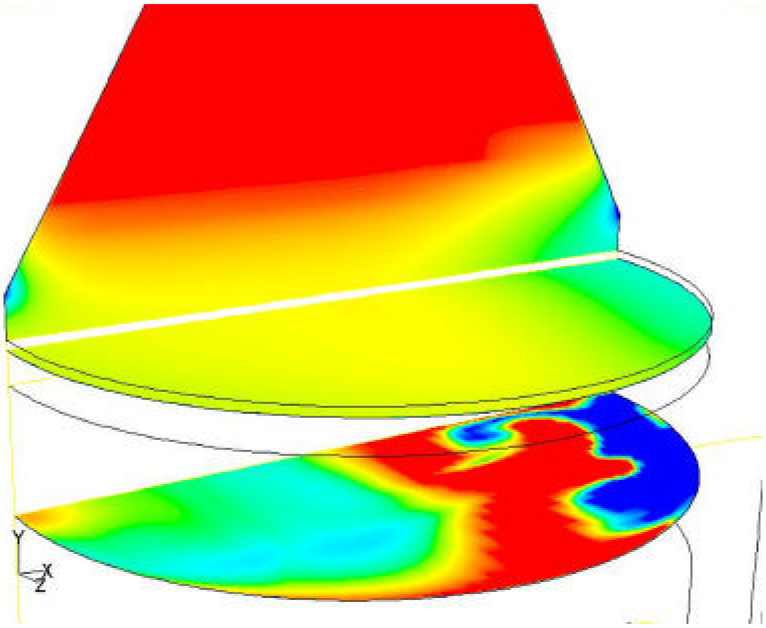

Figure 4: This analysis of the final geometry shows that maldistribution has been eliminated by the time the flow reaches the mist eliminator