Fall 2022

The Airflow Update

AIRFLOW SCIENCES CORPORATION

Flow Modeling and Testing to Reduce Droplet Fallout at a Plant Site

By: Brian Dumont, P.E., Jeffrey Franklin, Ph.D., P.E. Robert Mudry, P.E.,
and John Nitz, Airflow Sciences Corporation

Little can be more frustrating to plant staff than an intermittent and complex operational problem. This was the case at a power plant in the Midwest US with wet scrubbers. Plant staff reported that liquid droplets from the stacks would fall in the vicinity of the plant. Under some conditions, these droplets would land on the ground as a light mist, which was considered undesirable by plant personnel.

It proved difficult to pinpoint when the droplet fallout conditions happened, as it didn't seem to be related in a clear manner to ambient humidity, temperature, or wind conditions. Theories abounded to be sure, but there was no obvious answer. It was even unclear whether the droplets were formed in the stack or if they condensed in the ambient air as the moist plume cooled.

The plant contracted Airflow Sciences Corporation (ASC) to take a deep dive into the problem. ASC assembled a variety of applicable engineering tools – field testing, online cameras, computational fluid dynamics (CFD) models of both the flow inside the stack and the ambient air flow in the surrounding terrain, and a wet physical model of the gas and liquid flow inside the stack.

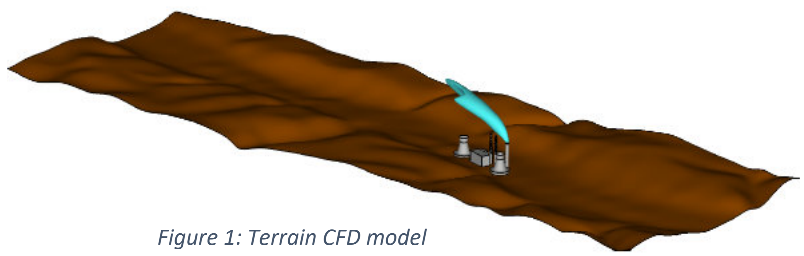

Figure 1: Terrain CFD model

Plant External Flow CFD Model

The terrain CFD model considered a 1 mile by 3 mile region around the plant (Figure 1) and a variety of different wind and weather conditions. Since it was not certain where the droplets were formed, the CFD model evaluated two possible sources. One potential source was the water vapor exiting the stack and condensing into fine droplets. The model can predict the physics of condensation and evaporation, so these droplets were tracked through the atmosphere to determine if they would rain out on nearby areas.

News from ASC

Our Equipment division, Airflow Sciences Equipment, rolled out a new and improved website. The new site makes it easier to learn about our offerings plus has two new features including a media library and blog. We encourage you to take a look at www.airflowsciencesequipment.com

Andrew Juergens joined our team as a Fluid Dynamic Engineer. He graduated with his Bachelor's in Biological Systems Engineering from Michigan State University and is currently pursing his Master's in Mechanical Engineering at Wayne State University. Welcome aboard, Andrew!

Congrats to Jorie and **Courtenay Tooles** for tying the knot this summer. We wish you the best!

Flow Modeling and Testing to Reduce Droplet Fallout at a Plant Site

(Continued)

The second potential source involved liquid droplets being directly discharged from the exit of the stack; the CFD model can track the aerodynamic behavior of these droplets, including evaporation, and determine if they land nearby.

The CFD results showed that droplets formed in the exhaust vapor plume due to condensation were very small and were carried away by the ambient breeze until they re-evaporated. This is shown in Figure 2, with 150 micron sized droplets formed and then tracked until they evaporate. Thus, the vapor discharged from the stack was unlikely to result in droplets that land on the plant grounds or nearby areas.

Similarly, very small droplets emanating from the stack were too light to fall to the ground before evaporating (Figure 3, 50 micron droplets). The model did show, however, that larger droplets that exit the stack would be able to fall in the affected areas before evaporating, as seen in Figure 4 (500 micron droplets) and Figure 5 (1500 micron droplets). It was uncertain, however, why 500 micron and larger sized droplets would be discharged out of the stack because the scrubber mist eliminators are intended to capture all droplets greater than 50 microns. Further investigation was warranted.

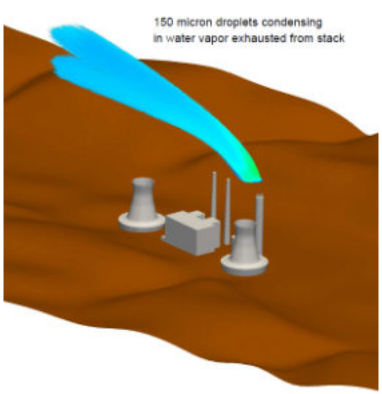

Figure 2: 150 micron droplets condensing in water vapor exhausted from stack

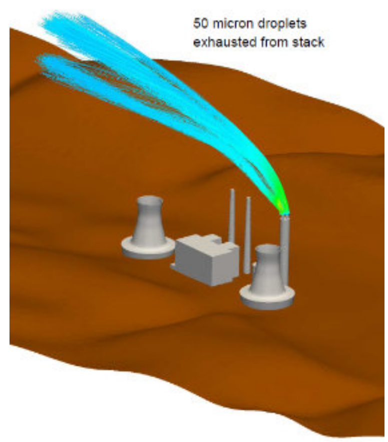

Figure 3: 50 micron droplets exhausted from stack

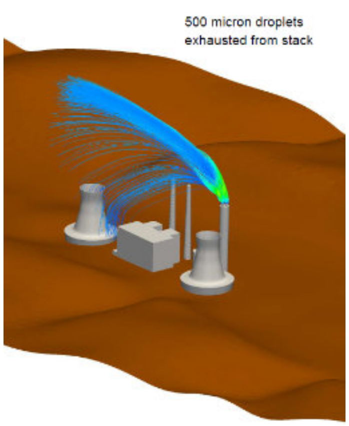

Figure 4: 500 micron droplets exhausted from stack

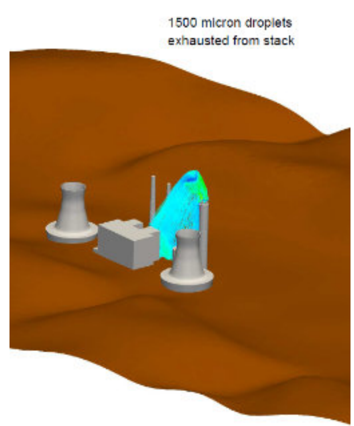

Figure 5: 1500 micron droplets exhausted from stack

Stack Internal Flow CFD and Physical Flow Models

The internal region of the stack was modeled using both CFD and a 1/12 scale physical model, starting at the mist eliminators of the wet scrubber (Figures 6 and 7). These models considered both the gas flow patterns and tracking of water droplets not captured by the mist eliminators. CFD gas flow patterns are shown in Figure 8, and Figure 9 depicts typical water droplet pathlines. A small percentage of the droplets are shown to exit the stack; a majority impinge on the walls of the ductwork and stack. This is an intended behavior, and the stack contains various gutters and drains to remove the water flowing along the walls and floor. The behavior of the water traveling along the walls and floor is modeled in the CFD using a "film model". This type of modeling approach calculates the liquid build up on the walls including the aerodynamic forces and effects of gravity and surface tension. Figure 10 is an example of the film model results, indicating where liquid pools and flows along the wall surfaces; the colors depict the thickness of the water film.

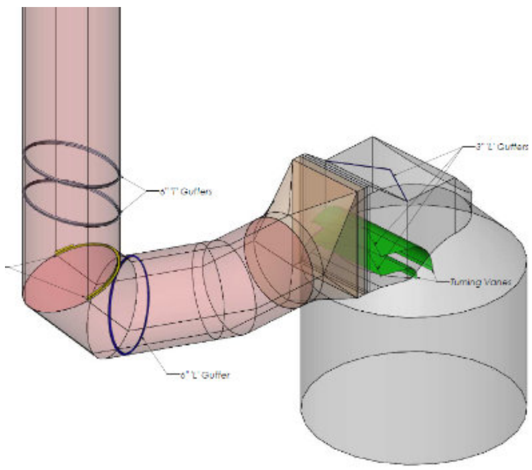

Figure 6: CFD model of the internal region of the stack

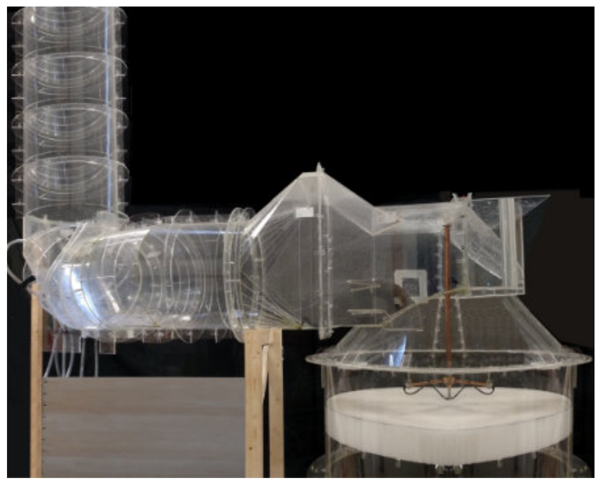

Figure 7: Physical model of the internal region of the stack

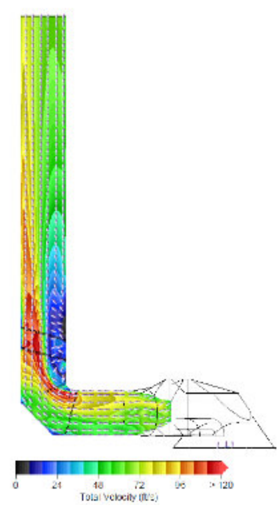

Figure 8: CFD gas flow patterns

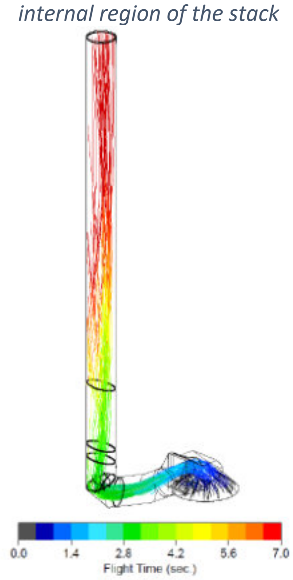

Figure 9: Water droplet pathlines

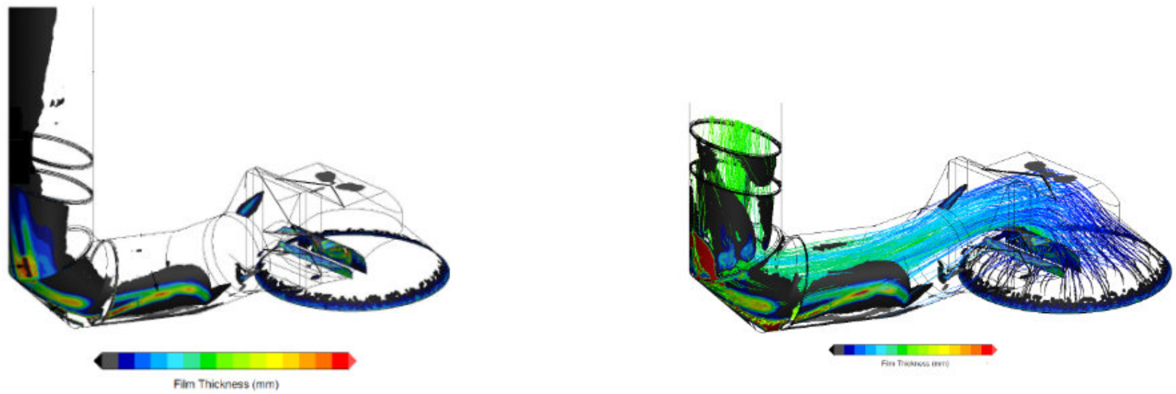

Figure 10: Film model results with color depicting thickness of water film

In the physical model, a fine mist of water droplets was injected and tracked. A blue dye was used in the water in order to better observe their accumulation and behavior in the clear Plexiglas model.

Similar to the CFD results, only a small amount of the injected droplets exit the stack; most droplets impinge on the walls and are removed in the various drains.

A key result, however, of both the CFD and physical models, was the presence of high velocities in the gutters which appeared to prevent drainage (see Figure 11 for the CFD and Figure 12 for the physical model). In fact, in the physical model, there were several areas where liquid captured in the gutters would be pushed upwards and out of the gutters. Along one side of the stack, the liquid film was, in fact, pushed upwards along the stack wall, circumventing all gutters, with the film traveling all the way to the top of the stack. The CFD film model indicated that almost 10% of the liquid captured on the walls exited the top of the stack (Figure 13). Some of the drains were also found to be located in flow recirculation zones, where negative air pressure pockets could actually suck flow out of the drain (Figure 14).

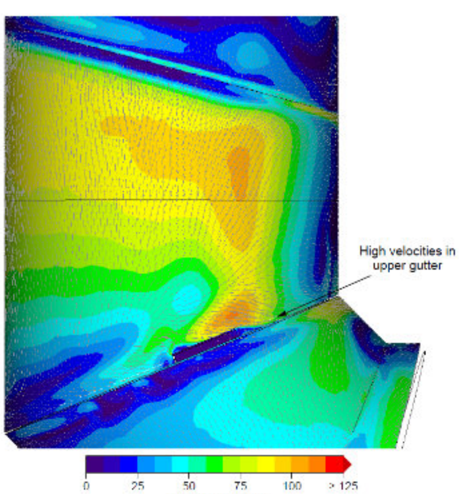

Figure 11: CFD model showing high velocities in upper gutter

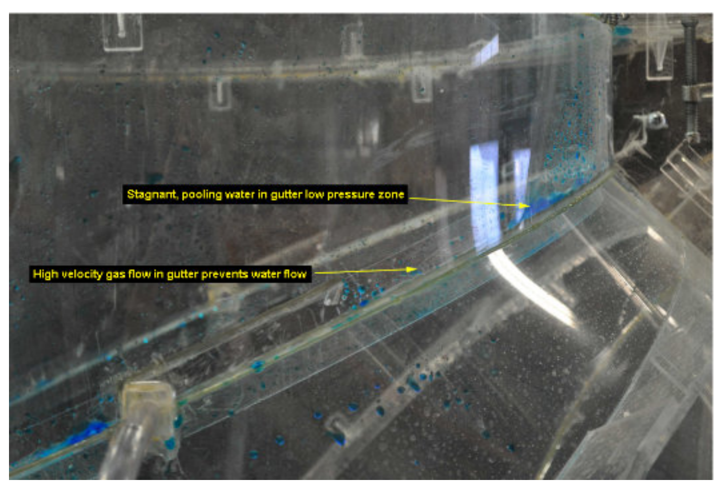

Figure 12: Physical model showing high velocities in upper gutter

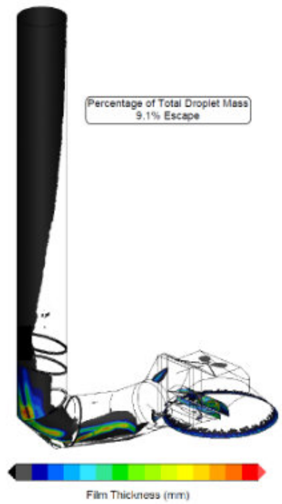

Figure 13: 9.1% of total droplet mass escapes

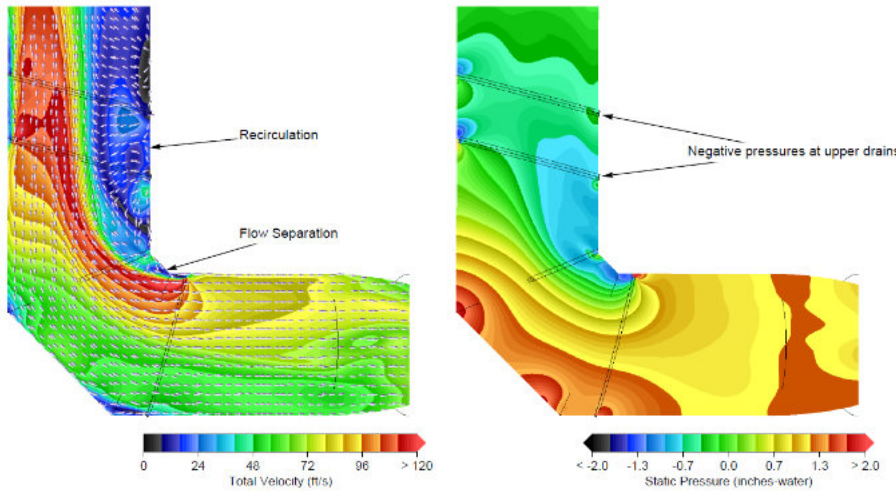

Figure 14: CFD model showing drains found to be located in flow recirculation zones

These all indicated that the problem was related to inadequate droplet capture in the stack rather than condensation outside of the stack. If a liquid film did travel up the walls, it could then shear off from the top of the stack into droplets that are large enough to fall back to the ground as noted in the CFD results of Figure 4 and 5.

The next step in the analysis was to perform field testing at the plant in order to corroborate with the CFD and physical model results. If the models were accurate, liquid and droplets should be visible in the stack and at the edge of the stack during plant operation.

Field Testing

To validate the model results, several different tests were conducted. In the stack, ASC personnel used a high-temperature inspection camera to record video of flow behavior during operation and a plant test crew performed isokinetic flow sampling to measure liquid mass flow. Outside the stack, a drone was used to take video near the top edge of the stack. Another small experiment was conducted to examine if reverse flow existed in the drains due to the negative pressure pockets noted in the modeling; this involved removing some of the drain plugs and replacing them with clear windows to observe the liquid flow.

Both the inspection camera and the drone captured footage that correlated with the model results. Although the stack had very low visibility, the inspection camera video showed a steady film of liquid traveling upwards on the stack walls. The drone clearly captured photos of moisture on the outside of the stack on a dry sunny day (Figure 15), indicative of liquid film running up the stack walls and shedding to droplets from the top edge. The drain experiment observations confirmed that reverse flow did indeed exist in some of the drains, where air was being drawn upwards through the drain piping, impeding the draining efficiency.

Flow Modeling and Testing to Reduce Droplet Fallout at a Plant Site

(Continued)

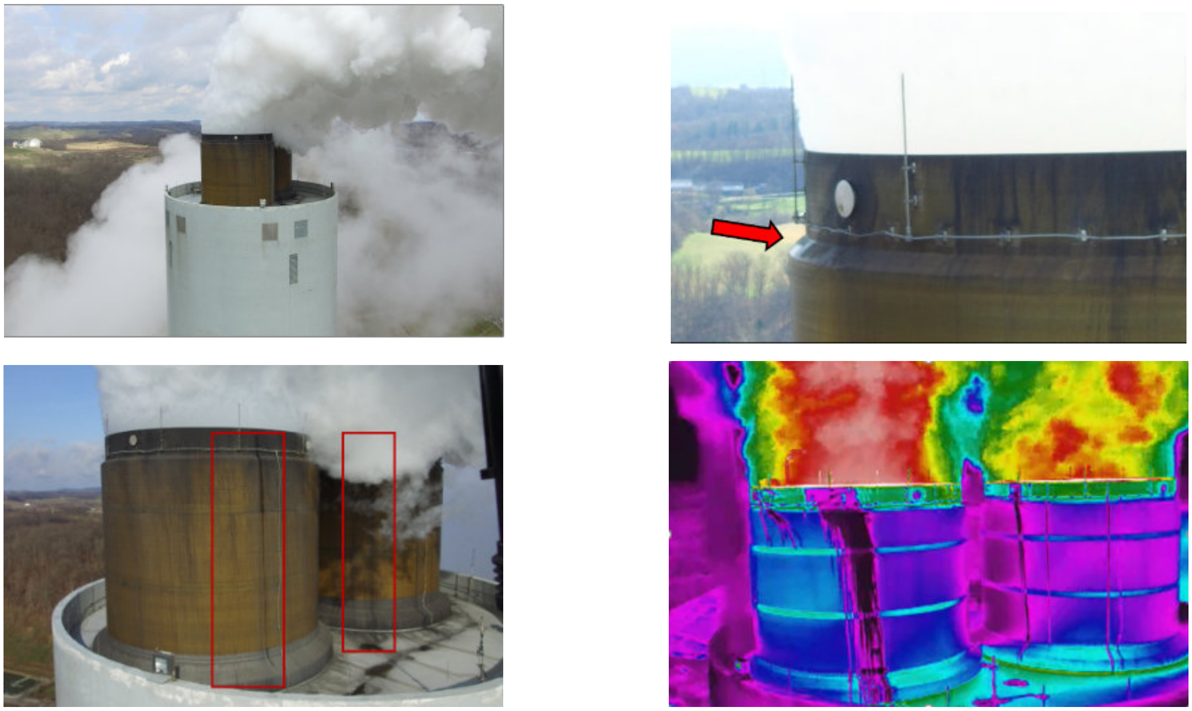

Figure 15: Moisture on the outside of the stack indicating liquid film running up the stack walls and shedding to droplets from the top edge

Modifications to Improve Droplet Capture

A series of modifications were developed in the CFD and physical models to improve droplet capture in the stack. These included gutter modifications, improved flow control in drainage areas, and protection of drainage paths from negative pressure zones. All these changes resulted in significantly increased liquid capture in the model results. The final design CFD film model results predicted that only 1.6% of the liquid exits the top of the stack, compared to almost 9.1% in the existing system.

Long Term Success

The plant installed the modifications as developed in the CFD and physical models in late 2017. Over the past 5 years of operation, there have been no additional reports of the droplet fall-out phenomena on the plant site and surrounding grounds.

ASC Bags Around the World

The ASC bags have traveled the world over the years and we've enjoyed collecting photos from employees and clients. Pictured left to right: sitting along the canals in Venice, kayaking in Missouri, on the beach in the Bahamas, gazing in wonder at Tahquamenon Falls State Park, and admiring Stonehenge.

Compressor Station Venting CFD Modeling

By: Dr. Kevin Linfield, P.E., P.Eng.

Natural gas compressor stations (Figure 1) are a hidden component of how many people heat their houses, dry their clothes, or cook their food. Usually located in rural areas, they often consist of stacks, vents, large sections of horizontal pipes, and power sources such as gas turbines to power the compressors.

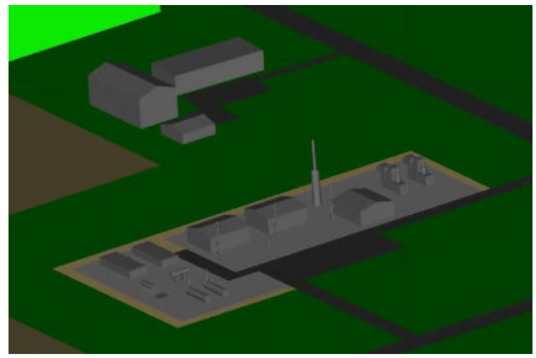

Figure 1: Compressor station in a rural area (left) and computer model of the compressor station (right)

During upset conditions or gas line purges, a small amount of natural gas is released to the atmosphere. Depending on the pressure and volume of the gas, these plumes may be dangerous if they intersect with nearby ignition sources, or displace oxygen for workers to breathe.

Many gas companies use CFD flow modeling to track the gas plumes for different ambient wind conditions. Modeling can be performed "steady state", simulating a continuous emission of gas, or "transient", where the volume of gas being vented varies with time.

Airflow Sciences recently performed external steady-state flow modeling of a natural gas compressor station to determine if vented gas would be ingested into a nearby pair of simple cycle gas turbines. This flow simulation using AzoreCFD (www.azorecfd.com) highlights the air flow patterns around the station, and tracks the concentration of natural gas as it mixes with atmospheric air.

Figure 2: Surrounding community

In order to accurately model the venting and dispersion process, a large portion of the surrounding area, including forests, roads, buildings, and rivers, needs to be incorporated into the CFD simulation (Figure 2). In some cases, the external model geometry may be over a mile in diameter and over 2000 feet high!

Historical wind speeds and directions are obtained, usually from local airports. Analysis of the wind rose in combination with the location of ignition sources or personnel areas determine the ambient wind conditions to model. Often, a series of simulations are undertaken to investigate "what-if" scenarios. CFD results featuring animations, streamlines, or color contours can show the path of the wind, and the effect of buildings, stacks, and other features can clearly be seen.

Airflow Sciences Corporation

Compressor Station Venting CFD Modeling

(Continued)

Along with the ambient wind conditions (Figure 3), the mass flow, temperature, pressure, and density of the natural gas play an important role in the plume height and throw. Modeling predicts the path of the natural gas, and streamlines help display this information. Here, for this particular wind condition, the natural gas plume (shown in magenta in Figure 3 and 4) carries over to gas turbine intakes - a potential hazard.

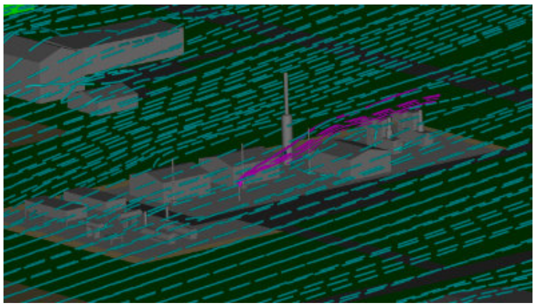

Figure 3: Ambient wind pathlines

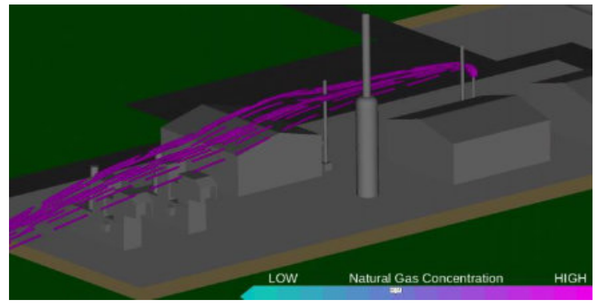

Figure 4: Natural gas plume carries over to gas turbine intakes

The gas is tracked and analyzed to determine if it is diluted sufficiently so no explosive or dangerous concentrations exist. AzoreCFD uses color bars to display the concentration values, and iso-surfaces allow a quick visualization of potential issues. Note the region of red and orange figure 5 - a potentially dangerous situation exists under this scenario. ASC would work with our client to offer solutions to mitigate the risk, which may include relocation of stack vents, increasing the height and/or diameter of the vents, or even stack geometry additions such as caps or diffusers.

Visit our YouTube channel to see a video animation of this compressor station model.

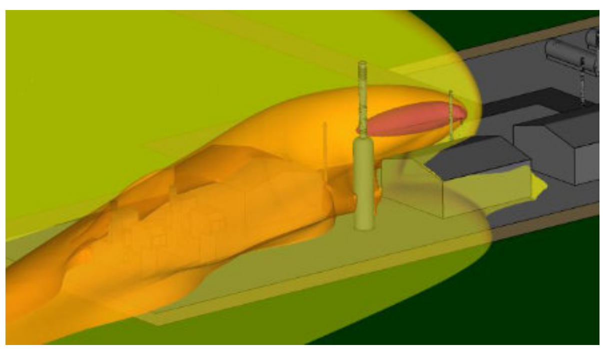

Figure 5: Dangerous concentration of natural gas shown in red and orange

Contacting ASC

ascinfo@airflowsciences.com

www.airflowsciences.com
www.airflowsciencesequipment.com
www.azorecfd.com

Corporate Headquarters
12190 Hubbard Street
Livonia, MI 48150
+1.734.525.0300

The Airflow Family of Companies

Engineering Consulting

Airflow Sciences Corporation

CFD Software

Azore® CFD

Flow Test Equipment

Airflow Sciences Equipment, LLC