ANL/NE-16/4

Development and Validation of Methodology to Model Flow in Ventilation Systems Commonly Found in Nuclear Facilities – Phase II

Specialized Technical Support to the Nuclear Safety Research and Development Program

Nuclear Engineering Division

About Argonne National Laboratory
Argonne is a U.S. Department of Energy laboratory managed by UChicago Argonne, LLC under contract DE-AC02-06CH11357. The Laboratory's main facility is outside Chicago, at 9700 South Cass Avenue, Argonne, Illinois 60439. For information about Argonne and its pioneering science and technology programs, see www.anl.gov.

DOCUMENT AVAILABILITY

Online Access: U.S. Department of Energy (DOE) reports produced after 1991 and a growing number of pre-1991 documents are available free via DOE's SciTech Connect (http://www.osti.gov/scitech/)

Reports not in digital format may be purchased by the public from the National Technical Information Service (NTIS):
U.S. Department of Commerce
National Technical Information Service
5301 Shawnee Rd
Alexandria, VA 22312
www.ntis.gov
Phone: (800) 553-NTIS (6847) or (703) 605-6000
Fax: (703) 605-6900
Email: **orders@ntis.gov**

Reports not in digital format are available to DOE and DOE contractors from the Office of Scientific and Technical Information (OSTI):
U.S. Department of Energy
Office of Scientific and Technical Information
P.O. Box 62
Oak Ridge, TN 37831-0062
www.osti.gov
Phone: (865) 576-8401
Fax: (865) 576-5728
Email: **reports@osti.gov**

Disclaimer
This report was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government nor any agency thereof, nor UChicago Argonne, LLC, nor any of their employees or officers, makes any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise, does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government or any agency thereof. The views and opinions of document authors expressed herein do not necessarily state or reflect those of the United States Government or any agency thereof, Argonne National Laboratory, or UChicago Argonne, LLC.

***Report No.**

Development and Validation of Methodology to Model Flow in Ventilation Systems Commonly Found in Nuclear Facilities

Specialized Technical Support to the Nuclear Safety Research and Development Program

Philip Strons, James L. Bailey, John Davis, James Grudzinski, and John Hlotke
Nuclear Engineering Division, Argonne National Laboratory

March 2016

1. Phase II – Modeling of Hot Cell Facilities

Understanding and modeling of particulate transport has importance related to safety in ventilated environments that contain hot cells, gloveboxes, and fume hoods. In particular, good modeling of the airflow in these spaces is required to understand areas of stagnation where particles drop out of the free stream and accumulate. This accumulation of particulate can cause either contamination and/or higher than acceptable local dose levels.

It is known that multiple sites across the DOE complex take credit for hot cells, gloveboxes, and/or hoods in their safety basis for providing a defense-in-depth benefit for both onsite and offsite releases. By providing confinement of radioactive materials, such features serve to reduce direct doses to facility workers and mitigate the consequences to the environment due to an uncontrolled release. Each of these features has access points that interface with the personnel space. Understanding how air flow behaves at these access points is of great interest to those performing hazard analysis.

Modeling the air flow behavior for glovebox and hot cell systems has been performed through the use of Computational Fluid Dynamics (CFD). The work followed an iterative process that began with CFD modeling of airflow in a glovebox or hot cell facility ventilation system. Field validation studies, using smoke releases for flow visualization, were conducted at existing facilities located on site at Argonne National Laboratory to both inform and validate the modeling assumptions. Based on the results of field tests, modeling assumptions and boundary conditions are refined and the process is repeated until the results are found to be reliable with a high level of confidence.

The work for this project was completed in two phases. The first phase of the work focused on the analysis of flow related to glovebox accident scenarios. Continuing this work into Phase II, we extended and built upon the results of analyzing glovebox systems during Phase I and applied this approach to the larger and more complex geometry of a hot cell facility.

In this report we present the results of the Phase II analysis and testing of the flow patterns encountered in the Alpha Gamma Hot Cell Facility (AGHCF), as well as the results from an opportunity to expand upon field test work from Phase I by the use of a Class IIIb laser. The addition to the Phase I work is covered before proceeding to the results of the Phase II work, followed by a summary of findings.

2. Improving glovebox field tests using particle image velocimetry

Using a laser to illuminate smoke was considered to be an excellent aid in flow visualization from the beginning of the project; however, initial results with a Class II laser with less than 1 mW of power were found to be unsuitable for photography. Although the Class II laser appears bright enough to the human

eye, it has insufficient brightness to record still images via camera at a high frame rate. A brighter laser was purchased, but the vendor was not able to deliver the laser in time for a test to be conducted and included in the Phase I report. However, a window of opportunity appeared after submitting the report for Phase I to conduct field tests with the small glovebox using a Class IIIb 500 mW laser for in-field particle image velocimetry (PIV).

2.1. Brief overview of the PIV method

Conceptually, PIV is a simple process comparing sequential images of smoke particles, and with the aid of image analysis in software, identify velocity vectors based on the difference between images. In practice, though, the task presents technical challenges in terms of providing enough laser light to illuminate the smoke particles, so that a camera is able to capture a sequence of adequately clear images for analysis.

There are also a couple of points to remember when examining the results from PIV. First, the images depict instantaneous velocity and do not always agree with the average velocity results from a steady-state CFD analysis. Also important is the fact that, with the laser illuminating only a 2-D plane, smoke moving in an out-of-plane direction can, at times, create misleading imagery.

2.2. Preliminary CFD model

The same glovebox and its corresponding CFD model developed during Phase I were used for the PIV test. The geometry of the small glovebox used in this test follows the standards outlined by the American Glovebox Society. A diagram from the AGS standard and photo of the actual glovebox are presented in Figure 1. The CFD model geometry and an example of calculated streamlines are shown in Figure 2.

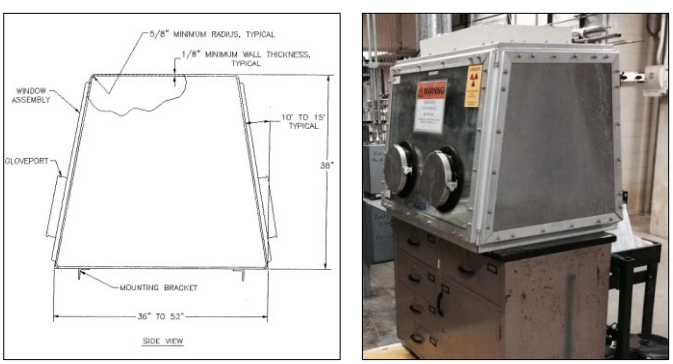

Figure 1. A diagram from the AGS standard and a photograph of the actual glovebox to be used in the in-field PIV test.

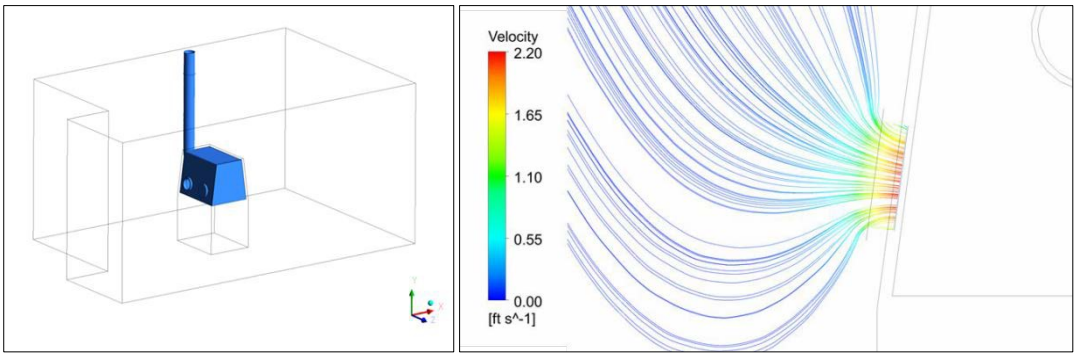

Figure 2. The data from the CFD model used in Phase I is used for comparison with the in-field PIV test. The CFD result plot, on the right, shows streamlines as air enters an open gloveport.

2.3. Initial PIV field test

The overall setup for the trial in-field PIV test was similar to previous tests in which smoke was photographed using only 1500 W halogen lights. After adjusting the laser to shine directly along the centerline of the open gloveport, the camera was aimed perpendicular to the 2-D laser sheet. The laser sheet is produced by passing the beam through an adjustable cylindrical lens. The lens allows the laser sheet to be rotated and also to adjust the angle of divergence. Minimizing the divergence of the beam to cover only the region of interest maximizes the illumination of the smoke. Figure 3 shows the test setup used and a laser image superimposed on a CFD result plot to show the size of the sheet with respect to the glovebox.

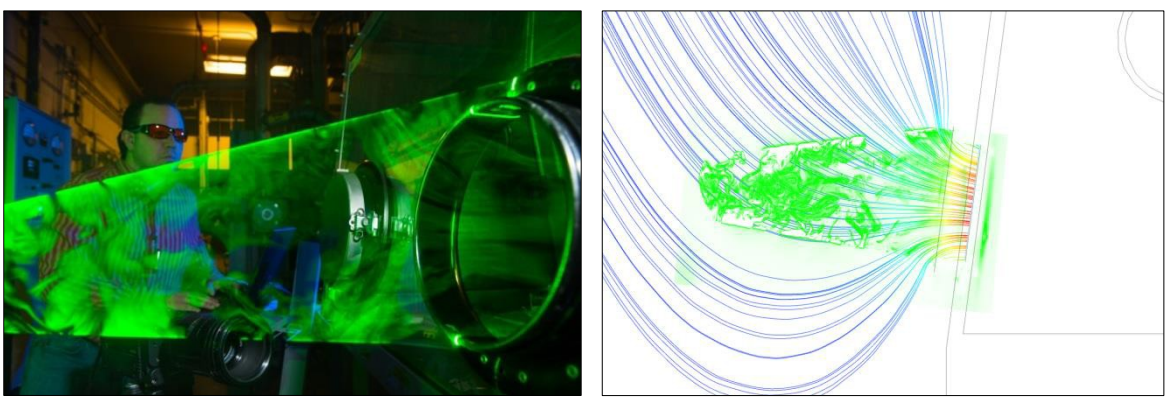

Figure 3. The overall setup for PIV is conceptually simple. The laser is adjusted to shine directly along the centerline of the open gloveport with the camera aimed perpendicular to the 2-D laser sheet. The angle of divergence for the laser sheet is adjusted to cover only the region of interest, maximizing the brightness. On the right, a laser image is superimposed on the CFD result plot.

2.4. Comparing PIV test to CFD results

Figure 4 shows two consecutive photo frames taken during the PIV test. Streamlines from the previous CFD model are scaled and adjusted to match the size and location of the photographed gloveport. A simple grid was also added for easier visual comparison. Near the gloveport opening, calculated streamlines match the patterns found in the illuminated smoke extremely well. Examining the center of each image, however, we can see swirls of smoke have an upward velocity component not present in the

streamlines of the CFD model.

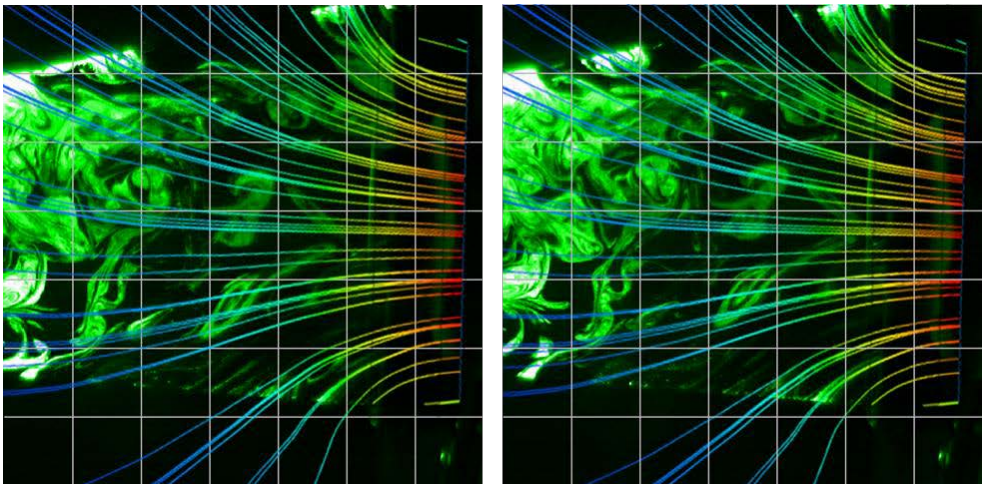

Figure 4. Two sequential frames compared side by side. The flow is similar to the streamlines predicted by CFD, but there is an upward velocity component in the smoke. Introducing the smoke modified the flow outside the glovebox. Using a tent around the glovebox, adequate smoke could be introduced without affecting normal flow conditions.

One major challenge encountered during the test was trying to introduce enough smoke into the laser sheet without affecting the flow of air being drawn into the opening of the glovebox. There was a balancing act of producing a sufficient volume of smoke that was dense enough to show up on camera while attempting to minimize the velocity of the smoke as it came out of the nozzle. For this particular setup, affecting the normal flow patterns near the gloveport was unavoidable. However, this apparent problem during testing produced interesting results from the PIV software.

Typically in PIV, a much brighter laser is used to produce precisely timed pulses. The increased level of light illuminating the smoke and the higher precision of the timing allow the camera to examine a smaller field of view and capture images of individual particles. The laser used in this test is not designed to produce carefully timed pulses. Therefore, the magnitude of the velocity vectors could not be calculated with reliable accuracy via software analysis of the images. Direction of the vectors was still shown to be accurate. Figure 5 shows two versions of a single photo from the PIV test, the lower image includes the velocity vectors calculated by image analysis.

Discrepancies between CFD and PIV results begin to occur just a few inches away from the gloveport where average velocities drop below approximately 1 ft/sec. Despite the discrepancies, this first run with in-field PIV showed potential merits. Although image analysis was unable to obtain velocity magnitudes, flow direction was easily discerned, producing an expedient tool for comparing field test data to CFD results. Also, recall that the primary interest in analyzing glovebox flow is the ability to identify flow direction at the gloveport.

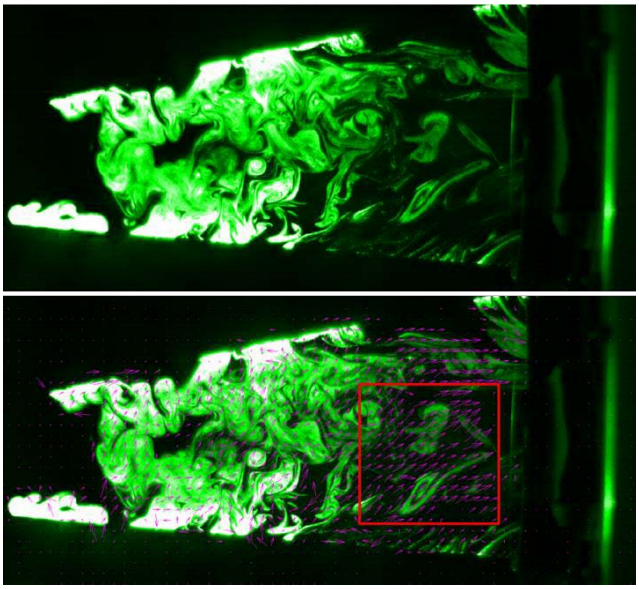

Figure 5. Both images presented in this figure are the same photo frame. The top is the raw image. The lower image shows velocity vectors, computed by PIV software, overlaid on the original image. The area outlined in red corresponds to the area shown in Figure 6.

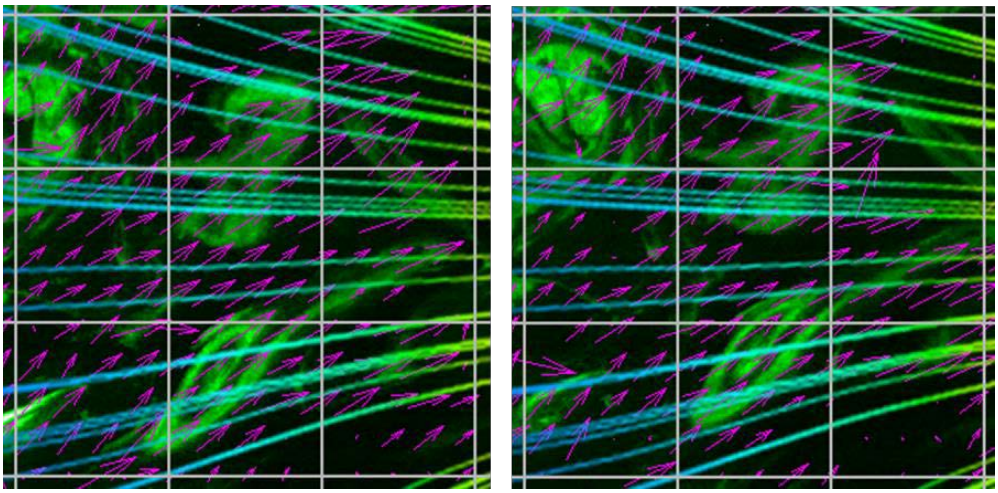

Figure 6. Using the same sequential images presented in Figure 4, and enlarging the area outlined in red in the previous figure, the velocity vectors from the PIV image analysis reveal the upward velocity in the smoke with greater clarity than relying on the unaided eye.

2.5. Comparing to previous test methods

With the brighter laser sheet illuminating a specific plane, a direct comparison to the CFD model results becomes more straightforward. The overall improvement in visualizing the actual flow and matching the captured image with the CFD results is readily apparent in Figure 7. The laser sheet also allows for the calculation of velocity vectors with PIV software. However, improvements still need to be made to eliminate interference with the normal flow patterns caused by the introduction of smoke.

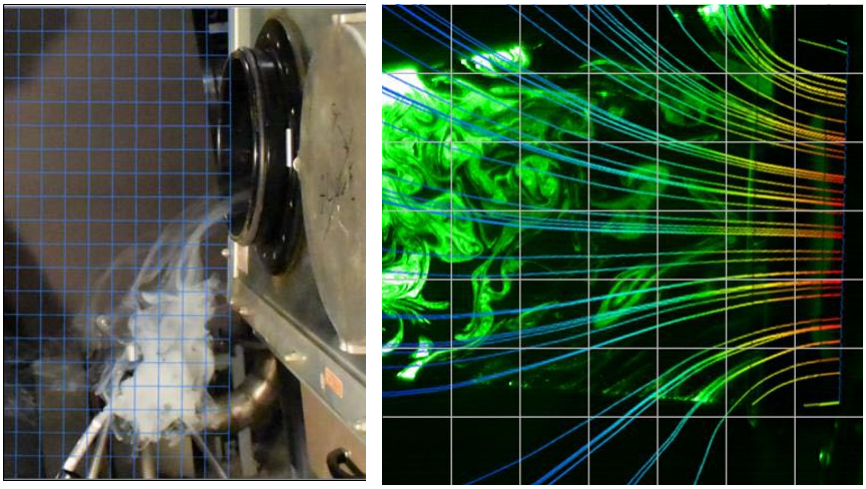

Figure 7. This sequence of photos, taken while viewing the duct from the side, shows the final result from development of flow visualization with smoke and photography. The earlier technique that simply relied on bright lighting is shown on the right, the benefits of using a laser sheet are clearly visible in the image on the right.

3. CFD analysis and field test for the Alpha Gamma Hot Cell Facility

Although the work from Phase I provided valuable experience on in-field testing of real world ventilation safety systems, additional challenges emerged as the work moved to the analysis of the AGHCF ventilation system. The hot cell facility, shown below in Figure 8, covers an area of approximately 10,000 sq ft and includes offices, corridors, two major work areas, plus additional adjacent rooms for specialized gloveboxes and other equipment. Ceilings vary in height from 9 to 13.5 feet. Modeling the flow of air through the AGHCF meant including enough of the facility to cover the ventilation system from all supply vents to all exhaust vents. The model included 13 supply vents and 12 exhaust vents, most of the vents are visible in Figure 8 below. The geometry was based on architectural drawings for the building, with supply and exhaust flow information taken from as-built ventilation diagrams.

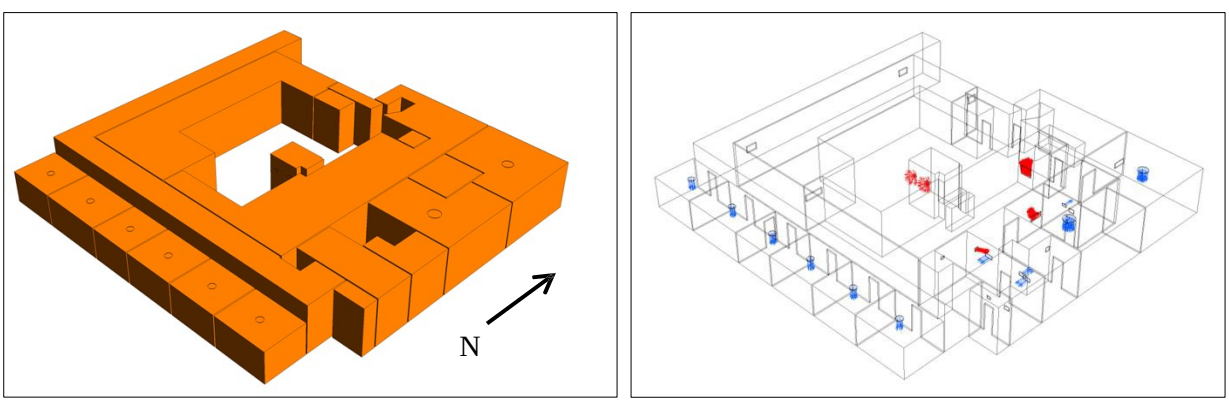

Figure 8. Overall geometry of the AGHCF covers an area of nearly 10,000 sq ft with ceilings that vary from 9 to as high as 13.5 ft. The AGHCF has a total of 13 supply and 12 exhaust vents, as shown on the right, with blue indicating supplied air and red indicating exhausted air.

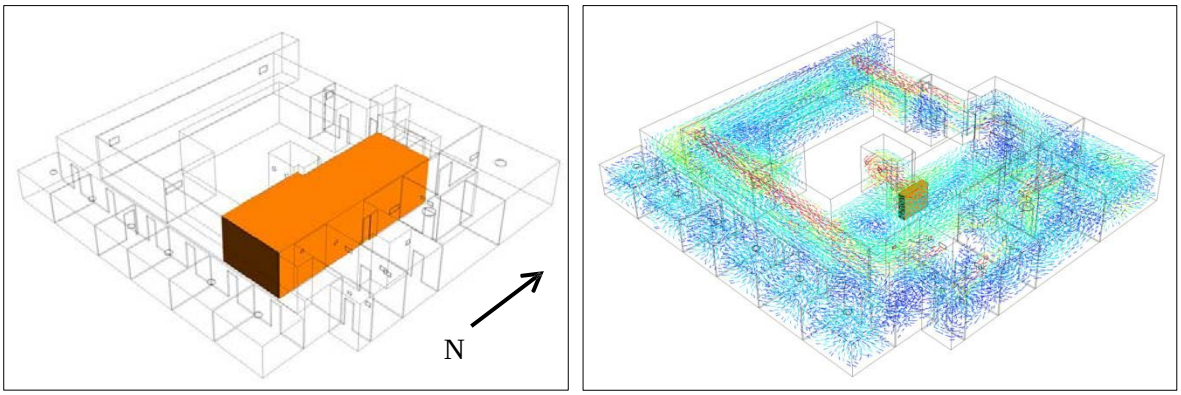

Figure 9. To reduce the scope of testing to a reasonable size, the area evaluated in this report is highlighted in the image on the left. As an illustration of the complexity and large amount of data produced by the model, a velocity vector field for the entire facility is shown on the right.

In order to establish an area of reasonable size conducive to conducting field tests by video recording smoke releases, the scope of the problem was reduced to focus on the CFD results of the work area (Figure 9) adjacent to the Clean Transfer Area (CTA), which is an intermediary space for moving items in and out of the hot cell. A large shielded door isolates the work area from the CTA. This work area was selected because the exhaust vents in the CTA maintain a negative pressure relative to the work area, and opening or closing the door allowed for testing of multiple flow patterns.

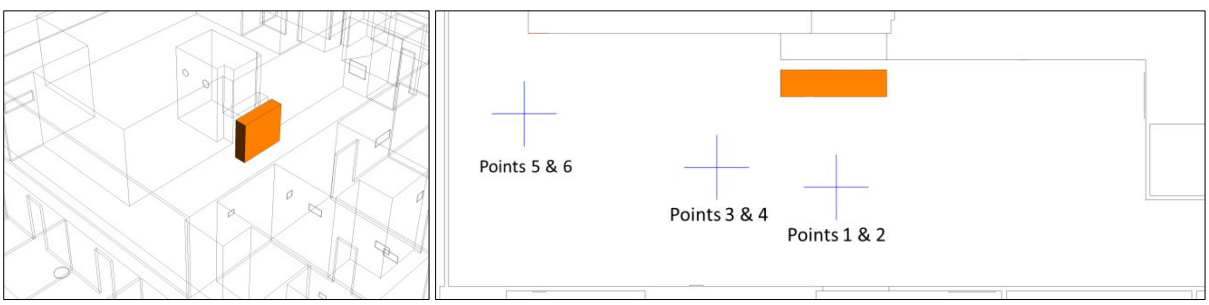

Figure 10. A 3-D view of the work area to be examined for field tests shows the CTA door (orange) in a partially open position. On the right, a plan view of the work area is marked with blue crosshairs indicating the locations to be used for field test smoke releases.

The work area used for comparing CFD results with field test data is approximately 1,000 sq ft and is presented in Figure 10 with both a 3-D view and a plan view. The CTA door is highlighted in orange as a reference point when viewing plots of CFD results, and the locations to be used as smoke release points in field tests are identified by blue crosshairs. The method of releasing smoke is a standard procedure used in industrial hygiene to verify the direction and speed of airflow. The smoke release points were selected based on preliminary results of the CFD model and on the positions that could be easily located within the hot cell with simple measurements using a tape measure.

3.1. Preliminary CFD model results

CFD analysis was completed for three different CTA door positions, including: closed, open 33 inches, and open 66 inches. The CTA door open at 33 inch configuration happened to yield the best set of data during field tests. The remaining results presented in this report cover only this scenario due to time

constraints in comparing field data with CFD model results. Plots for the paths of simulated smoke releases are shown below in Figure 11. For all six points, there is an initial downward motion of the smoke with horizontal motion from the left side to right side of the images.

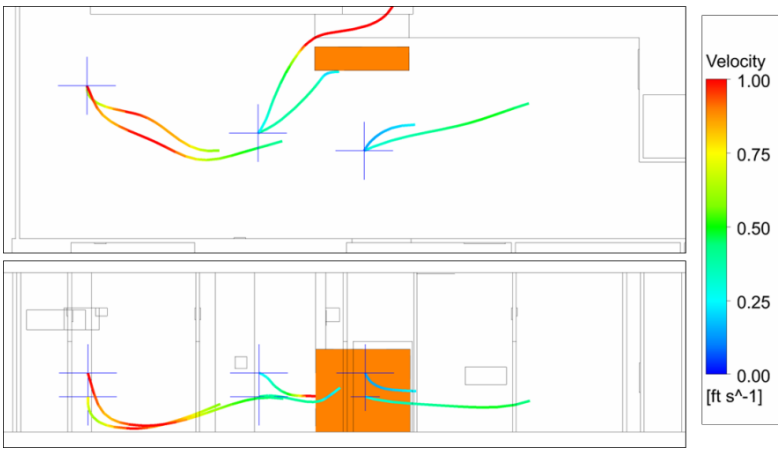

Figure 11. Results from the preliminary model of AGHCF work area, showing streamlines starting from the smoke release points shown in Figure 10.

3.2. AGHCF field test with smoke releases

The previous methods used in flow visualization for gloveboxes, such as high-speed still photography or overlaying a grid on photos to measure relative motion between photo frames, were not applicable to the larger scale of the AGHCF. Smoke releases were captured on video for later review on a qualitative basis in comparison to the CFD results. A stopwatch and tape measure provided an approximate average velocity of the smoke. Hot-wire anemometer readings were made and recorded as well.

Before releasing any smoke, the first measurement made in the field test was a survey of the area with an IR thermometer to check for any significant temperature differences within the work area. Walls, floor, and ceiling were all within 1 °C. After checking temperatures, smoke releases were made at the points indicated in Figure 10 at elevations of both 3 ft and 5 ft above the floor with the CTA door at the three different positions analyzed in the initial model. A total of 18 smoke releases were made and recorded on video. The field test velocity data is summarized in a table in the appendix.

Although velocity magnitudes measured in the field were similar to CFD calculations, the observed flow patterns differed considerably. Figure 12 shows the results of smoke released in front of the CTA door with the door opened 33 inches. If one were to view the smoke flow patterns from above, the horizontal path agreed with calculated flow directions. However, unlike the initial CFD model results, there was no downward motion immediately after the release. All smoke releases exhibited either a pattern of an initial upward motion or no vertical motion.

Figure 12 presents the motion of the smoke releases in front of the CTA door. The measured average velocities were approximately 0.5 ft/sec, similar to the calculated values. The smoke moved primarily in a horizontal direction from left to right when facing the CTA door. The release from 5 ft above the floor

traveled several feet without any observable vertical motion before beginning to both disperse and move downward. The release from 3 ft above the floor had no apparent vertical motion before diffusing.

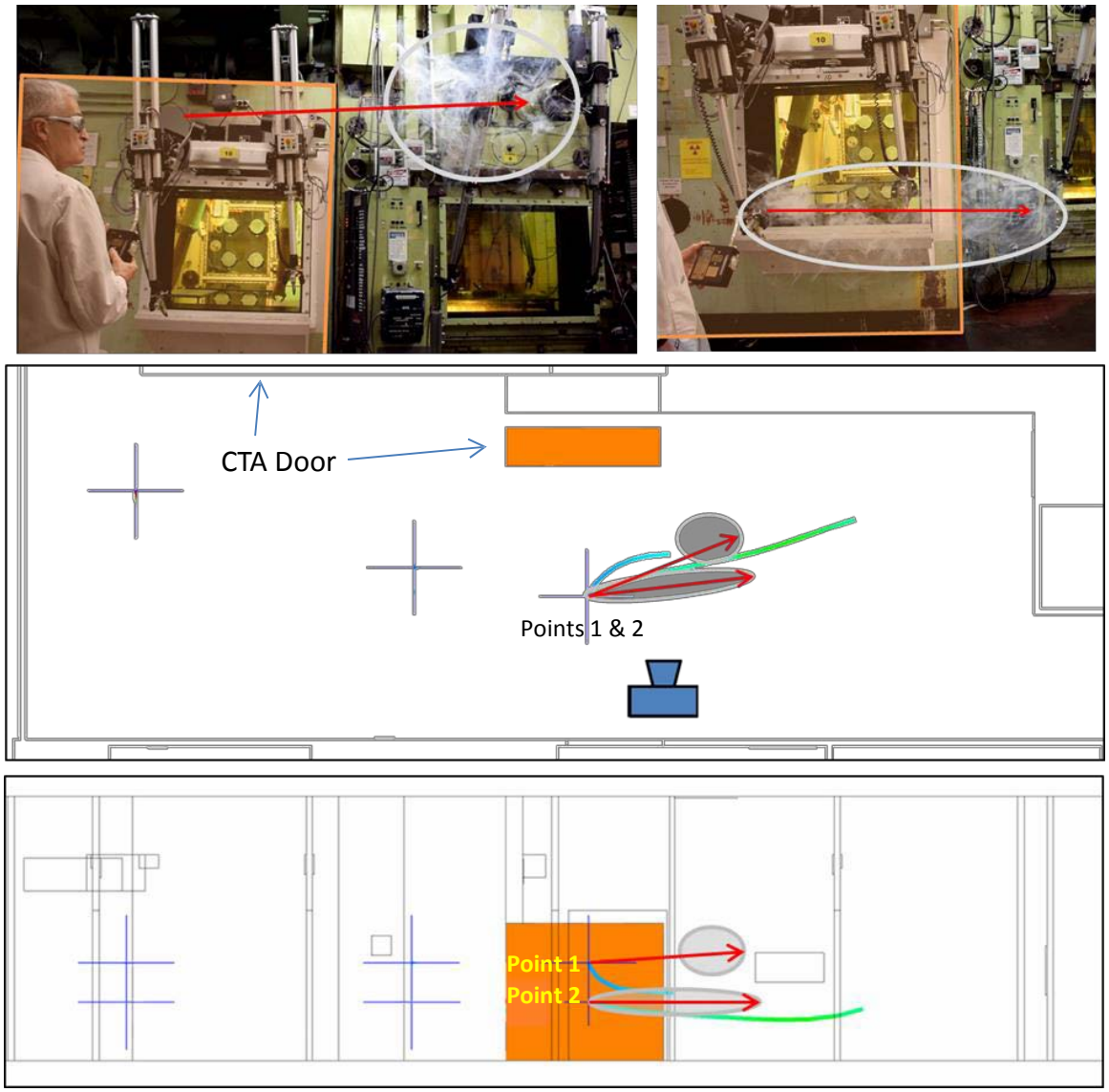

Figure 12. The first two images show the path of smoke released at a height of 5 ft and in front of the CTA door. The third image shows smoke released at a height of 3 ft, also in front of the CTA door. Unlike the CFD model, the smoke does not exhibit an initial downward velocity component.

The largest discrepancies between field test smoke releases and results predicted by CFD analysis occurred at the points 5 and 6 near the south wall. The observed flow patterns, shown in Figure 13, in the smoke released from 5 ft above the floor moved upward and towards the adjacent corridor. The release from 3 ft had an immediate vertical component to its horizontal motion that was similar to the model. If one were able to view the motion from above, it would nearly match the predicted path.

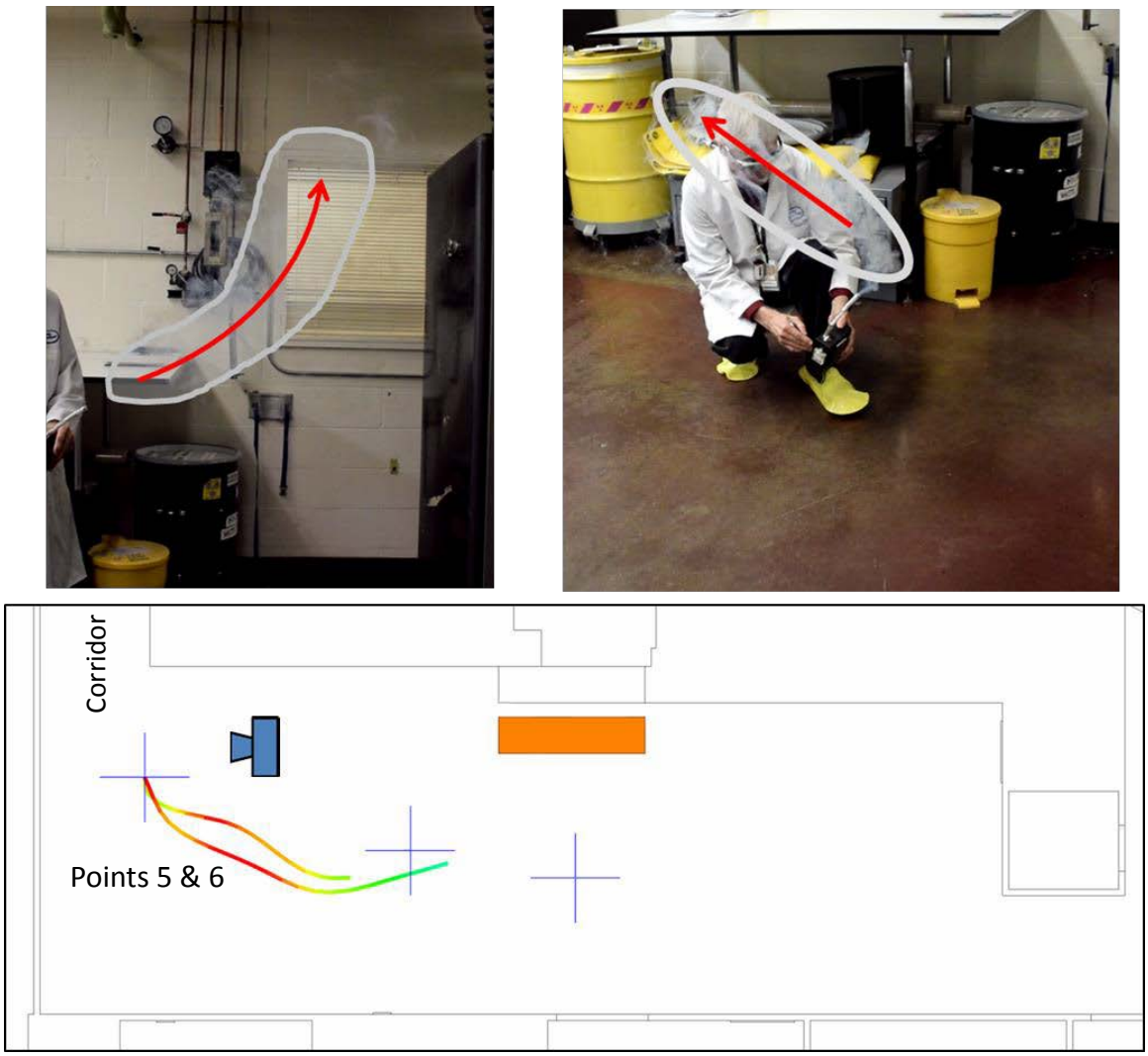

Figure 13. The most interesting smoke releases occurred at the point identified by the far left crosshair in Figure 10. The first release, at a height of approximately 3 ft, and a second release, made near the floor level, both exhibited flow patterns that are significantly different than the results of the initial CFD model.

3.3. Modification of the CFD model based on field test results

One obvious reason for the differences between the model and field test results is the number of geometric features that can interfere with the originally expected flow patterns. The first photo of Figure 14 shows a temporary shielding wall and also, in the second photo, two waste drums. There were also two cabinets near the waste drums, but they are not visible in the photo. Many other objects and features were present that could be candidates for modifying the expected flow behavior, but the temporary shielding wall was the first modification made to the CFD model geometry. A subsequent iteration included the drums and cabinets. Figure 15 shows 3-D representations of the original and updated geometry with the CTA door and new objects highlighted in orange.

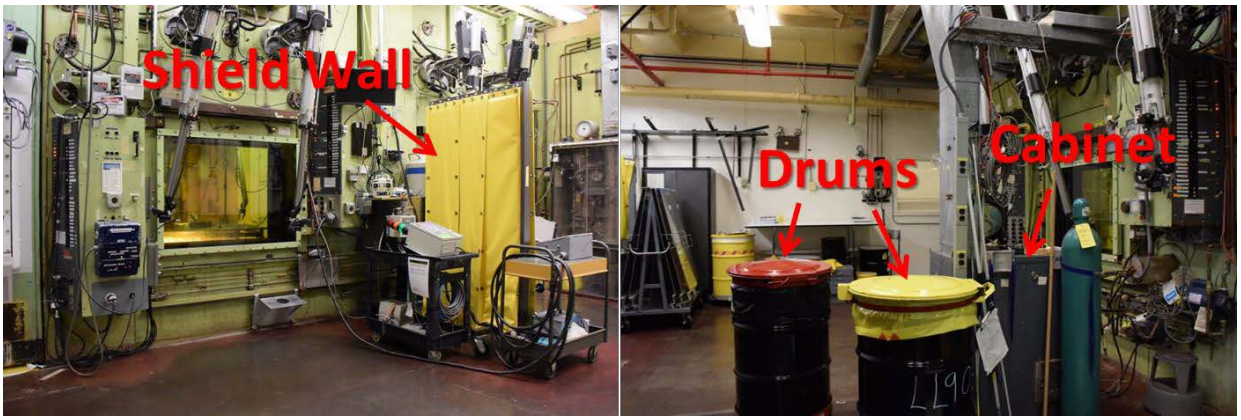

Figure 14. The first geometric feature added to the CFD model was a temporary shielding wall (shown on the left) positioned between the CTA door and one of the larger exhaust vents outside the CTA. Other features were added, including two waste drums and two cabinets (one cabinet is not visible in the photo).

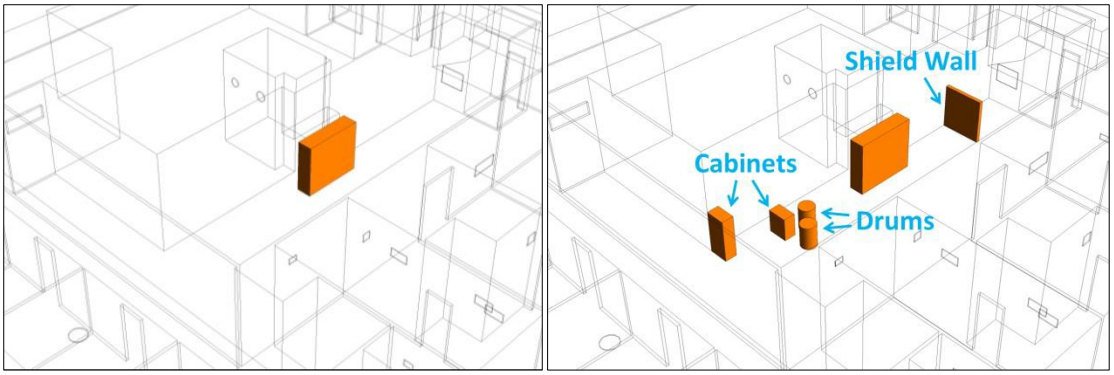

Figure 15. Additional geometric features were added to the model to reflect the conditions present at the time of the field test. The original model with the CTA door highlighted in orange is on the left, with the updated geometry displayed on right.

A noticeable change in the flow patterns is presented in Figure 16 by comparing velocity vector plots at elevations of 2, 4, and 6 feet above the floor. However, only the flow coming from the corridor, located at the upper left corner of the plots, seems to be affected by the addition of the new features. The most significant changes occur at the 2 ft elevation.

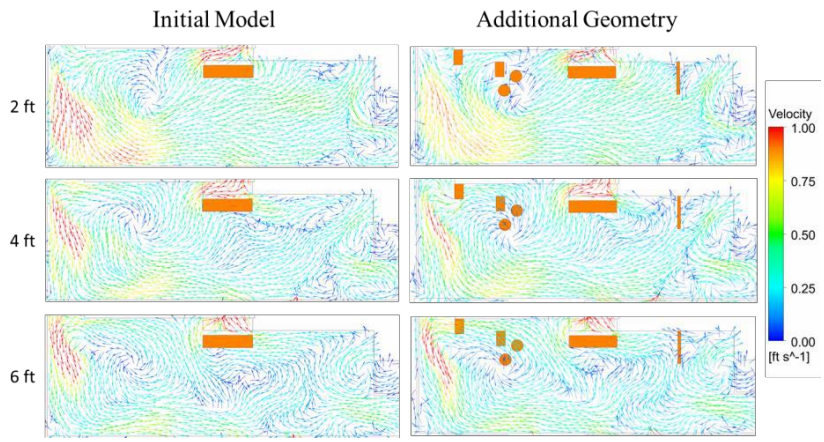

Figure 16. Including additional geometric features found in the AGHCF work area had a significant impact on the calculated flow patterns. However, we were not able to achieve good agreement between the model predictions and the field test smoke releases.

Figure 17 shows the plots of the simulated smoke releases of both the previous and modified geometries, which display very little change in the flow patterns. Many additional features found in the work area were also suspected of having an effect on the expected behavior of the air flow within the work area, but time constraints prevented further investigation of their significance.

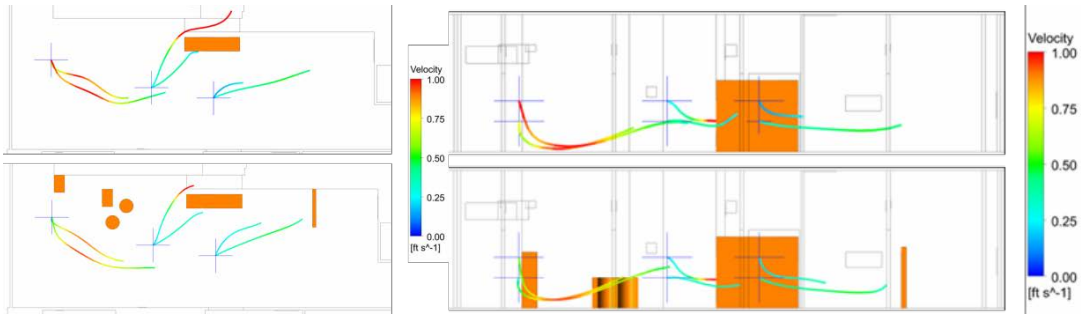

Figure 17. Plan views of the initial model (above) and the modified model (below). Although the additional geometric features showed a significant change apparent in the velocity vector plots, the simulated smoke releases are practically unchanged. As shown in the previous figure, the simulated smoke release results changed very little after modification of the model geometry. The large discrepancy with field test data in the vertical flow direction remains about the same.

The photos in Figure 18 provide just a few examples of other modifications that could be included in the geometry of the CFD model in locations outside the area that had been selected for field testing. During the field test, a technician was operating the manipulator arms at one workstation of the hot cell. There are several workstations which all have a set of controls for the remote manipulator arms, as can be seen in the right hand photo of Figure 18. Other objects with expected significance include piping shown near the ceiling with the photo of the technician and multiple tables along the perimeter of the work area. Examination of velocity vectors, shown in Figure 19, plotted in a vertical plane through the corridor where the technician was located shows a higher velocity flow along the ceiling, where the piping, manipulator controls, or even the presence of the technician could modify the flow into the work area.

Figure 18. There were many geometric features in the AGHCF that could affect flow patterns: manipulator controls at the work stations, other equipment, and piping and ductwork that were not present on the drawings used to create the initial geometry for the CFD model.

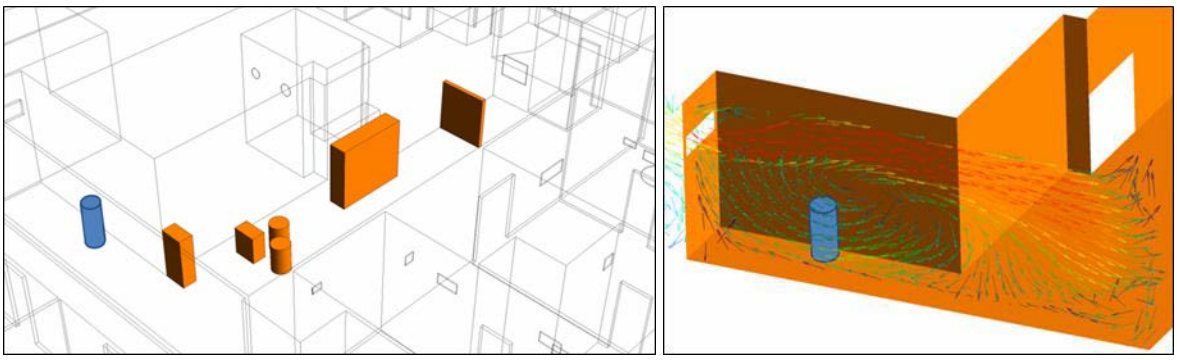

Figure 19. The image on the left displays the CTA door and geometry (highlighted in orange) that had been added to the CFD model following the field test. The blue cylinder represents the location of the worker visible in the left photograph of the previous figure.

4. Summary of findings & future work

4.1. Deviations from Original Phase II Plan

Based on the results and experience from Phase I, it became clear that standard smoke trace methods typically used in ventilation studies did not provide sufficiently accurate measurements that would be required for the analysis validation of critical ventilation systems such as found in nuclear facilities. From the outset of Phase II, it was determined that first, a more accurate means of field measurements needed to be developed. The PIV method that incorporates laser lighting and rapid imaging of the smoke traces was investigated. This method is currently being used at Argonne in a laboratory environment and thus provided an opportunity to test it in a field application.

As a result of the PIV testing, the resources for the particle testing in the hot cell area were more limited. Further, in cell particle flow tests proved impractical due to regulations for nuclear facilities. Also, in lab mock up tests for particle flows that simulate the actual flows in the hot cell was investigated and was determined to be a larger effort to perform than was possible within the remaining limited recourses available.

As a compromise, a methodology using in hot cell smoke tests and the identification of stagnations area were conducted in lieu of actual particle tests. Stagnation areas were considered to be places of contaminated particle accumulation. However, this was not considered conclusive validation and mock up testing is still considered viable alternative.

4.2. Benefits and limitations of in-field PIV with glovebox systems

Use of in-field PIV showed great potential for verifying CFD model results for flows in glovebox ventilation systems. Along with the benefits, some limitations were observed. One limitation encountered with our specific PIV test setup was the level of light produced by the 500mW laser. Even with the more powerful laser, the illuminated smoke was not bright enough for true PIV. Image analysis software for PIV is intended to track individual particles of smoke between photo frames in order to measure velocity. Even with the camera set at its widest aperture, the amount of light in our test would require long exposure times to capture images of the particles, resulting in images that would be too blurry to analyze. However, this might not be a significant problem for evaluating glovebox flows, as we are looking at a macroscopic level of air flow rather than a microscopic view used to verify a new turbulence model. As long as our in-field PIV setup can determine accurate flow direction vectors, this could be all that is needed to verify CFD results and determine possible limits for minimum or maximum values of average face velocity at an open gloveport. The only remaining issue then becomes introducing adequate smoke for photographing the flow without interfering with the normal flow conditions.

4.3. Modeling and verifying the air flow in a hot cell facility

As expected, attempting to model and accurately predict the flow patterns in the AGHCF is a challenging task. The complexity and scale of the hot cell ventilation system, when combined with the challenge of identifying specific regions with potential for accumulation of particulate matter, provides so many variables to include and adjust in the CFD model. Thermal conditions were checked. Some new geometric features were added to the model. Although the exhaust vent flow rates adjacent to the field tested area might seem to be an obvious boundary condition to test and verify, the entire ventilation system for the AGHCF is well documented with reliable data. There are several more significant geometric features which should be added to the CFD model. Several features noticed while photographing the test area, such as the pipes near the corridor ceiling, were not added due to time constraints. Additional field tests to collect more data and complete more iteration cycles would help to determine at what scale geometric features need to be included or can be safely considered negligible.

4.4. Closure

Typically, ventilation studies performed for non-hazardous facilities are common and do not require careful evaluation of local flow conditions at low flow velocities where contaminated particles may accumulate. However, for installations, such as nuclear facilities, the local flow conditions need to be well known and validated. These analysis requirements present considerable difficulties particularly in obtaining quality field measurement validation. Further, the accuracy and reliability required to validate these analyses were found to be beyond the ability of standard smoke trace methods such as now used by

Argonne's Industrial Hygiene for field measurements. The PIV methods that are currently used at Argonne in laboratory environments provide this accuracy and reliability, but are not yet readily extended to field use. However, preliminary field testing using the PIV methodology was performed in this study and the method, with further development, showed considerable promise. Now, the major challenge, which was not fully understood at the outset of this study, is to develop an accurate method of field measurements that are suitable for analysis validation for critical ventilation requirements such as found in Nuclear Facilities.

References

1. Marcel Loomans, The Measurement and Simulation of Indoor Air Flow, University of Eindhoven, 1998.
2. AGS Guideline for Gloveboxes, 3rd Ed. AGS-G001-2007.
3. AGS Standard of Practice for Leak Test Methodologies for Gloveboxes and Other Enclosures, 1st Ed. AGS-G004-2014.
4. AGS Standard of Practice for the Design and Fabrication of Nuclear-Application Gloveboxes, AGS-G006-2005.

APPENDIX

Summary of AGHCF field test data

CTA door closed

Location	Height above floor [ft]	Calculated Velocity [ft/s]	Anemometer Velocity [ft/s]
8 feet east of CTA door	3	0.36	0.49
	5	0.26	0.46
10 feet east of Window 7	3	0.36	0.43
	5	0.30	0.39
6 ft N of S wall, 4 ft E of corner	3	0.36	0.07
	5	0.43	0.20

CTA door open 5.5 ft

Location	Height above floor [ft]	Calculated Velocity [ft/s]	Anemometer Velocity [ft/s]
8 feet east of CTA door	3	0.46	0.66
	5	0.49	0.39
10 feet east of Window 7	3	0.33	0.39
	5	0.43	0.39
6 ft N of S wall, 4 ft E of corner	3	0.23	0.52
	5	0.36	0.46

CTA door open 33 inches

Location	Height above floor [ft]	Calculated Velocity [ft/s]	Anemometer Velocity [ft/s]
8 feet east of CTA door	3	0.30	0.36
	5	0.33	0.52
10 feet east of Window 7	3	0.33	0.33
	5	0.33	0.26
6 ft N of S wall, 4 ft E of corner	3	0.33	0.59
	5	0.33	0.59

Nuclear Engineering Division
Argonne National Laboratory
9700 South Cass Avenue, Bldg. 362
Argonne, IL 60439

www.anl.gov

Argonne National Laboratory is a U.S. Department of Energy
laboratory managed by UChicago Argonne, LLC