Fall 2021

AIRFLOW SCIENCES CORPORATION

The Airflow Update

HVAC Design for Critical Cooling Applications

By: John Nitz, P.E. and Robert Mudry, P.E.

Many buildings house a significant amount of electronics equipment, including data centers, battery storage buildings, electricity converter warehouses, and bitcoin mining rooms. The electronics generate considerable amounts of heat, and the facilities need well-designed air-cooling systems to keep the servers and computer components from overheating. The air conditioning and ventilation system engineering for these facilities involves Computational Fluid Dynamics (CFD) modeling to optimize the cooling and protect the high-dollar electronic equipment.

An electrical converter warehouse in the eastern US required consistent cooling over a range of ambient weather conditions and equipment operating scenarios. The system converts AC to DC power and monitors voltage and amperage over a multi-state electrical transmission grid serving millions of homes. The heat generated by each converter cabinet needed to be dissipated to keep the room and the equipment temperatures consistent day and night. Because this warehouse is part of a critical infrastructure system, it warranted careful design of the cooling system to protect the equipment and ensure reliable operation.

Initial engineering calculations using American Society of Heating, Refrigerating and Air-Conditioning Engineers (ASHRAE) guidelines provided estimates for air flow rates and thermal loads, and CFD modeling was used to examine the 3D nature of the velocity and temperature patterns within the facility. Figure 1 shows the layout of the warehouse building as it was modeled for this project.

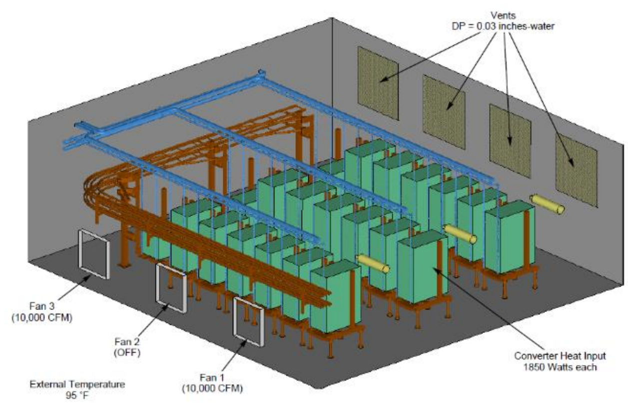


Figure 1. CAD model of warehouse system

Staff News

We are excited to announce the hiring of three new ASC employees: **Elliott Phillips**, Fluid Dynamics Engineer, **Kelly Hile**, Project Engineer, and **Courtenay Coleman**, Marketing Coordinator. Welcome aboard!

Quentin Minaker, Engineer, and new wife, Miranda, were married on October 2nd. Best wishes on your exciting new future together!

We're Hiring! We are looking for an experienced field services supervisor to help us expand our testing department. If that sounds like you or someone you know, we'd love to hear from you. Learn more on our website: airflowsciences.com/contact/jobs

The flow modeling was performed using the Azore® CFD program from Azore Software, LLC. This is a 3D polyhedral CFD tool that includes flow and heat transfer simulation.

Initial Model Results

Initial CFD modeling evaluated the efficacy of using two large blower fans on one wall of the warehouse, with four exhaust vents on the opposite wall. Three fans would be installed, but only two would be in use at any given time. These fans would draw outside ambient air into the room with the hope of keeping temperatures below a critical level. The model simulated a 95°F (35°C) day. Although this model did show that air exiting the warehouse through the exhaust vents was at roughly the temperature expected based on the ASHRAE calculations, it also revealed a stratified temperature distribution within the warehouse that could lead to unacceptably high temperatures in some areas.

Figure 2 shows a cross-section of the temperature profile throughout the room at the fan centerline elevation. This simulation predicted that the average air temperature in the room was 103.3°F (40°C), and peak temperatures exceeded 125°F (52°C). CFD results indicated that this circulation system did not offer adequate cooling for all the equipment to operate safely and reliably.

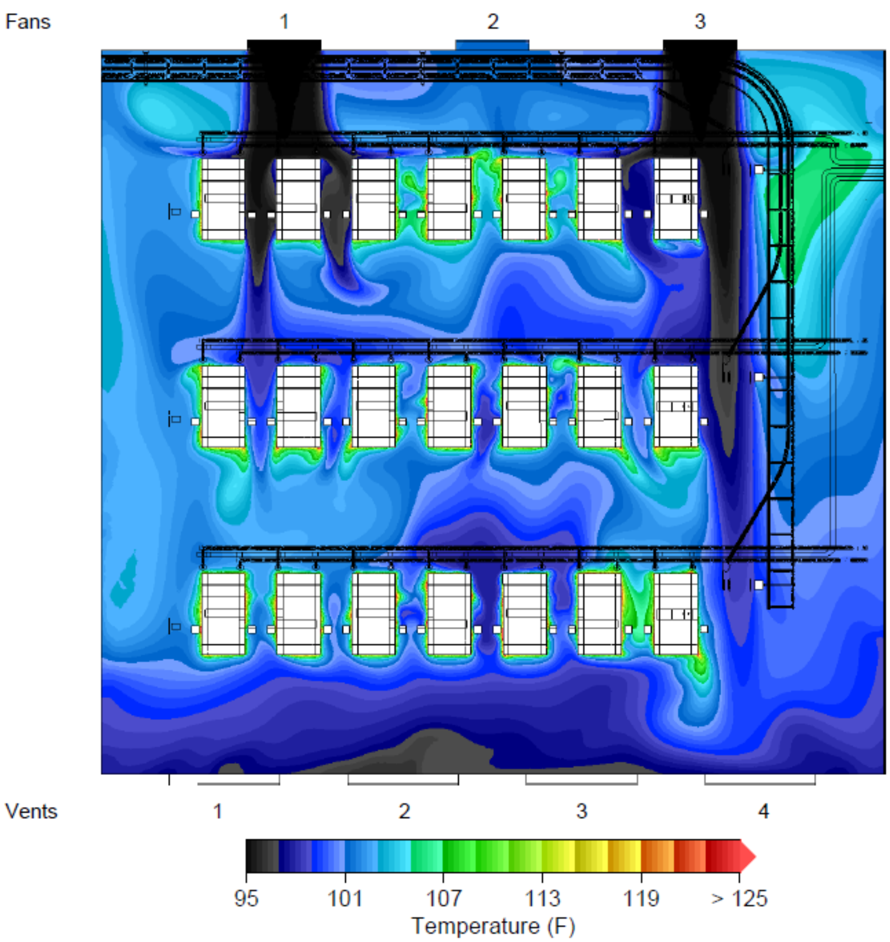


Figure 2. Air temperature distribution for initial design

HVAC Design for Critical Cooling Applications

(Continued)

A redesign of the cooling system focused on adding HVAC cooled air into the room to further reduce maximum temperatures. The original blower fans and exhaust vents were removed from the model. Multiple CFD simulations of various system designs provided the data required to optimize the positioning of the inlet and exhaust vents, size the air conditioning system and fans, avoid local hot spots, and minimize stagnant flow areas.

Final Design Modeling

The final design that achieved the customer's specifications included six inlet vents that injected 75°F (24°C) air into the warehouse 2' (0.6m) above floor level. A single return vent was placed 11' (3.4m) above the floor on the same wall. This design reduced peak temperatures significantly and allowed for consistent and reliable equipment operation. The average air temperature in the room was 89.7°F (32°C) for the final design. Figures 3 and 4 show the regions with air temperature above 110°F (43°C) in the initial simulation and the final design.

This converter warehouse designed with Azore has been operating successfully since fall 2019.

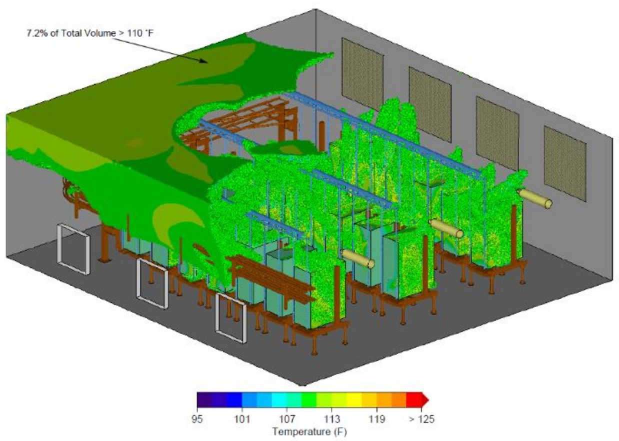


Figure 3. Percentage of air hotter than 110°F (43°C) initial design

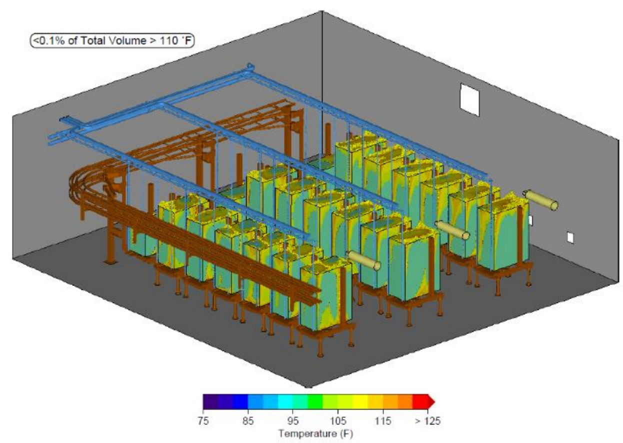


Figure 4. Percentage of air hotter than 110°F (43°C) final design

A video animation of the flow patterns in this warehouse can be found on the Azore Software website at the following link: https://www.youtube.com/watch?v=dgTH3jsQNjQ

Innovations at ASC

ASC personnel had a big year for patents, with inventors Brian Dumont, P.E., Jim Paul, P.E., Matt Fleming, and Paul Harris, Ph.D. collaborating on four US patents related to velocity measurement for moving vehicles:

- 11,061,048 Systems and methods for communicating information associated with wind pressures
- 10,935,564 Systems and methods for determining wind velocity
- 10,921,344 Pressure sensing probe
- 10,921,343 Systems and methods for converting wind pressure to wind velocity

Also, ASC received an additional grant for R&D from the US Small Business Innovative Research (SBIR) program:

- Advanced Measurement Probe System for Non-Nulling Stack Velocity Testing

Airflow Sciences Corporation

Medical Campus Laboratory Plume Dispersion Modeling

By: Jeff Everett and Paul Harris, Ph.D.

Introduction

A primary consideration in the design of building ventilation systems and exhaust stacks is to ensure that the stack exhaust plume is adequately dispersed into the atmosphere. Particularly with multi-story buildings and skyscrapers, the aerodynamic conditions in the vicinity of the building can vary widely under different wind speed and weather conditions. Proper exhaust stack design must ensure that the stack extends high enough above the top of the building, and expels gas at an appropriately high velocity, to minimize the risk of downdraft that could pull the exhaust plume back toward the building ventilation intakes, or toward the ground or other pedestrian walkways. While several industry standards and stack design parameters have been developed over time to aid in this process, Computational Fluid Dynamics (CFD) modeling has become a powerful tool in analyzing the plume behavior under a range of external conditions, considering the actual building geometry, the influence of surrounding buildings and other structures, and the local topography.

Background

At a medical campus research center, the small amounts of gases released from laboratory experiments are collected in fume hoods and exhausted through vent stacks on the roof of the building. Figure 1 shows a close-up view of the building and vent stack arrangement.

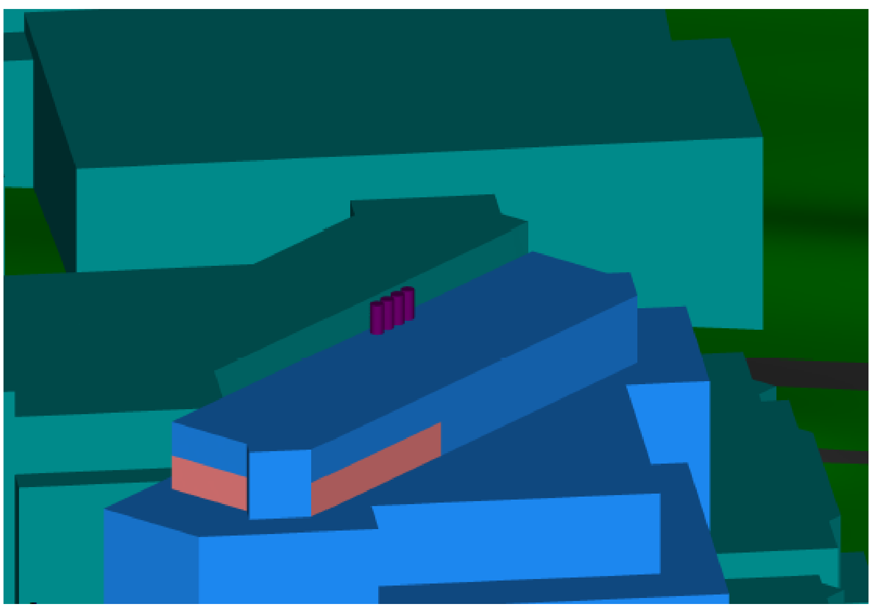
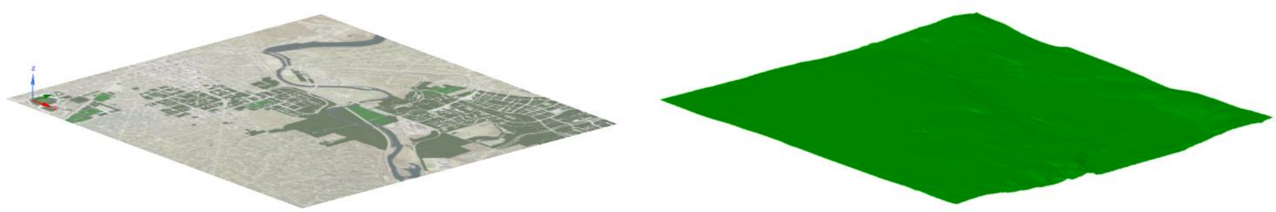


Figure 1. Research building roof and exhaust stack geometry (Building intake louvers in orange)

As part of a recent upgrade project, the facility planned to replace existing vortex dampers in the exhaust system with variable-speed drives to modulate the exhaust fan flow rates. As part of the re-design, a dispersion study was desired to evaluate the exhaust plume behavior over a range of potential system flow rates and aid in developing an optimized fan duty cycle. A survey of the nearby area highlighted several potential concerns, including building intake vents, pedestrian gathering places, and parking garages. There was concern that these gases might be blown down to ground level or be entrained into other nearby buildings before the gas concentration had been diluted to a harmless level. Given the variability of the wind, the variety of nearby locations the gases might travel to, and the low concentration of the gases involved, physical testing to determine the gas dispersion would have been time consuming and costly. Therefore, Computational Fluid Dynamics (CFD) modeling was chosen to analyze the plume behavior under a range of different ambient wind conditions and fume hood gas flow rates.

CFD Model Development

A three-dimensional CFD model was constructed of the exterior of the laboratory buildings, the nearby medical campus, and the surrounding community. A combination of 2D campus maps, 3D building CAD of campus buildings, and Google Earth 2D/3D measurement tools were utilized to develop the geometry of buildings, trees, and the overall layout of the terrain. Topological data of the ground elevation, which varied significantly across the model domain, was imported from USGS survey records, and integrated with the building geometry to ensure that each building's overall height was correctly positioned relative to the ground elevation. Figures 2 and 3 provide details of the terrain and CFD model geometry.



(a) *(b)*

Figure 2. 2D aerial map (a) and topological surface generated from USGS data (b)

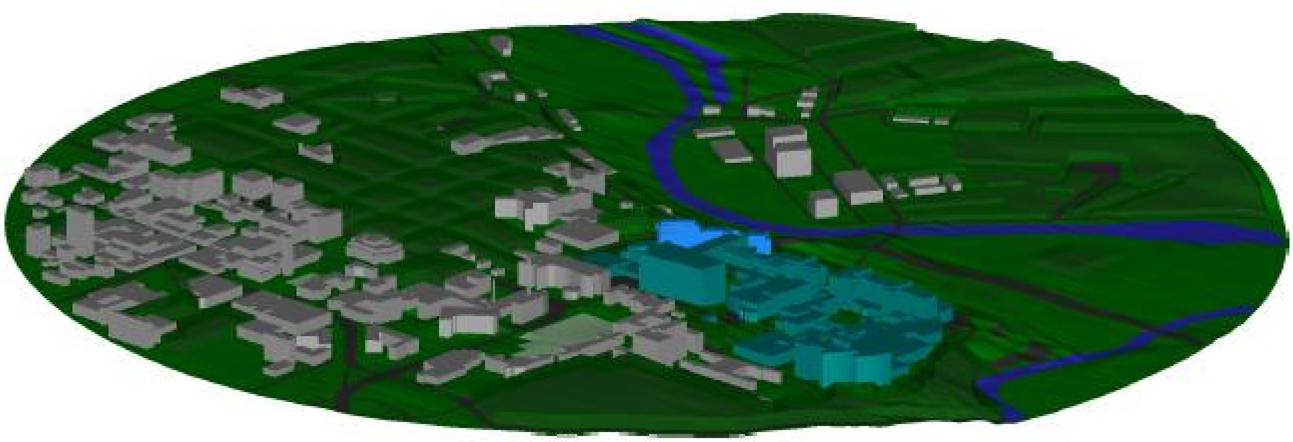
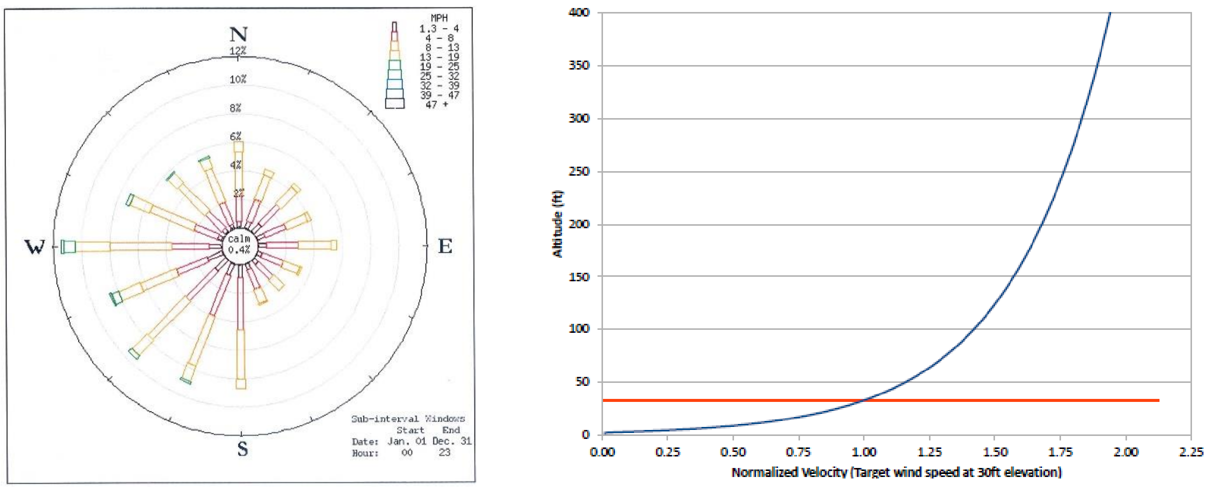


Figure 3. Completed CFD model geometry

Over 100 million computational cells were included in the CFD mesh. The mesh had a great variation in length scales; the domain was nearly a mile across, but also included the detail of the 48-inch exhaust stacks on the roof of the laboratories, with carefully selected cell sizing parameters to accurately track the plume dispersion. The model inlet boundary condition was applied around the outside perimeter of the domain – a circular domain with multiple inlets was constructed to allow the wind to enter the model from any direction. A wind rose map (Figure 4a) displays the most common wind conditions for a particular area and is used as a guide for selecting simulation scenarios. The wind velocity distribution is applied in the form of an atmospheric boundary layer (ABL) profile (Figure 4b). Numerous methods exist for calculating the shape of the ABL, but it is generally accepted that the profile curve is equal to the “prevailing wind speed” at a point located about 30 feet off the ground – this corresponds to the typical elevation of an anemometer or other measuring device at local weather stations.



(a) *(b)*

Figure 4. Meteorological wind rose (a) and calculated atmospheric boundary layer profile (b)

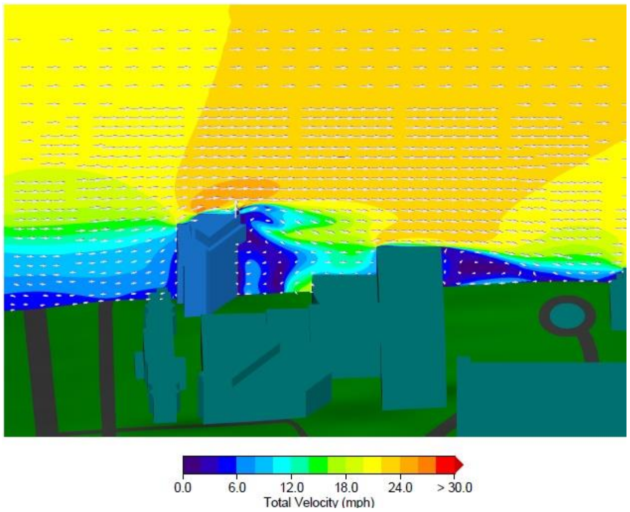
Although the model construction allows simulation of any wind direction and velocity, typical analyses focus on a combination of the prevailing wind directions as reported by the wind rose, as well as any other condition that could result in a sub-optimal dispersion pattern due to the position of the exhaust location relative to nearby buildings or other features that could influence the aerodynamics.

Using the Azore® CFD solver, air flow patterns were calculated across the domain based on several different wind speeds and directions. The exhaust gases emitted from the stacks on the roof of the laboratory were also tracked as a separate species. The HPC parallel processing of the Azore solver allows for the timely calculation of multiple solutions with varying inputs, even with many computational cells. For these simulations, two computers, each with 18 CPUs, 36 cores and 384 GB of RAM, were utilized in parallel.

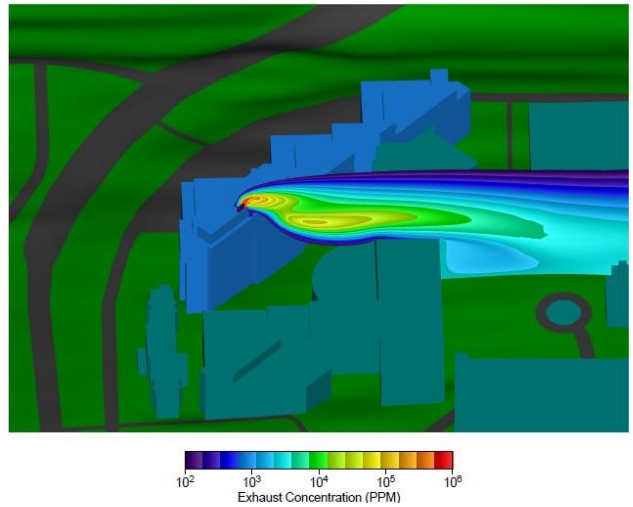
Once the CFD simulations had converged, Azore's integrated post-processing tool was used to quantify the gas concentration at several critical locations and visualize the dispersion of the gas species spacially. The post-processing workflow includes generation of gas concentration statistics, color contour plots, and animations, all of which can be scripted within Azore for easy, accurate, and repeatable analysis of multiple cases. Visualization of the gas dispersion using color contours and path lines superimposed on the geometry of the medical campus buildings clearly communicates the flow patterns and gas concentration levels at key locations. Example plots from one of the CFD model simulations are displayed in Figure 5 below.

Medical Campus Laboratory
Plume Dispersion Modeling

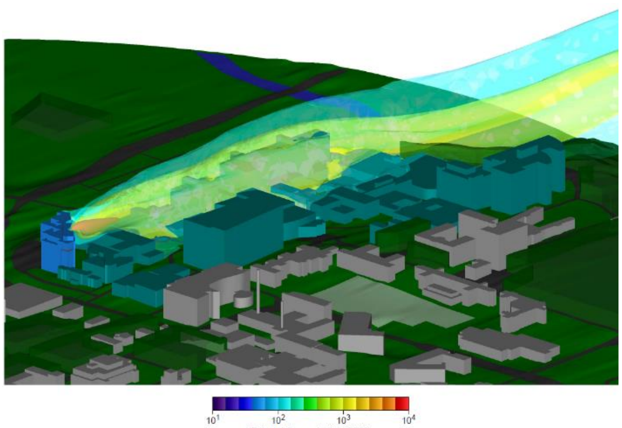
(Continued)



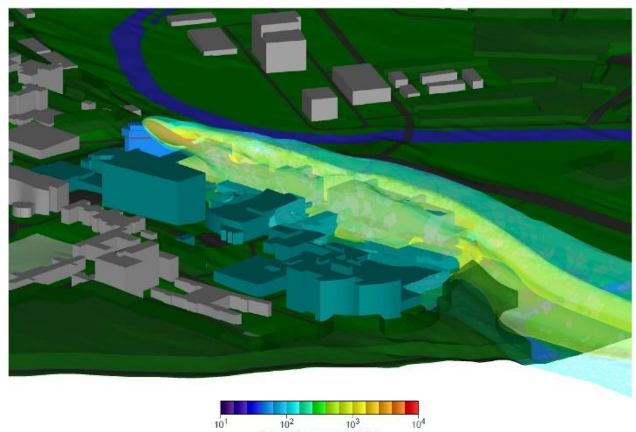
(a)



(b)



(c)



(d)

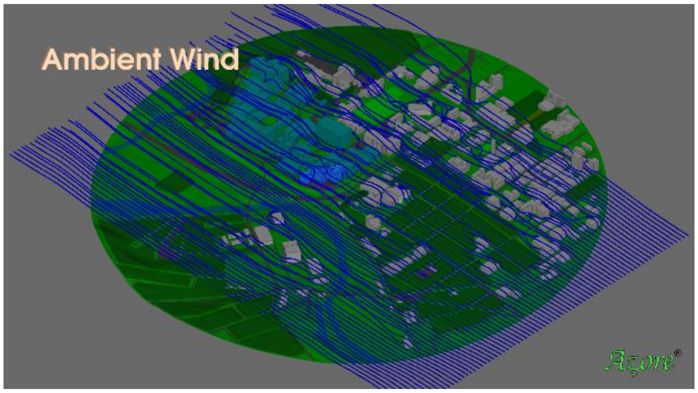
Figure 5. CFD color contour plots of (a) wind velocity, (b) exhaust concentration, (c)-(d) exhaust concentration iso-surfaces

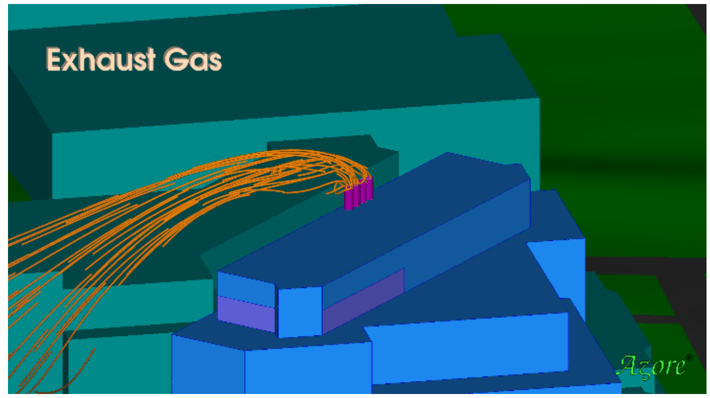
Update on Azore CFD, Practical CFD You Can Trust

Our staff are continuing to develop and make enhancements to the transient analysis and data visualization functions in Azore CFD. Watch for a new release coming soon!

Medical Campus Laboratory
Plume Dispersion Modeling

(Continued)





(a) *(b)*

Figure 6. Streamline plots of (a) ambient wind and (b) exhaust gas

In the simulation shown in Figures 5 and 6, the wind accelerates over the west side of the building and then separates, creating a re-circulation zone immediately downstream of the exhaust stacks. The exhaust spreads vertically in this wake, allowing a region of non-zero concentrations to remain close to the ground as it flows to the east. A typical design modification to address this would involve a re-design of the stack to either increase the height of the exit plane above the roof, or increase the exit velocity of the exhaust, such that the exhaust enters the ambient air in the orange region of the wind profile and avoids the re-circulation zone.

Overall, for this analysis, six different CFD simulations were conducted with different combinations of wind speed, wind direction, and exhaust stack flow rate. The results were utilized to help the facility operations department determine an appropriate VFD duty cycle to ensure that the exhaust emissions did not exceed dangerous concentrations of contaminants in the vicinity of the research building or other nearby locations. The results also illustrated the sensitivity of the exhaust dispersion to ambient conditions, and the potential need to correlate the exhaust system control strategy with the wind speed and/or direction.

A video animation of the flow patterns in this simulation was also created using the Azore CFD software. The animation can be viewed at the following link: https://www.youtube.com/watch?v=KmlNka_3aNE

Contacting ASC

asc@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