SANDIA REPORT
SAND2021-13511
Printed October 2021

Sandia National Laboratories

Studies of Alternative Ventilation Configurations to Mitigate Airborne Exposure Risks in Office Spaces

In alphabetical order:

Armenta, Casiano C.
Armijo, Roberto D.
Garcia, John M.
Ho, Clifford K.
Naber, Nicole L.

Prepared by
Sandia National Laboratories
Albuquerque, New Mexico 87185
and Livermore, California 94550

Issued by Sandia National Laboratories, operated for the United States Department of Energy by National Technology & Engineering Solutions of Sandia, LLC.

Printed in the United States of America. This report has been reproduced directly from the best available copy.

Available to DOE and DOE contractors from U.S.
Department of Energy
Office of Scientific and Technical Information
P.O. Box 62
Oak Ridge, TN 37831

Telephone: (865) 576-8401
Facsimile: (865) 576-5728
E-Mail: reports@osti.gov
Online ordering: http://www.osti.gov/scitech

Available to the public from U.S.
Department of Commerce
National Technical Information Service
5301 Shawnee Rd
Alexandria, VA 22312
Telephone: (800) 553-6847
Facsimile: (703) 605-6900
E-Mail: orders@ntis.gov
Online order: https://classic.ntis.gov/help/order-methods/

ABSTRACT

The objective of this study was to evaluate the impact of alternative ventilation configurations on airflow patterns and potential exposure risks in office spaces. Two existing conference rooms at Sandia NM were modeled using Computational Fluid Dynamics (CFD) simulations to characterize airflow patterns and potential airborne exposure risks in well-mixed and once-through (through-flow) ventilation conditions. Multiple scenarios were studied to evaluate the impact of occupancy, plexiglass barriers, and a modified-return airflow configuration. Experimental and visualization tests were also conducted to validate the well-mixed and through-flow models and findings.

The simulations demonstrated that the modified-return airflow configuration that promoted through-flow conditions reduced pathogen concentrations within the space compared to the well-mixed airflow configuration; occupancy reduction only reduced the number of exposed individuals, and plexiglass barriers had almost no effect. The experimentally measured air speeds at nine anemometer locations generally matched the simulated airflow velocities, and a fog-purge visualization test was also consistent with simulated results of plume movement and dissipation. The visualization tests demonstrated improvements in air change rate with the modified return, which promoted through-flow conditions, versus the original well-mixed ventilation configuration.

The results of this study demonstrate that minor modifications to a space that promote through-flow conditions can improve air quality and reduce pathogen concentrations. Additional airflow modeling and testing of alternative occupied space configurations are recommended to further inform room designs that mitigate airborne exposure risks for occupants.

ACKNOWLEDGEMENTS

The authors thank Mark Coffing for initiating and supporting this work. The authors also thank Robin Jones and Wes Mozley for their reviews of this report.

CONTENTS

LIST OF FIGURES

EXECUTIVE SUMMARY

In traditional heating, ventilation, and air-conditioning (HVAC) configurations, well-mixed airflow is important to enhancing thermal comfort in rooms and office spaces. However, as recent studies have shown during the COVID-19 pandemic, poorly ventilated or well-mixed room conditions can lead to increased exposure risks and transmission of airborne pathogens. This study evaluated existing and modified HVAC configurations of Sandia conference rooms to determine if simple modifications can reduce exposure and transmission risks. The objective was to identify simple, cost-effective, and efficient HVAC modifications that can be safely implemented as Sandians return to on-site work.

CFD models of airflow patterns and expelled pathogen/vapor concentrations in 802/2000 and 886/114 showed that simple reconfigurations to the supply and return vents that promoted a once-through (or through-flow) airflow pattern significantly reduced simulated pathogen concentrations by reducing the amount of dispersion and mixing throughout the room. Validation tests were performed in 886/114 to compare measured and simulated air velocities for different ventilation configurations using nine anemometers placed throughout the room. Fog-purge tests were performed to evaluate the effectiveness of air-change rate and flow patterns on purging a room filled with fog using original and modified ventilation configurations. Additional tests using smoke emitters and tissue tests further confirmed simulated airflow patterns.

Additional details of the simulations and tests are summarized below, along with key takeaways:

Modeling of 802/2000 Conference Room

- Plexiglass barriers and increased occupancy did not significantly change the room concentrations
    - Increased occupancy may increase the likelihood of infected individuals and/or receptor exposures, but the simulated concentrations throughout the room were not impacted significantly by increased occupancy
- Single central return vent (fume hood) decreased room concentrations by up to 2 – 3 orders of magnitude
    - Once-through flow condition reduced mixing, pathogen concentrations, and exposure

Anemometer Testing in 886/114 Conference Room

- Average measured vs. simulated air velocities at nine anemometer locations were within ~6% for original vents and 0.2% for modified vents
- Measured and simulated air speeds were between ~1 – 30 ft/min at all anemometer locations
- Results and comparisons to model results provide confidence in the CFD models and simulation methods

Fog Purge Tests in 886/14 Conference Room

- Simulated time to purge fog to ~10% of peak concentrations was ~10 – 15 min, consistent with subjective observations from five individuals
- Both visualization tests and simulations showed that modified vents purged fog faster by 1 – 2 minutes (~12 percent)
- Simulations showed that modified vents yielded a more spatially uniform purge throughout the room relative to original vents

Key Takeaways
Room ventilation configuration can be modified to increase purge rate and minimize airborne exposures by promoting a through-flow condition:

- Air flow from diffusers should be maximized and oriented toward periphery of room, if possible
- Return vents should be located in the center of room to minimize mixing and promote through-flow ventilation
- Seating should be arranged to prevent occupants from sitting in between the return vent(s) and any upstream occupants
- Area of centralized return should be sized to maximize air velocities while yielding acceptable noise levels (and thermal comfort)

ACRONYMS AND DEFINITIONS

Abbreviation	Definition
AHU	Air Handling Unit
ASHRAE	American Society of Heating Refrigeration and Air Conditioning Engineers
BTUH	British Thermal Units Per Hour
CAD	Computer Aided Design
CDC	Centers for Disease Control
CFD	Computational Fluid Dynamics
CFM	Cubic Feet Per Minute
COVID-19	Coronavirus Disease of 2019
DB	Dry Bulb Temperature
FANS	Favre-Averaged Navier-Stokes
GPM	Gallons Per Minute
HVAC	Heating Ventilating and Air Conditioning
LES	Large Eddy Simulation
MERV	Minimum Efficiency Reporting Value
RANS	Reynolds Average Navier Stokes
VAV	Variable Air Volume Box
WB	Wet Bulb Temperature

Page Left Blank

1. INTRODUCTION

Transmission of the COVID-19 virus has been thoroughly researched by scientists around the world, and specifics recognized by the Centers for Disease Control and Prevention (CDC) state that fine droplets of the virus can remain suspended within the air for minutes or hours [1, 2]. The air quality provided by HVAC systems has been a major concern throughout the pandemic and has been often studied Efforts have been made by the industry to analyze and study air quality improvements in public spaces to reduce the spread of the virus using HVAC technology.

Some of the more popular technologies studied have been the cleanliness and Minimum Efficiency Reporting Value (MERV) rating of HVAC filters, increasing the amount of outside air introduced into a building to improve ventilation, and running the building air handling system for longer periods of time to flush out potentially contaminated air from the building. ASHRAE (American Society for Heating, Refrigerating and Air Conditioning Engineers) has provided recommendations related to the pandemic for these improvements. A combination of maximum outside air ventilation, filtration of recirculated air using MERV 13 filters or higher, and the flushing of building air to achieve three air changes of outdoor air per hour is considered an effective mitigation tactic [3]. These options improve air quality within a building, but with a price of increased material costs and greater energy consumption.

This study investigates low-cost and easily implementable modifications to the ventilation system in offices and/or conference rooms that can minimize internal air recirculation and exposure risks. At Sandia, many buildings employ an open plenum return above the ceilings to convey air from the rooms to the central air handling system. These open plenums above the ceilings allow for convenient reconfiguration of the return vents in any desired location in the ceiling. By modifying the direction of the supply vents and location and number of return vents, through-flow conditions can be induced that may minimize airborne exposures. "General airflow direction should be from cleaner air to less clean air, and processes and workers should be kept on the clean side of the general airflow pattern" [4].

Many businesses, companies, and local and federal government agencies are already running under a tight maintenance budget. A goal of this study was to find cost effective ways to improve the air quality without suffering the long-term increased costs. These improvements would not only protect the health of the community in the current environment, but also reduce the transmission of viruses year-round, including the common cold and flu. Two existing Sandia conference rooms were evaluated using simulated modeling methods to better understand the impact of potential modifications on existing spaces.

CFD modeling allows for the evaluation of these changes in a simulated environment. Modeling of two existing conference rooms (802/2000 and 886/114) were performed to evaluate the impact of alternative ventilation configurations on airflow patterns and potential exposure risks. The first scenario in each simulation utilized the existing ceiling supply and return air distribution as it is currently installed and operating in the space. The second scenario modified the supply and/or return air distribution to alter the airflow patterns and promote a once-through sweeping motion of the airflow and improve ventilation effectiveness within the conference room. Simulation results indicated that inducing a through-flow ventilation configuration would reduce pathogen concentrations. As a result, experimental tests were performed in 886/114 to validate the findings.

During testing in 886/114, the existing return air grill locations were covered and a new return air opening was created approximately in the center of the conference room to induce this condition. The model results of airflow velocities and plume dissipation were compared to the measured airflow velocities and observed dissipation during fog-purge visualization tests.

Section 2 describes the CFD modeling approach that was used in this study. Simulation results that informed the testing and preferred room configuration are summarized. Section 3 describes the experimental testing that was performed to gain confidence in the models and evaluate the proposed improvements to the room configurations. Section 4 summarizes the results of the testing and modeling, and Section 5 provides conclusions and recommendations.

2. MODELING APPROACH

2.1. CFD Modeling

The general modeling approach in this study is to use CFD models to simulate expelled aerosol plume dispersion and perform comparative studies of exposure risks of expiratory events under various scenarios. Spatial and temporal simulations of the relative concentrations of the expelled pathogen (assumed to be uniformly distributed in the vapor plume) are compared and used to determine risks of exposure and probability of infection. High-fidelity turbulence models are available to simulate time-varying turbulent processes initiated by violent expiratory events (e.g., Large Eddy Simulation (LES) [5]). However, in these studies, a time-averaged turbulence model (e.g., Reynolds-Averaged or Favre-Averaged Navier Stokes equations using a k-□ turbulence model) was implemented to reduce the computational expense and evaluate a large number of scenarios.

Relative trends in time-integrated concentrations and exposure risks as a function of time and location are assumed to be adequately captured by the time-averaged turbulence models; the objective is to perform comparative risk analyses of different configurations and scenarios rather than to make absolute predictions. In addition, we assume that the relative distribution and concentration of pathogens (droplet nuclei) can be represented by the transient dispersion of the expelled vapor plume. We neglect potential transmission from large droplets, which we assume will fall out due to gravity. Thus, the analyses in this work focuses on small droplets (aerosols) that remain aloft in the air for long periods and have been identified as a significant contributor to airborne transmission [6].

Several modeling and experimental studies have shown that these small particles (a few microns or smaller) follow the bulk airflow and can be accurately represented by a tracer gas [7-10]. In the experimental study of Bivolarova et al. [7], particles of three sizes (0.07, 0.7, and 3.5 microns) and nitrous oxide tracer gas were generated in a room simultaneously at the same location with various ventilation rates and configurations. Sampling at different locations within the room showed that "tracer gas can be used to evaluate the distribution of aerosol particles in ventilated rooms." Gupta et al. [8] and Zhang et al. [9] also concluded that small particles behaved like a tracer gas and followed the bulk airflow during testing and modeling of particle transport in an airplane cabin, and Gupta et al. [8] simulated various expiratory events including coughing, talking, and breathing.

Solidworks Flow Simulation is a commercial software package [11] that was used to perform the CFD simulations in this study. Flow Simulation solves the conservation of mass, momentum, energy, and species equations using a discrete numerical finite-volume approach. For turbulent flows, Flow Simulation solves the Favre-Averaged Navier-Stokes (FANS) equations. FANS uses a mass-weighted time-averaging scheme, which can avoid complications associated with the Reynolds Averaged Navier Stokes (RANS) solutions for compressible flows (for incompressible low-Mach flow conditions such as those in the current study, FANS and RANS solutions are similar). Previous studies have demonstrated the use of FANS turbulence models for incompressible flows at various Mach numbers [12-14]. A k-ε turbulence model is employed using a laminar/turbulent nearwall model with modified wall functions [11]. Meshing is performed using a combination of hexahedral and polyhedral elements, which accommodate curved boundaries between phases or materials. Spatial derivatives are approximated with implicit difference operators of second-order accuracy, and time derivatives are approximated with an implicit Euler scheme of first-order accuracy. The time-step size at each iteration is determined using the Courant–Friedrichs–Lewy convergence criterion, where the smallest cell size and a characteristic velocity of the flow field are used. Additional details of the numerical formulations, conservation equations, constitutive relations, meshing, and solution techniques can be found in the technical reference manual [11]. Flow Simulation is integrated within the 3D CAD package Solidworks, which makes geometry and mesh creation seamless and efficient for various scenarios and configurations.

2.2. Modeling of 802/2000 Conference Room

Conference room 802/2000 was the first space that was modeled to investigate the impacts of alternative ventilation configurations on airflow patterns and potential exposure risks. The results of these simulations informed the decision to test and further evaluate alternative through-flow ventilation designs in 886/114 (see Sections 2.3 and 3).

2.2.1. Description and Objectives

Figure 1 shows a photograph and model of conference room 802/2000. It consists of one large conference table in the middle of the room with smaller arc tables on either side. The model captures most of the salient features in the room, but it omits the monitors, easels, computer peripherals, and smaller items that are either mobile or not expected to impact the bulk airflow in the room.

Figure 2 shows the different room configurations that were simulated:

- **Baseline: Minimal occupancy**: Seven occupants socially distanced
- **Minimal occupancy with plexiglass barriers**: Seven occupants with plexiglass barriers in front of each occupant
- **Full occupancy**: 21 occupants; each seat occupied
- **Minimal occupancy with modified central return**: Seven occupants with all returns replaced with a single centralized return

In each case, an infected individual was assumed to be exhaling pathogens into the room. The simulated transient concentrations were recorded and compared to determine the impacts of the different room configurations.

Figure 1. Photograph (left) and 3-D model (right) of conference room 802/2000.

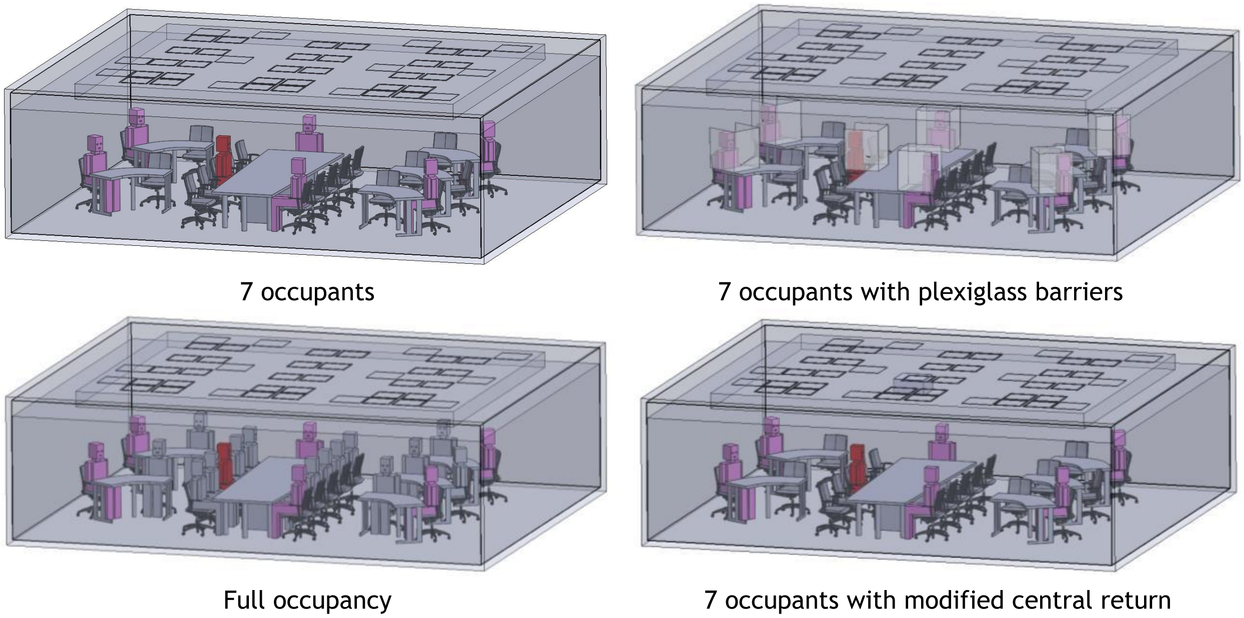

Figure 2. Configurations modeled for conference room 802/2000.

2.2.2. Boundary Conditions

The 802/2000 conference room supply air is distributed throughout the space by twelve perforated supply diffusers with recessed directional louvers. Thirty volumetric troffer lights, a type of recessed lighting that has return or supply air ventilation along the periphery of the light, provide the return air openings (see Figure 3). This configuration with multiple distributed supply and return vents promotes a well-mixed airflow configuration.

Figure 3 shows that each of the 12 diffusers provided supply air at 100 cfm (1200 cfm total) at at 90 °F (winter heating). The return vents were prescribed with an ambient (environmental) pressure to allow for mass balance between the supply and return flow of air. The walls were maintained at a constant 65 °F.

Figure 4 shows the boundary conditions that were assumed for the room occupants. Each simulated body was maintained at 98.6 °F, and the seven "receptors" (colored in magenta) were prescribed with a continuous inhalation rate of 8 L/min (average tidal breathing rate). The infected individual (highlighted in red) was prescribed with a continuous exhalation of aerosolized pathogens, which were represented as exhaled water vapor at 98.6 °F at a rate of 8 L/min.

The steady-state airflow was first simulated, followed by the transient pathogen transport. During the steady-state airflow simulations, exhaled air from the infected individual was still prescribed at 8 L/min, but it was not tracked as a separate "pathogen" constituent. Once the steady-state airflow simulations converged, the flow field was "frozen," and the exhalation of pathogen-laden water vapor was simulated for six minutes, which was sufficient to observe salient trends in the simulated concentrations inhaled by the receptors. The simulated concentrations near the mouths of the seven receptors were used as a proxy for exposure risk to compare the different cases.

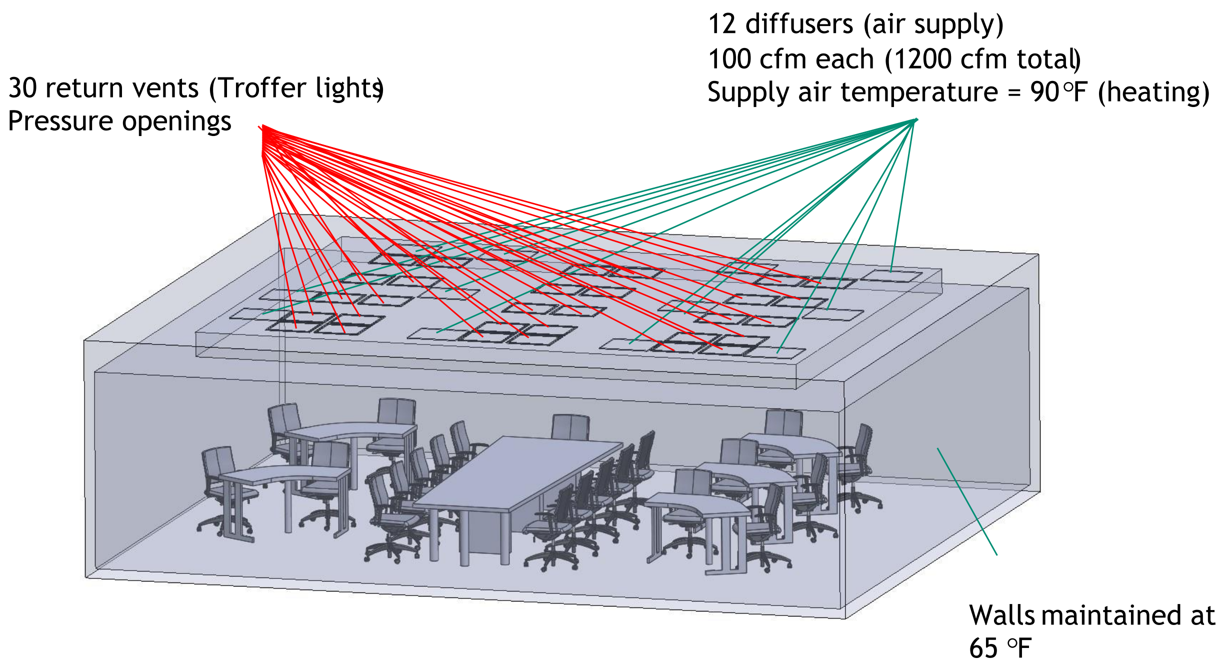

Figure 3. Boundary conditions for ventilation and walls for room 802/2000 simulations.

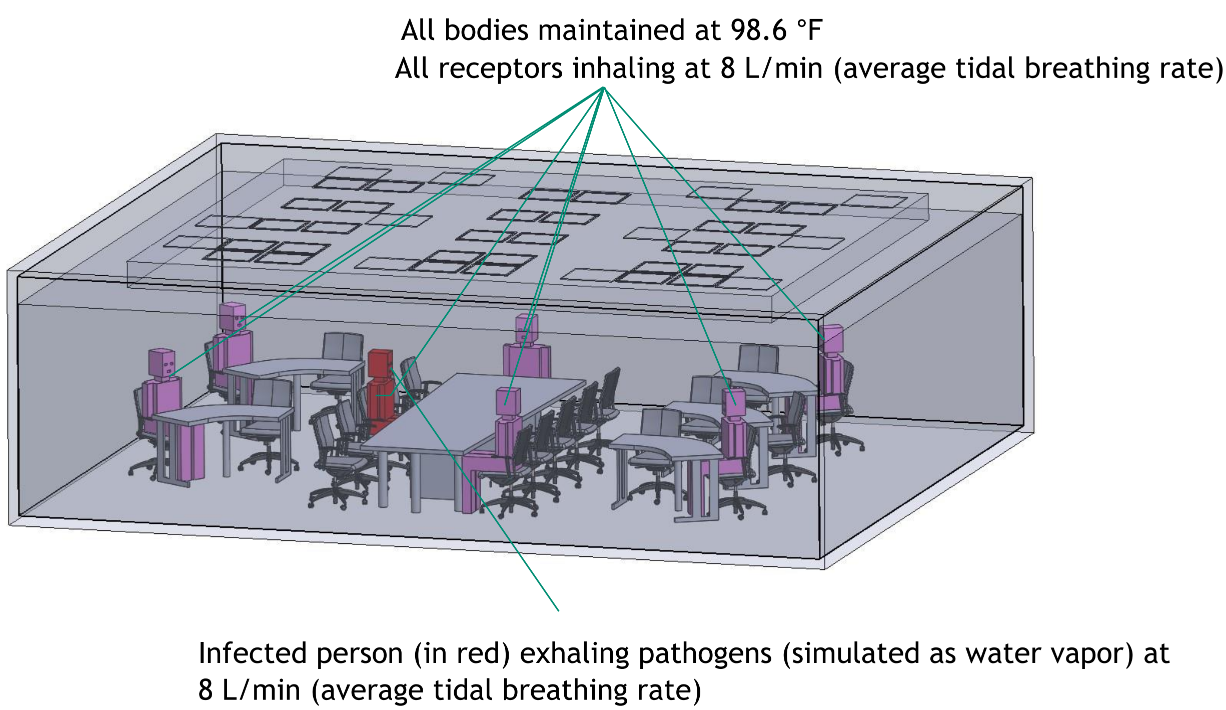

Figure 4. Boundary conditions for bodies and pathogen for room 802/2000 simulations.

2.2.3. 802/2000 Modeling Results

Figure 5 shows the simulated steady-state temperatures in 802/2000 resulting from the prescribed boundary conditions described in the previous section. The bulk average temperature in the room is ~77 °F, and there is slight stratification with warmer temperatures near the ceiling and cooler temperatures near the floor.

Figure 6 shows representative simulated steady-state flow field in 802/2000 with seven occupants. The airflow is well mixed with numerous recirculation patterns at multiple scales. The majority of the air velocities are less than ~0.1 m/s. Although the specific airflow patterns will change depending on the simulated scenario shown in Figure 2, all cases with the original supply and return vents yield well-mixed conditions.

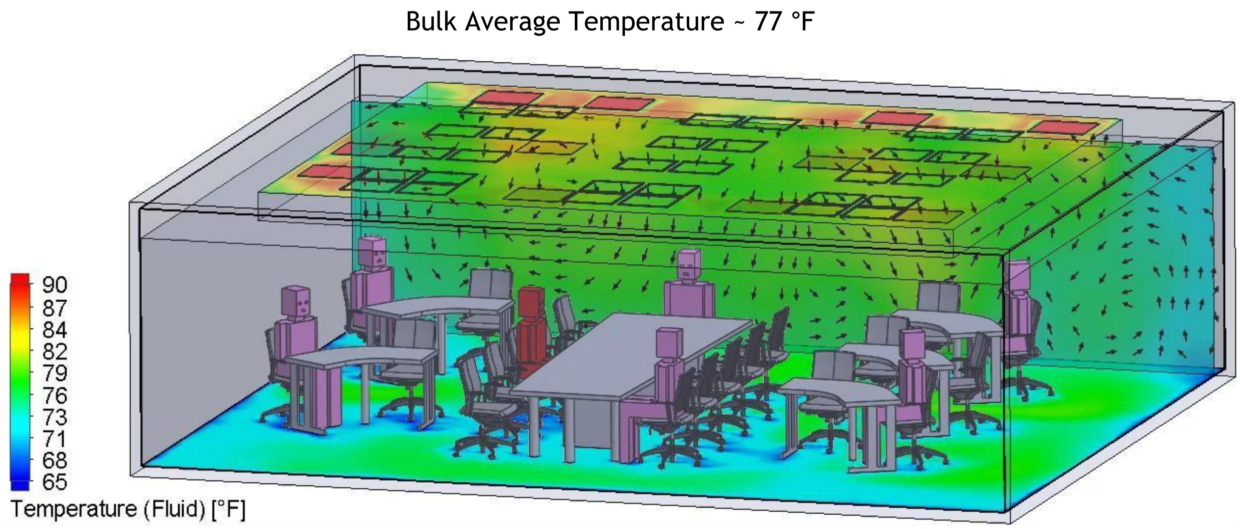

Figure 5. Simulated steady-state temperatures in room 802/2000.

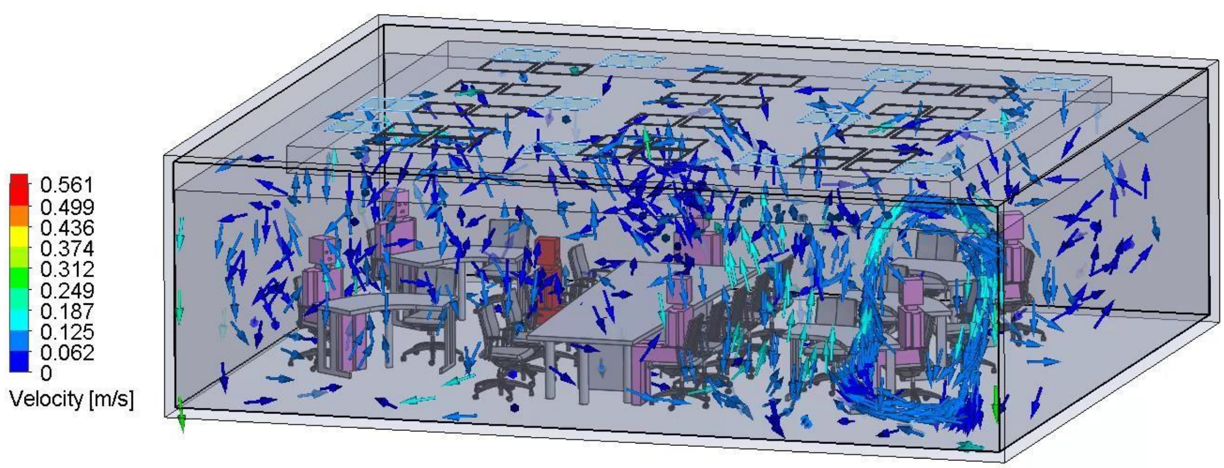

Figure 6. Simulated steady-state flow field in room 802/2000.

After steady-state flow conditions were simulated, exhalation of pathogens from the infected person was simulated. Figure 7 shows an example of the simulated pathogen distribution after several minutes in the minimal-occupant scenario with the original ventilation configuration. Figure 8 shows the simulated transient pathogen concentrations (mass fraction) at each of the seven receptor locations for six minutes. The horizontal line represents the source concentration (mass fraction of water vapor) emitted from the infected individual. The objective was to identify scenarios in which the simulated pathogen concentrations were significantly lower than the baseline configuration. Results show that neither plexiglass barriers nor the increased occupancy had a significant impact on the overall pathogen concentrations in the room relative to the baseline scenario. In each of these scenarios, the pathogen concentrations began to asymptote at ~1 – 10 ppm. The full-occupancy scenario had a greater variability in the simulated concentrations. Only the modified central return scenario showed a notable decrease in the simulated concentrations throughout the room, by up to 2 – 3 orders of magnitude. Figure 9 shows the simulated velocity vectors for the modified central return scenario. Rather than a well-mixed condition, air entering the room from the supply vents moves in a more direct fashion to the central return. This through-flow conditions minimizes the dispersion of exhaled pathogens throughout the room. Therefore, placement of supply and return vents that promote through-flow conditions is recommended. In addition, the seating should be arranged to prevent occupants from sitting in between the return vent(s) and upstream occupants.

Based on these results, additional modeling and testing were performed in 886/114 to validate these findings (see Sections 2.3 and 3). Air speeds were measured in original and modified ventilation scenarios, and fog-purge tests were performed to evaluate the air exchange rate and effectiveness of dissipating pathogens from a room for different ventilation configurations.

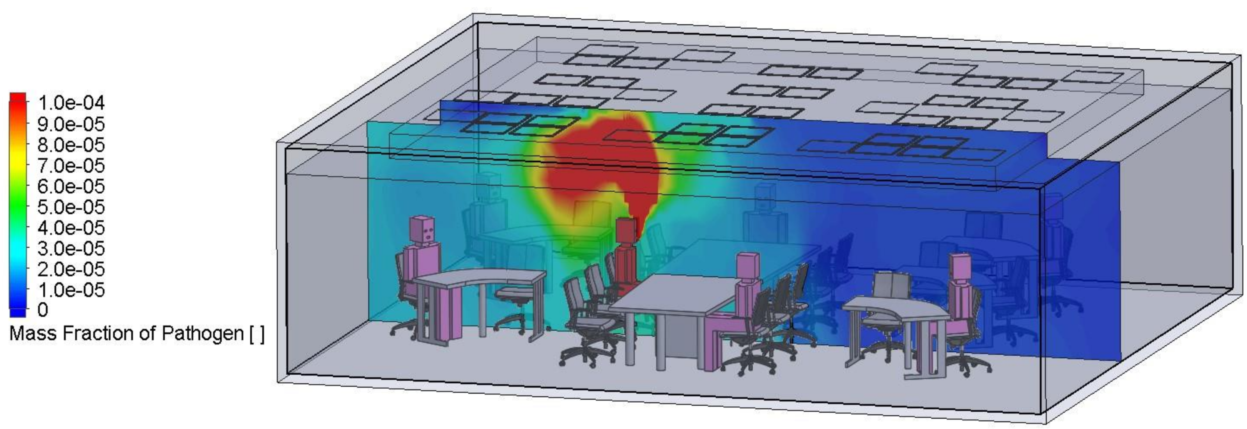

Figure 7. Simulated transient pathogen concentrations.

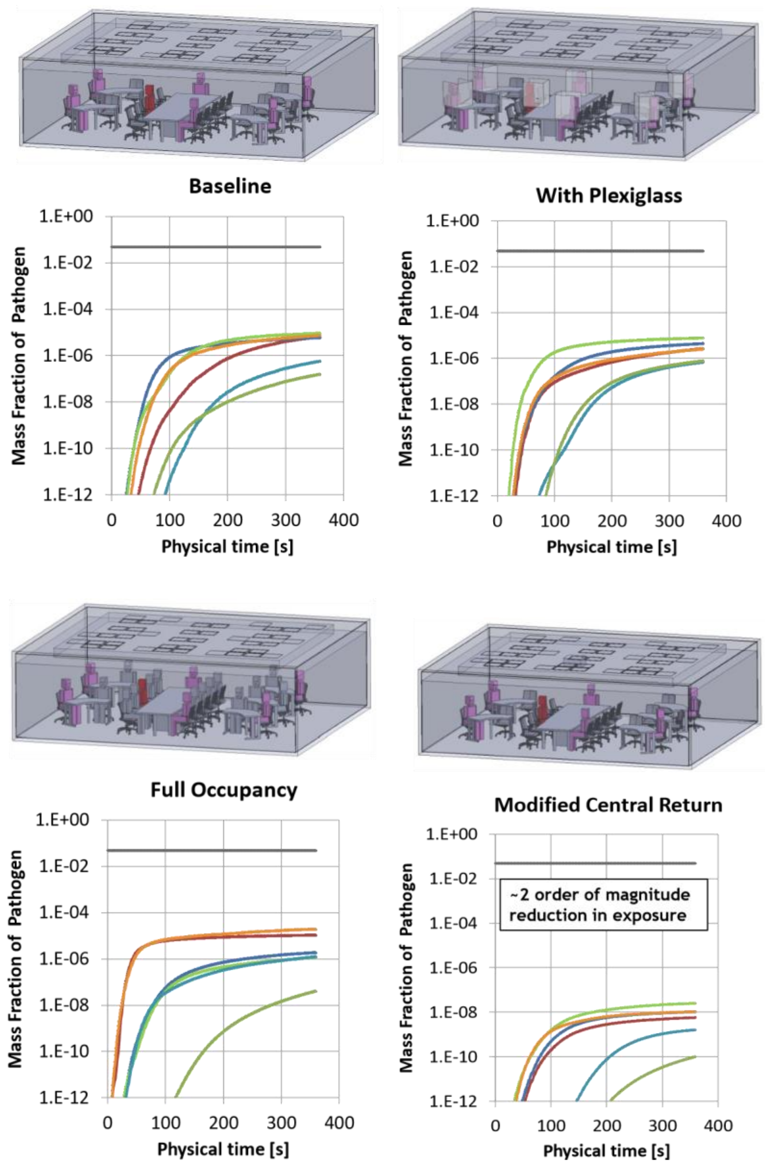

Figure 8. Simulated pathogen concentrations for different room scenarios in 802/2000.

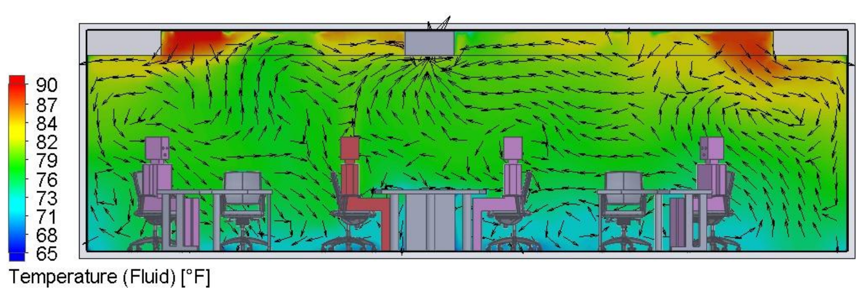

Figure 9. Simulated temperatures and velocity vectors for modified central return scenario in 802/2000.

2.3. Modeling of 886/114 Conference Room

Additional modeling of conference room 886/114 was performed because that room was under the control of Sandia Facilities, and validation tests could be performed. Air speeds were measured using anemometer probes, and fog-purge visualization tests were performed for different ventilation configurations.

2.3.1. Description and Objectives

Photographs of the 886/114 conference room are shown in Figure 10, and Figure 11 shows the corresponding 3-D model created in Solidworks. Two different ventilation configurations were simulated and eventually tested:

- Original
    - Four adjustable modular core directional supply air diffusers
    - Two perforated return air vents
- Modified
    - All four diffusers modified to blow air toward perimeter of room
    - Two original return vents covered
    - Single central return vent opened in middle of ceiling

The measured and simulated air velocities and fog purge rates were compared to gain confidence in models.

Figure 10. Photographs of 886/114

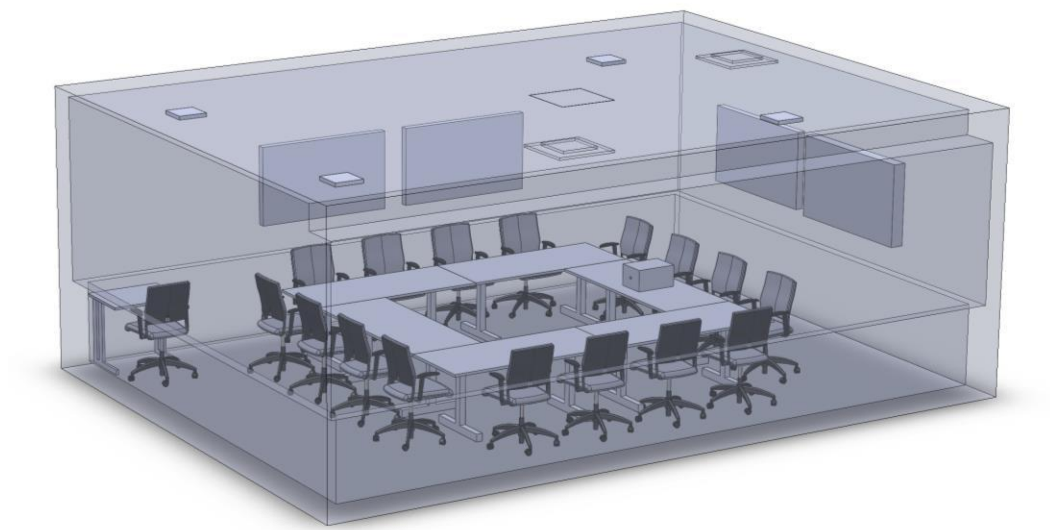

Figure 11. Model of conference room 886/114.

2.3.2. Boundary Conditions

Figure 12 shows the supply and return vents in the original and modified configurations, along with measured air velocities that were used together with the total flow rates to specify boundary conditions for the simulations. In the modified configuration, portions of the supply vents were covered to direct the airflow toward the periphery of the room. In addition, the original return vents were covered, and the central ceiling tile was removed, allowing air to return to the central air handling unit in the ductless plenum above the ceiling.

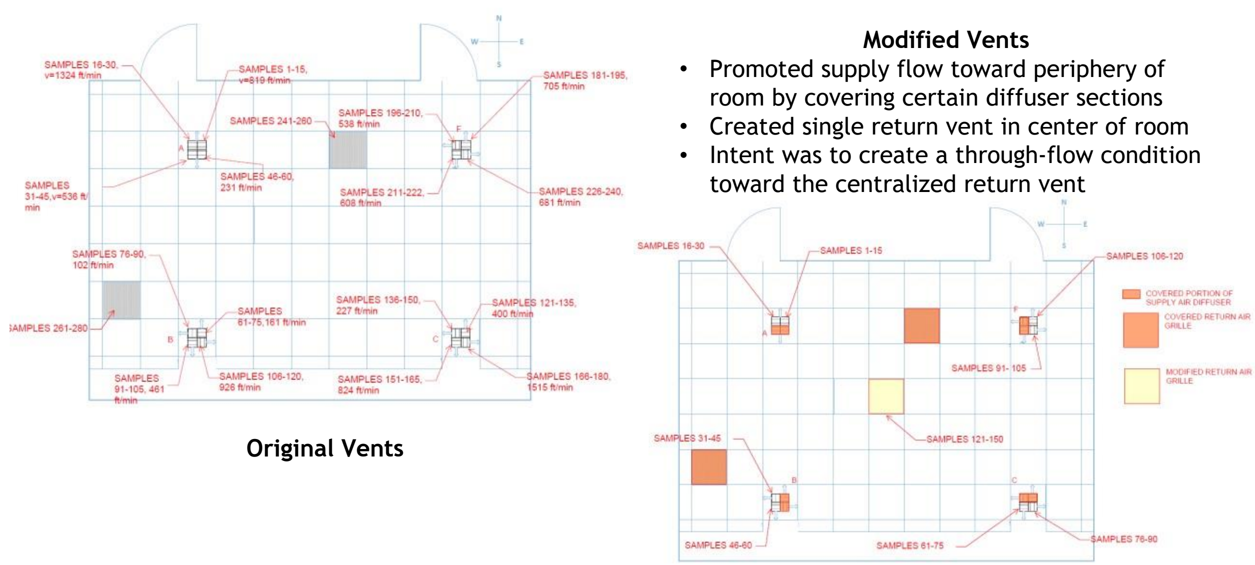

Figure 12. Original (left) and modified vent (right) configurations for 886/114.

3. TEST AND VALIDATION APPROACH

3.1. Conference Room for Validation Testing (886/114)

The building used for validation testing is a standard office design. It is occupied with Facilities employees and has general open cubical office space, closed offices, a director's suite, common break area and two conference rooms. The HVAC system consists of a built-up air handling unit (AHU) complete with fan section, cooling coil section, heating coil section, and mixing box section with all the accessories required for a complete and operational unit. The fan assembly is configured for variable flow service and a listed capacity of 18,610 CFM at 5,500' elevation against 4" w.c. total static pressure. The cooling coil capacity is listed to be capable of cooling 18,282 CFM of air at 5,500' elevation from 84.0 °F DB, 63.0 °FWB to 56.0 °FDB, 53.6 °FWB with 94.1 GPM of 48 °F water at a 10°F rise. Heating coil capacity is listed to be capable of heating 18,282 CFM of air at 5,500' elevation from 50 °F to 65 °F with 24.2 GPM of water entering at 180 °F. The mixing box is configured with full air flow capacity damper sections to be able to utilize full outside air for free cooling during colder ambient conditions. The air is distributed through a single duct system to the terminal units throughout the building. The AHU is operated based on an occupancy schedule and is set to operate M-F 6:00AM to 6:00PM. The variable volume fan is adjusted according to the sensed internal loads of the building. The temperature delivered from the AHU is adjusted according to the sensed internal loads of the building as well.

This conference room is served from the main air handling unit AH01 in the mechanical room through two different variable air volume (VAV) terminal units. These VAV units have re-heat coils to provide additional heat to the space. The local controls are field level direct digital controls, capable of adjusting the air flows from a minimum setting to a maximum setting. VAV-36 has a listed capacity of 660 CFM max. and 200 CFM min. air flow and 0.5 GPM heating water flow rate VAV-23 has a listed capacity of 600 CFM max. and 180 CFM min. air flow rate and 0.5 GPM heating water flow rate. During this study, a work request was placed to re-configure the branch duct work in the conference room with VAV-36 terminal unit. There is a supplemental fan coil cooling unit that also serves the conference room and has a listed capacity of 400 CFM at 0.5" external static pressure. The cooling coil has a listed capacity of 6,760 BTUH. During the testing phase of this project, the fan coil unit was off.

3.2. Test Equipment

3.2.1. Instrumentation

As the scope of work materialized for testing and validation, it was decided that ten hot wire anemometers would be required and utilized for obtaining temperature and airflow throughout the conference room. Nine individual hot wire anemometers were ordered for placement throughout the conference room and one additional anemometer for conducting spot checks.

The selected tool was a hand-held VELOCICALC® Multi-Function Ventilation Meter Model 9565 by TSI with a telescoping 964 Thermoanemometer Straight Probe to measure velocity, temperature, and relative humidity. It was attractive due to the ease of use, multiple capabilities, and logging setup via a computer-software interface. The resolution of the airflow velocity measurement was an important factor in the selection since the velocities were unknown and the device provided the capability to measured velocities down to 1 ft/s. Figure 13 depicts the tools.

Figure 13. Meter and thermoanemometer tools used for testing

The strategic placement of each anemometer as indicated in Figure 14 below was utilized to validate the CFD model. The anemometer locations were intended to simulate room occupants by attaching them to the chairs spaced throughout the room. Measurements of test probe locations were made for use in the CFD simulations.

Figure 14. Taking measurements of probe locations before testing

3.2.2. Smoke Generation and Other Tools

In addition to using the anemometers, the team decided that visually seeing the airflow patterns would enhance the study effort. Out of several options discussed, smoke emitters, a fog machine, and a pom-pom-like tool were employed to provide a better visualization of the airflow direction in the room and at the supply and return locations.

Short and long duration smoke emitters, produced clean, non-toxic, oil free smoke that is generated by a chemical reaction. These are normally used to find airflow leaks in systems or validate fume hood operation in lab spaces. The 90-second short duration emitters provided 600 cubic-feet of smoke over the life of the emitter. The 4-minute long duration emitters provided 2,500 cubic-feet of smoke, which was enough to fill the room. Glass jars were used to contain the smoke emitters and prevent damage to the table. These items including the lighters can be seen in Figure 15.

Figure 15. Smoke emitters before testing

The electronic fog machine "Fog Fury 2000", Figure 16, used a mixture of propylene glycol and water applied to an 1100-Watt heating element to produce the smoke. The equipment was controlled remotely and could be operated via cable from outside the room. It produced fog output of 7000 cubic-feet of fog per minute and was used to produce between five and thirty seconds of fog during the tests.

Figure 16. Fog machine used for smoke purge testing

The tissue-tester tool, or pom-pom, was created by tearing small strips of tissue paper and taping them to the tip of a long pole. This simple tool was used to provide quick analysis of airflow directions at the supply and return air registers.

3.3. Ventilation and Visualization Tests

3.3.1. Preparation

Testing was completed on both the mixed airflow and modified airflow scenarios. Tests were completed with the mixed airflow configuration first, performing the anemometer tests to gather airflow data, tissue tests, smoke emitter testing, and fog purge. The modified airflow configuration was tested second using the same strategies. All smoke tests were completed within a day. Anemometer testing was completed in multiple iterations over several days.

To achieve the modified airflow configuration without actual site modifications, the supply air diffusers and return air grilles were blocked off using card stock, paper, and magnets to adjust the direction of airflow. These methods can be seen in Figure 17 below.

Figure 17. Modified Airflow Configuration

In preparation for the smoke tests the environmental health and safety (ES&H) team was contacted for best practices regarding the testing. A routine maintenance checklist was completed to evaluate the hazards. Proper personal protection equipment (PPE) (gloves and goggles) were used during the smoke tests. Masks were worn as a requirement by Sandia for protection of the workforce during the pandemic. The facilities maintenance team was contacted to provide a fire alarm outage to prevent any potential emergency alarms going off because of the testing. The fire alarm system was disabled, and a fire watch scenario was enacted for the duration of the tests.

Prior to all testing, the space was configured using the facilities control system (FCS) to control the quantity of airflow being distributed to the space. The VAV boxes were set to provide approximately 200 CFM of airflow to each of the four diffusers, as seen in Figure 18. The layout of the supply diffusers in the controls systems can be seen in Figure 19.

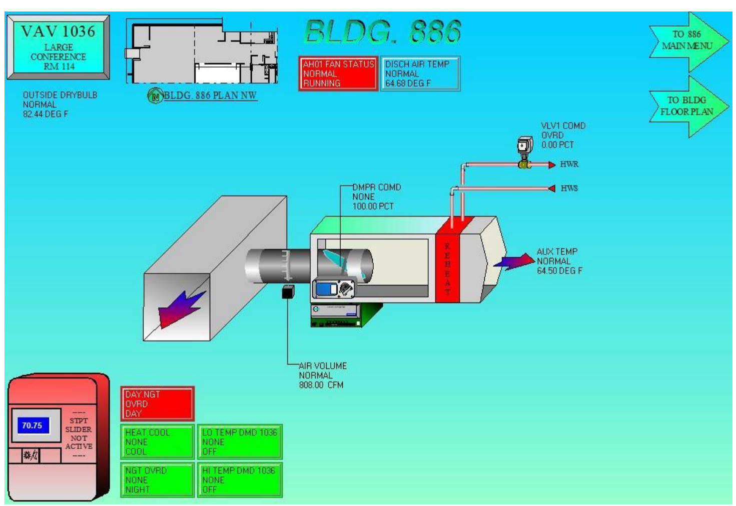

Figure 18. Screen shot of controls system set to 800 cfm.

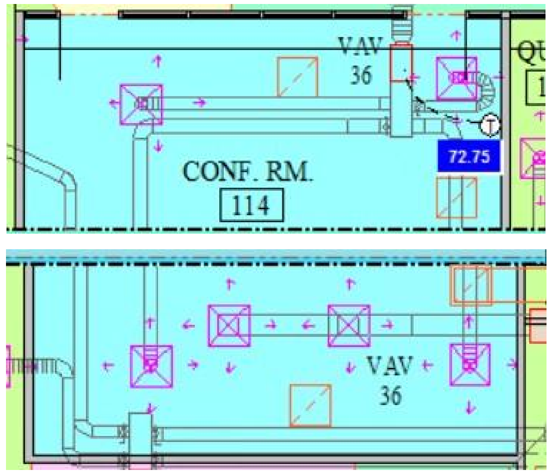

Figure 19. Monitoring the room in the controls system.

3.3.2. *Anemometer Testing*

The anemometers were spaced evenly throughout the room in locations where people would most likely sit during a meeting. As shown in Figure 20, locations 1 thru 8 were placed around the conference room table to simulate a well-attended meeting. Location 9 is considered the person running the meeting with control of the monitors and audio in the NE corner of the room. The height of each anemometer was set to a uniform length of 65" above the finished floor level with the directional flow sensor (anemometer probe) positioned to be perpendicular to the length of the table. This measured air flow in the same direction as a person breathing or talking during a meeting.

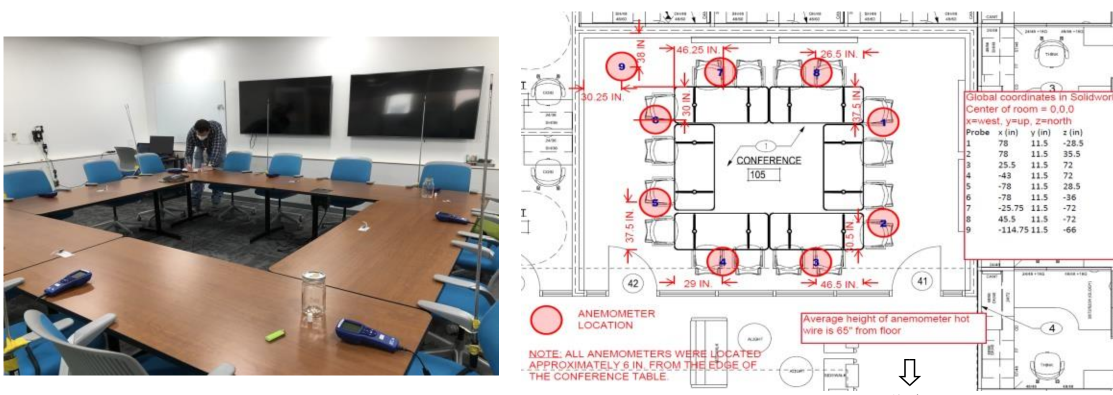

Figure 20. Anemometer placement in 886/114 before testing.

The four supply air diffusers are adjustable modular core directional supply allowing airflow in two or four directions into the space. When measuring the velocity for each quadrant on the diffuser, the anemometer face containing the probe was placed perpendicular to the airflow exiting the supply air diffuser and moved in a slow sweeping motion to record an average velocity across the quadrant of that diffuser. During the modified test only two quadrants were measured because the other two were covered. The return air grilles were measured in a similar method employed for the supply air diffusers.

Steady-state air velocity measurements were taken over a duration of 1 hour. Following multiple test scenarios, the data gathered by the equipment, as seen in Figure 21, were averaged and compared to similar coordinates within the SolidWorks CFD model.

Figure 21. Taking velocity measurements.

3.3.3. Tissue Tests

A "Tissue Test" was performed to visualize the airflow coming out of a supply air diffuser or entering a return air grille. The tool, as shown in Figure 22, was waved slowly across the supply air diffuser and or return air grille to test the direction of airflow and view that some airflow velocity was present from a specific diffuser or grille.

Figure 22. Tissue Test Tool used to understand airflow direction

3.3.4. Smoke Tests

Smoke tests were conducted using both the 90-second and 4-minute smoke emitters. The emitters were lit by hand and placed in the glass jars (Figure 23). The person(s) would then exit the room slowly, as to not create an air wake. All involved would then observe any visible air movements from behind the glass wall.

The first test was a pilot test employing a ninety second emitter, which was placed in the glass jar and partially covered in order to comprehend the volume of smoke that would be released from the emitter. It was quickly realized that one emitter would not generate enough smoke to fill the entire room but could be used to diagnose and validate single supply/return airflow directions. It was decided the next test would be to light three ninety second emitters at the same time to better fill the space. The three emitters created a greater amount of smoke, but obscured the view of observers looking through the glass wall of the conference room. It was therefore more difficult to evaluate the individual airflows. The air currents were discernable washing up along the edge of the interior glass wall. Several more tests were performed with the emitters placed at different locations throughout the room.

Figure 23. Observation of a smoke emitter during a test

3.3.5. Fog Purge Tests

A fog machine was used to provide additional visualization tests. Individual "puffs" of fog could be tracked, or the room could be filled with fog and then purged with clean air from the ventilation system to assess the air change rate and dissipation effectiveness for different ventilation configurations (Figure 24).

Figure 24. Purge test using emissions from fog machine

During the fog-purge tests, the fog machine was turned on for approximately twenty-five seconds to generate a large amount of fog (also referred to as "smoke"). The team observed and timed how long it took for the fog to dissipate to a point where all were in agreement the room was clear, which was approximately 10 - 15 minutes. This test was conducted for the mixed air and modified scenarios where the room overhead supply/return sir distribution was in its original state as well as with the minor modifications to the supply/return air configuration. The fog machine was determined to be the best way to visualize air exchanges.

4. RESULTS AND DISCUSSION

4.1. Anemometer Tests

Figure 25 shows the results of the anemometer tests together with the simulated air speeds from the CFD models. Results show that the average measured vs. simulated air velocities at the nine anemometer locations were within ~6% for original vents and 0.2% for modified vents. In addition, the measured and simulated air speeds were comparable and between ~1 – 30 ft/min at all anemometer locations. The peak diffuser velocities for the original and modified vents were measured to be ~1500 – 1600 ft/min. These tests and comparisons with the CFD simulations provided confidence in the models and simulation methods.

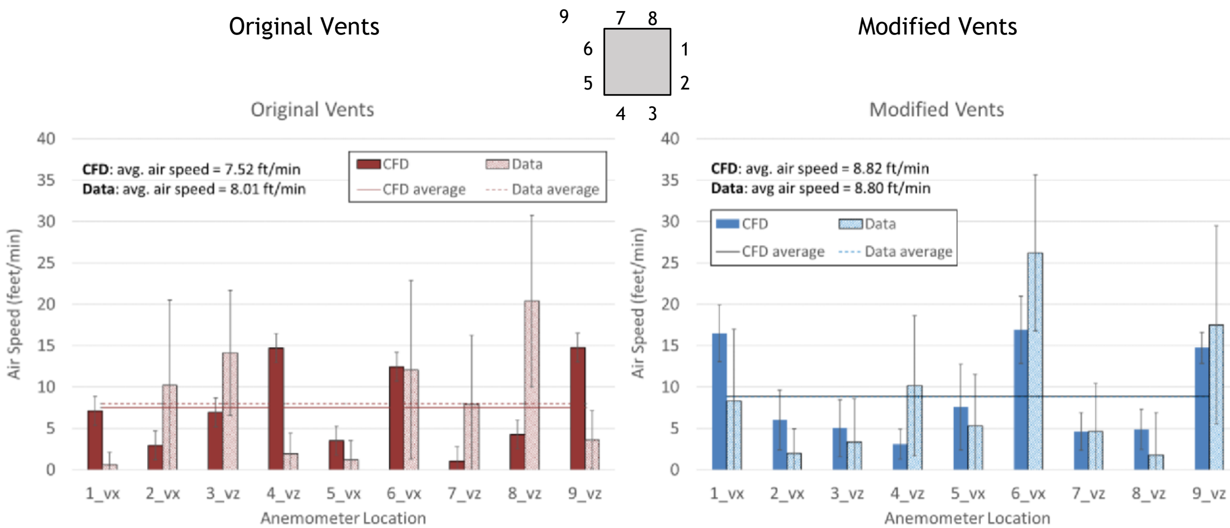

Figure 25. Simulated and measured air speeds at nine locations in 886/114 for original (left) and modified (right) vent configurations.

4.2. Tissue Tests

The Tissue Test was a simple procedure that was especially helpful for providing a swift validation of directional air flow prior to using any anemometers for testing. When modifying many spaces, the Tissue Test would be very useful to validate air direction and a general gauge of velocity.

The mixed airflow configuration had great visual results at the supply air diffusers. The multidirection supplies proved to not have even distribution of airflow; this was also noted during the anemometer measurements. One or two sides of the diffuser had much higher airflow than the other. This was not originally considered but proved to be a result of the routing of the ductwork attached to the diffuser. The sides that had more airflow were the side opposite the direction of the bend in the ductwork above the ceiling. The return air grilles in this configuration did not see much movement with the tool.

The modified airflow configuration had much better results. Blocking of half of the supply air diffuser showed that the air was moving in the direction intended. The single return air opening showed great improvement in velocity as shown in Figure 26.

Figure 26. Modified ventilation configuration tissue tests.

4.3. Smoke Tests and Intermittent Fog-Burst Tests

The primary findings from the smoke tests were the direction of local air flow for different ventilation configurations. The use of a smoke emitter was found to be most beneficial when locating individual currents at various locations within the conference room, especially at the supply diffusers and return grills. Emitters were located at all four corners of the conference room table. Several tests were done with multiple smoke emitters, which allowed several sections of the room to be observed at one time (Figure 27). When trying to flood the room with smoke, the emitters didn't provide enough smoke to fill the entire room even with four 90 second emitters. Hence, the fog tests were performed.

The use of a fog machine with intermittent bursts of fog helped to provide an understanding of the dynamics of air flow for the entire room. Unlike the smoke emitter, the fog machine would provide a burst of fog in a certain direction depending on how long the machine was activated. The fog machine best represents an expiratory event from an individual who is coughing, sneezing, or talking. Another benefit of the fog machine was that the remote control could be located outside the room, so no additional air currents were created when a person was leaving the room, which was a possibility during the use of the smoke emitters.

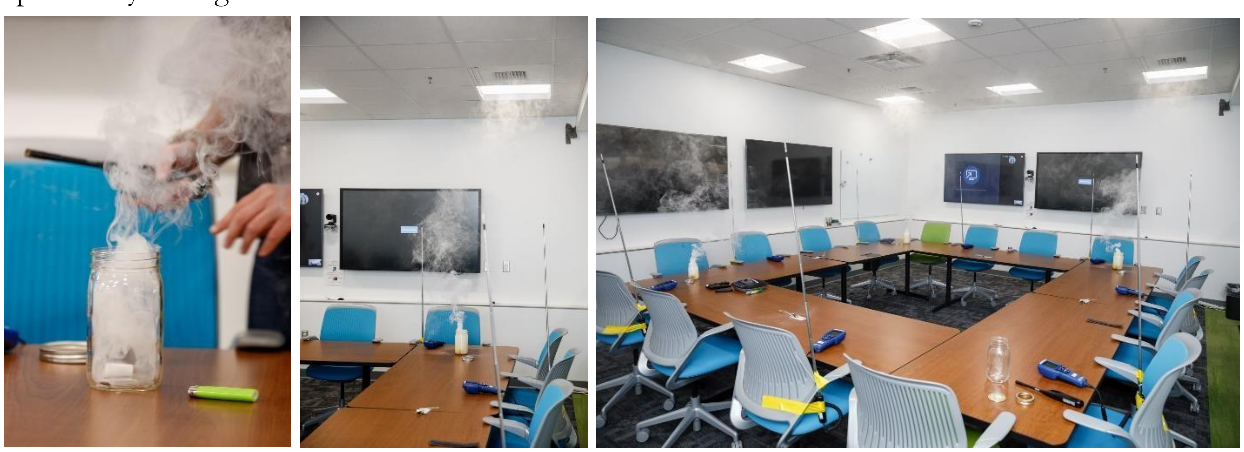

Figure 27. Showing how the smoke emitters were used and located in the room

4.4. Fog Purge Tests

Figure 28 shows a sequence of photos during fog-purge tests in which 886/114 was initially filled with fog and then allowed to be purged with ventilated air using the original and modified-vent configurations.

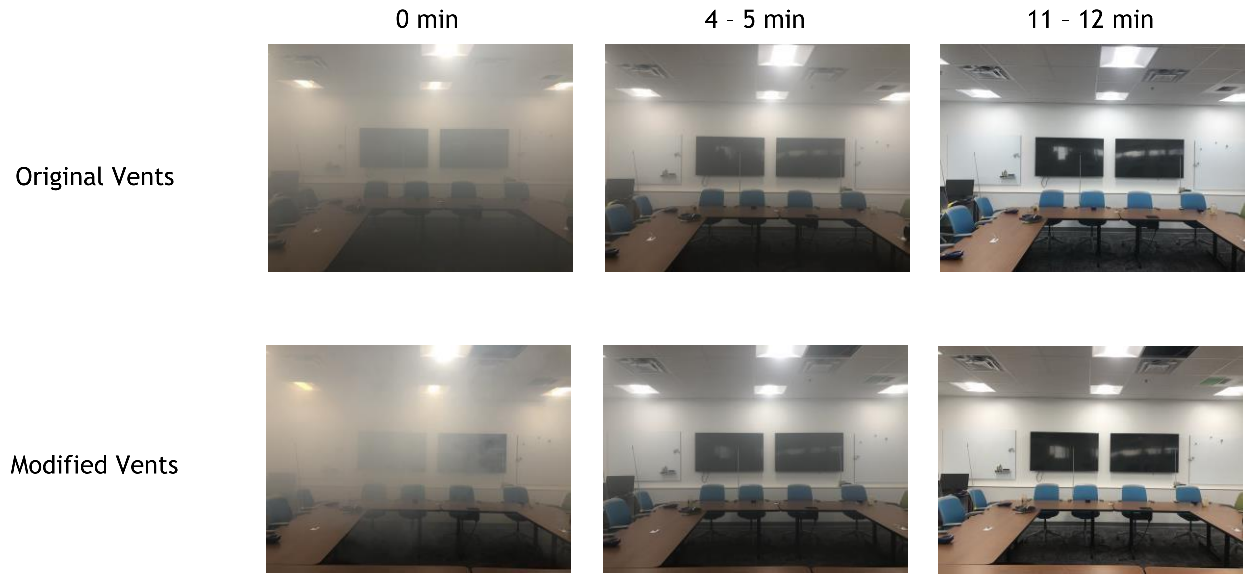

Figure 28. Sequence of images during fog-purge tests at ~600 cfm.

Figure 29 shows simulations of the fog-purge test in which the room was initially saturated with water vapor and then allowed to purge with ventilated air at the measured ambient relative humidity (~15%). Results show that the modified ventilation configuration was more effective at purging the room. At three minutes, the simulations show a noticeable reduction in the water vapor concentrations (relative humidity). The plots in Figure 29 show the simulated relative humidity along three transects through the middle of the room: up-down (y-sketch), east-west (x-sketch), and north-south (z-sketch).

Figure 30 shows the simulated transient relative humidities at the nine anemometer locations for the original and modified ventilation configurations. Results show that the modified vents provided a more spatially uniform and slightly faster purge throughout the room relative to the original ventilation configuration.

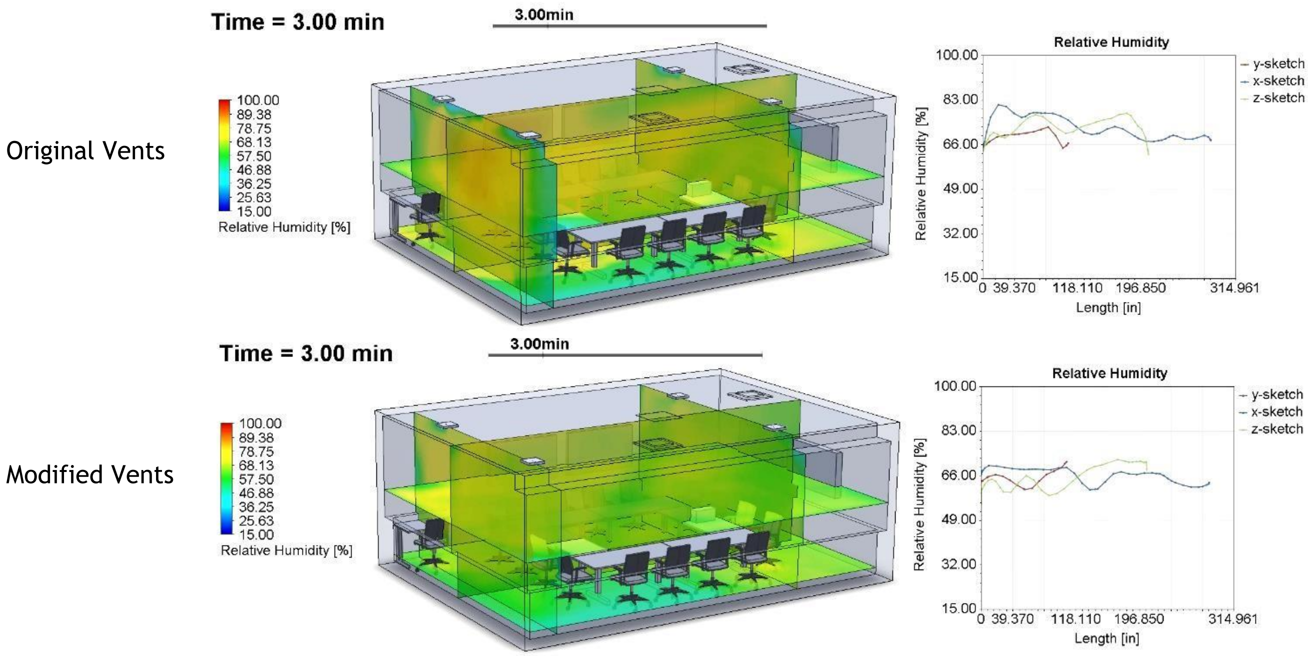

Figure 29. Simulations of original (top) and modified (bottom) vent configurations at 3 minutes into purge test.

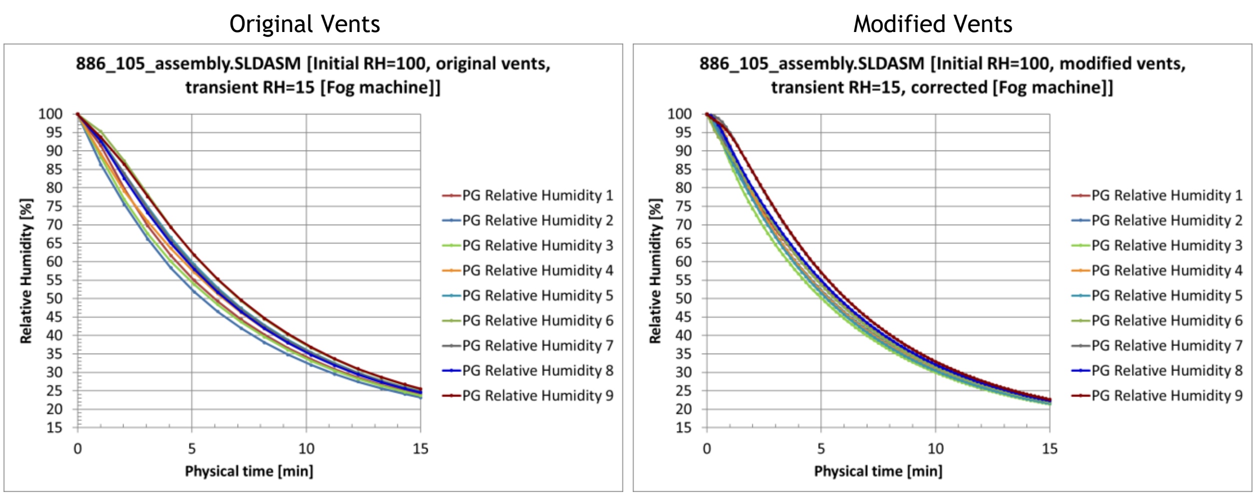

Figure 30. Simulated relative humidity at nine locations as a function of time for original (left) and modified (right) vent configurations in 886/114.

Figure 31 shows the normalized and spatially averaged simulated vapor concentrations as a function of time during the fog-purge tests for both the original and modified vents. The plot also includes vertical bands representing the time reported by five observers to "clear out" the room during the fog-purge tests. The blue vertical band represents the range of reported clear-out times for the modified vents (~12.5 min), and the vertical red band represents the reported range of clear-out times using the original ventilation configuration (~13 – 14 min). At ~12.5 min, the simulated normalized concentrations using modified vents was ~0.13. The corresponding time to reach 0.13 using the original vents was ~14 min, which was consistent with the experimental observations. These results further build confidence in the models and the general recommendation to arrange the supply and return vents to promote a once-through or through-flow condition.

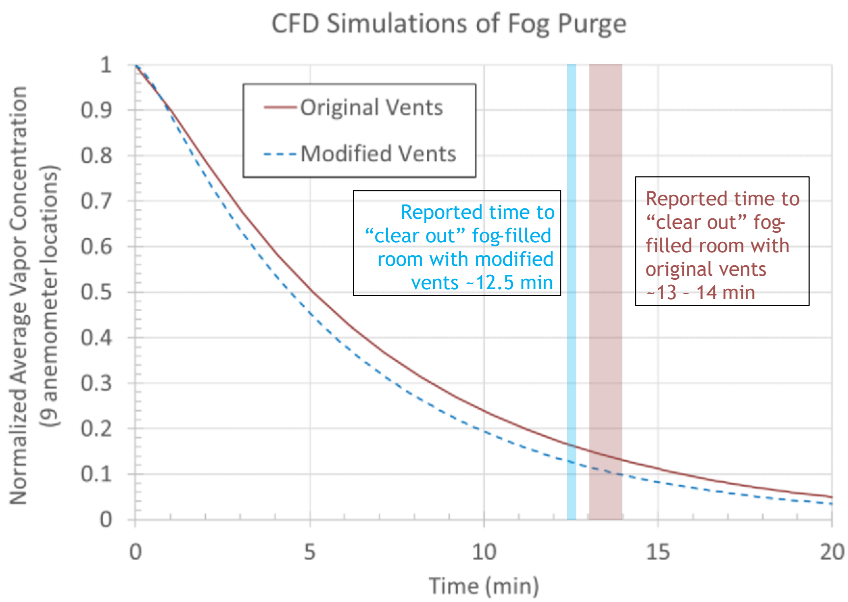

Figure 31. Simulation normalized average vapor concentrations over nine locations as a function of time for original and modified vent configurations in 886/114.

4.5. Discussion

The CFD simulations and corresponding tests confirmed that low-cost modifications to a space can improve ventilation effectiveness. Overall, these improvements limit exposure of occupants to airborne pathogens and are the next step forward in protection of the workforce. Like with any experiment, unexpected discoveries were made by the team while trying to validate a CFD model and make minor air flow modifications to improve ventilation. The following lessons learned may help to expedite future evaluation of other spaces.

4.5.1. Observations and Lessons Learned

During this study many observations were made at all phases. At the beginning when collecting information about the conference room, many discrepancies were discovered for a space that completed a major remodel project two years earlier. Examples include the following:

Do not assume that drawings match what is installed in the field. The HVAC system drawings were discovered to be incorrect and ductwork modifications were required to achieve a single VAV configuration as was originally planned. Unexpectedly, one of the supply air diffusers was found above the ceiling connected to the ductwork blowing air in the plenum space and another was found connected to a neighboring VAV.

Balancing the supply and return flow rates. The airflow measurements of the supply and return air were contrary to the values shown in the building design. The team looked at other conference rooms in the same building and discovered some similar issues. A validation should be conducted to ensure that the system is properly tested and balanced.

The use of a tissue to quickly identify the direction of airflow proved to be a valuable tool. While conducting the visualization tests, the 90-second smoke emitters were better for observations in a very small area. The 4-minute emitters filled the space to better see the airflow dynamics. Filling the entire room with smoke was an issue for observation since smoke activity could only be seen from one side of the space.

For the fog-purge study, the fog machine was the tool to use because it could be remote controlled. It provided the capability to adjust the amount of fog based on timed release intervals established during testing.

As a general observation, the only moving air that could be seen during the smoke tests was near the supply air diffusers, return air grilles, and some eddies at corners of the room. The standard design of mixing air in a space is a slow process. The most effective way to view air changes was to flood the space with fog and time how long it took until the space was considered "clear." In hindsight, having observers or cameras placed in different areas of the room may have been more beneficial to view and discern air flow within the space.

4.5.2. Recommendations

During the evaluation of the conference rooms, a list of items to inspect before developing the CFD model or collecting field data was generated. The following checklist is a suggestion as a general starting point for the evaluation a space.

Characterize the local HVAC system:

1. Collect and analyze the HVAC drawings for the space. Especially study the layout of the supply & return air paths, supply diffusers, return grilles, and air control boxes (Constant Volume, Variable Air Volume, or a Small Packaged Unit)
2. Validate the drawings with the actual space. This will require bringing a ladder and flashlight to observe the space above the ceiling.
3. Look for a current Test & Balance Report for the area. If the report is over 5 years old, it would be best to regenerate the report and make sure the system is performing as required.
4. Take lots of photos.

Modifying and correcting the HVAC system (if needed):

1. Correct the drawings to reflect the actual system
2. Correct the existing system to the design drawings
3. Verify the airflows on the HVAC drawings and Test & Balance Report are within reason

5. CONCLUSIONS AND NEXT STEPS

5.1. Key Findings

This study explored the impact of different ventilation configurations on potential exposure risks in conference rooms and office settings. Key findings from this study include the following:

- CFD simulations can be used to characterize the general airflow patterns and exposure risks in a room as a function of ventilation configuration, room layout, and occupancy. ○ CFD simulations were compared against tests in 886/114 that measured air velocities and air-change effectiveness to build confidence in the models and simulation methods.
- Plexiglass barriers did not significantly reduce exposure risks to occupants in simulations; however, it is important to note that talking, coughing, and sneezing were not simulated, and transmission by direct spray from these events could be mitigated by barriers.
- Limiting the quantity of occupants in an enclosed space potentially reduces the number of those in the transmission path; however, the room occupancy did not have a significant impact on room air concentrations in the simulations.
- A once-through (through-flow) airflow configuration was shown to improve the air exchange rate in a space by removing air more quickly and directionally
- The results and recommended ventilation designs from this study will be generally applicable to minimize risks of airborne transmission for future seasonal flu outbreaks or epidemics

5.2. Next Steps

During testing, the 886/114 conference room was determined to have poor air balance in general. Airflows and velocities were inconsistent throughout which could be problematic in providing well distributed airflow. The return air locations should be relocated to the center of the room to promote unidirectional flow. A test and balance contractor will be engaged to re-balance the airflow distribution in the space to provide better space conditioning.

Next steps include the following:

- Rebalance the 886-conference room
- Relocate the return air grilles within the 886-conference room to the center of the room ○ Perform additional modeling and testing of alternate office configurations that have multiple occupants such as open office cubicles, auditoriums, and break rooms
- Modify existing spaces by relocating return air grilles
- Improve space design by strategically placing supply air and return air vents to promote a once-through airflow configuration
- If additional tests and studies confirm the findings in this study, modify the Design Manual (MAN-004, Section 8.13, Ductwork Design) to incorporate recommended changes to promote through-flow conditions to minimize airborne exposure risks

Page Left Blank

REFERENCES

[1] Centers for Disease Control and Prevention (CDC). *Scientific Brief: SARS-CoV-2 Transmission.* [Accessed September 19, 2021]; Available from: https://www.cdc.gov/coronavirus/2019https://www.cdc.gov/coronavirus/2019-ncov/science/science-briefs/sars-cov-2-transmission.htmlncov/science/science-briefs/sars-cov-2-transmission.html.

[2] van Doremalen, N., T. Bushmaker, D.H. Morris, M.G. Holbrook, A. Gamble, B.N. Williamson, A. Tamin, J.L. Harcourt, N.J. Thornburg, S.I. Gerber, J.O. Lloyd-Smith, E. de Wit, and V.J. Munster, 2020, Aerosol and Surface Stability of SARS-CoV-2 as Compared with SARS-CoV-1, *New England Journal of Medicine*, **382**(16), p. 1564-1567.

[3] American Society of Heating Refrigerating and Air-Conditioning Engineers (ASHRAE). *Guidance for Re-Opening Buildings* [Accessed 9/19/21]; Available from: https://www.ashrae.org/file%20library/technical%20resources/covid-19/guidance-for-rehttps://www.ashrae.org/file library/technical resources/covid-19/guidance-for-re-opening-buildings.pdfopening-buildings.pdf.

[4] ACGIH and ASHRAE White Paper Series. *Ventilation for Industrial Settings During the COVID-19 Pandemic.* [Accessed American Conference of Governmental Industrial Hygienists (ACGIH), Industrial Ventilation Committee, and American Society of Heating Refrigerating and Air-Conditioning Engineers (ASHRAE), Technical Committee 9.2 Industrial Air Conditioning and Ventilation, ; Available from: https://1lnfej4c7wie44voctzq1r57https://1lnfej4c7wie44voctzq1r57-wpengine.netdna-ssl.com/wp-content/uploads/2021/07/ASHRAE-ACGIH-COVID-19-White-Paper_2021-07-26.pdfwpengine.netdna-ssl.com/wp-content/uploads/2021/07/ASHRAE-ACGIH-COVID-19https://1lnfej4c7wie44voctzq1r57-wpengine.netdna-ssl.com/wp-content/uploads/2021/07/ASHRAE-ACGIH-COVID-19-White-Paper_2021-07-26.pdfWhite-Paper_2021-07-26.pdf.

[5] Khosronejad, A., C. Santoni, K. Flora, Z.X. Zhang, S. Kang, S. Payabvash, and F. Sotiropoulos, 2020, Fluid dynamics simulations show that facial masks can suppress the spread of COVID-19 in indoor environments, *Aip Advances*, **10**(12).

[6] Fennelly, K.P., 2020, Particle sizes of infectious aerosols: implications for infection control, *Lancet Respir Med*, **8**(9), p. 914-924.

[7] Bivolarova, M., J. Ondracek, A. Melikov, and V. Zdimal, 2017, A comparison between tracer gas and aerosol particles distribution indoors: The impact of ventilation rate, interaction of airflows, and presence of objects, *Indoor Air*, **27**(6), p. 1201-1212.

[8] Gupta, J.K., C.H. Lin, and Q.Y. Chen, 2011, Transport of expiratory droplets in an aircraft cabin, *Indoor Air*, **21**(1), p. 3-11.

[9] Zhang, Z., X. Chen, S. Mazumdar, T.F. Zhang, and Q.Y. Chen, 2009, Experimental and numerical investigation of airflow and contaminant transport in an airliner cabin mockup, *Building and Environment*, **44**(1), p. 85-94.

[10] Feng, Y., T. Marchal, T. Sperry, and H. Yi, 2020, Influence of wind and relative humidity on the social distancing effectiveness to prevent COVID-19 airborne transmission: A numerical study, *Journal of Aerosol Science*, **147**.

[11] Dassault Systems, 2019, Technical Reference Solidworks Flow Simulation 2019, www.solidworks.com.

[12] Schwarzkopf, J.D., D. Livescu, J.R. Baltzer, R.A. Gore, and J.R. Ristorcelli, 2016, A Twolength Scale Turbulence Model for Single-phase Multi-fluid Mixing, *Flow Turbulence and Combustion*, **96**(1), p. 1-43.

[13] Seshadri, P.K. and A. De, 2020, A novel sharp interface immersed boundary framework for viscous flow simulations at arbitrary Mach number involving complex and moving boundaries, *Computers & Fluids*, **206**.

[14] Burns, A.D., T. Frank, I. Hamill, and J.M. Shi, 2004, *The Favre Averaged Drag Model for Turbulent Dispersion in Eulerian Multi-Phase Flows*, in *5th International Conference on Multiphase Flow, ICMF'04*, Paper No. 392, Yokohama, Japan, May 30 - June 4, 2004.

DISTRIBUTION

Email—Internal

Name	Org.	Sandia Email Address
Technical Library	01977	sanddocs@sandia.gov

Sandia National Laboratories

Sandia National Laboratories is a multimission laboratory managed and operated by National Technology & Engineering Solutions of Sandia LLC, a wholly owned subsidiary of Honeywell International Inc. for the U.S. Department of Energy's National Nuclear Security Administration under contract DE-NA0003525.