PNNL-27826

Pacific Northwest NATIONAL LABORATORY

Qualification of ANSI/HPS N13.1-2011 Mixing Criteria by Computational Fluid Dynamics Modeling for the 3420 Building Fan Addition and Increased Ventilation Capacity

KP Recknagle
SR Suffield
JM Barnett

August 2018

U.S. DEPARTMENT OF ENERGY

Prepared for the U.S. Department of Energy
under Contract DE-AC05-76RL01830

PACIFIC NORTHWEST NATIONAL LABORATORY
operated by
BATTELLE
for the
UNITED STATES DEPARTMENT OF ENERGY
under Contract DE-AC05-76RL01830

Printed in the United States of America

Available to DOE and DOE contractors from the
Office of Scientific and Technical Information,
P.O. Box 62, Oak Ridge, TN 37831-0062;
ph: (865) 576-8401
fax: (865) 576-5728
email: reports@adonis.osti.gov

Available to the public from the National Technical Information Service
5301 Shawnee Rd., Alexandria, VA 22312
ph: (800) 553-NTIS (6847)
email: orders@ntis.gov <https://www.ntis.gov/about>
Online ordering: http://www.ntis.gov

PNNL-27826

Qualification of ANSI/HPS N13.1-2011 Mixing Criteria by Computational Fluid Dynamics Modeling for the 3420 Building Fan Addition and Increased Ventilation Capacity

KP Recknagle JM Barnett
SR Suffield

August 2018

Prepared for
the U.S. Department of Energy
under Contract DE-AC05-76RL01830

Pacific Northwest National Laboratory
Richland, Washington 99352

Executive Summary

Additional ventilation capacity has been designed for the 3420 Building filtered exhaust stack system. The updated system will increase the number of fans from three to four and will include ductwork to integrate the new fan into the existing stack. Stack operations will involve running various fan combinations at any given time. The air monitoring system of the existing three-fan stack was previously found to comply with the ANSI/HPS N13.1-1999 standard. Full-scale, three-dimensional computational fluid dynamics (CFD) modeling was used to evaluate the modified four-fan system for compliance with the ANSI/HPS N13.1-2011 standard, which essentially is equivalent to the ANSI/HPS N13.1-1999 standard. The four mixing criteria evaluated are 1) flow angle, 2) velocity, 3) gas tracer, and 4) particle tracer.

Benchmarking of the CFD modeling methodology showed good agreement with previous testing used to qualify the stack, and modeling of the existing three-fan system showed good agreement with test data collected from the 3420 Building stack. Modeling was performed to develop a suitable four-fan design. Initial modeling of the four-fan design and basic ductwork showed that flow angles and velocity uniformity were acceptable; however, the gas tracer and particle tracer mixing results were not acceptable. To meet ANSI/HPS N13.1-2011 criteria, an air blender was added to the stack design. The modeled final four-fan design, which meets the all the mixing criteria, included an air blender that is oversized relative to the main duct diameter to minimize the additional pressure drop created.

Acronyms and Abbreviations

3-D	three-dimensional
ANSI	American National Standards Institute
HPS	Health Physics Society
CFD	computational fluid dynamics
COV	coefficient of variance
PNNL	Pacific Northwest National Laboratory

Contents

Figures

Tables

1.0 Introduction

The 3420 Building at Pacific Northwest National Laboratory (PNNL) houses radiological capabilities so air discharged from the building filtered exhaust stack system must be monitored for radionuclides. The air monitoring system must comply with applicable federal regulations, which subsequently require a sampling probe in the exhaust stream to conform to the uniformity criteria of the ANSI/HPS N13.1-2011 standard. The criteria include the average angle between the flow and duct axis, the uniformity of flow velocity, the uniformity of tracer gas, and the uniformity of tracer particles. The uniformity is expressed by the coefficient of variance (COV), which is defined as the standard deviation divided by the mean. For a sampling location to be acceptable, the average flow angle must be less than 20° from the duct axis (aligned with the sample probe) to prevent cyclonic flow, COVs for velocity, tracer gas concentration, and tracer particle concentration must be less than 20%. An additional criterion is that at no point in the sampling plane will the maximum concentration of tracer gas exceed the mean by more than 30%.

An option in the ANSI/HPS N13.1-2011 standard allows adoption of results from a previously performed full test series for a stack system of similar configuration as the basis of compliance with the standard. Compliance then is confirmed by partial testing performed on the actual stack system. This approach was used to qualify the location of the monitoring probe and configuration of the original three-fan 3420 Building filtered exhaust stack as documented by Glissmeyer and Flaherty (2010). This testing was performed on the actual system and included flow angle and velocity uniformity measurements. The previous full test series applied as the basis for compliance was that performed on a scale model of the Waste Treatment Plant's HV-C2 air exhaust stack by Glissmeyer and Droppo (2007). The HV-C2 stack, with two fans entering a horizontal main duct, both at 45 degree angles, is similar to the original configuration of the 3420 Building exhaust stack.

The original testing of the HV-C2 scale model was performed to establish the sampling probe location for the actual HV-C2 stack (Glissmeyer and Droppo 2007). The scale model showed small flow angles and good velocity uniformity. However, tracer gas/particle test COV values were greater than 20% at all but the test port furthest downstream. This is not surprising because a substantial length of duct is required to achieve the fully developed flow needed to provide mixing energy. For turbulent flow, this flow development length is considered to be roughly independent of the Reynolds number, and is at least 10-diameters of length from the last disturbance (Incropera and DeWitt 1985). The furthest test port on the HV-C2 scale model is similar in scaled distance to that of the 3420 Building sampling location. Thus, it is reasonable to expect, all of the main duct length of the 3420 Building exhaust system will be needed to provide sufficient mixing of tracer gas and tracer particles.

The 3420 Building exhaust stack system will be updated with additional ventilation capacity. The updated system will incorporate a fourth fan and associated ductwork to integrate the new fan into the existing stack. As a result, the stack configuration will be changed substantially. The nominal operating condition will have three fans operating with one fan in standby. The average overall flow rate also will be increased significantly. In the absence of data from a similar system, it is not known if the updated four-fan system will qualify as readily as the three-fan system. Therefore, before making a final decision on installation of a proposed design, a decision was made to use modeling to gain more insight into the expected performance of the modified stack and sampling location. The final modeled design with four fans includes an oversize air blender to improve gas and particle mixing. The modeled design effectively acts as a similar stack design. Therefore, the differences between the flow angle and velocity uniformity COV of the full-scale stack testing and those modeled cannot be more than 5° or 5% COV respectively, to confirm the stack system and sampling location are equivalent with the ANSI/HPS N13.1-2011 standard.

2.0 Modeling Methodology

The purpose of modeling the 3420 Building stack system is to simulate the stack flow, including distributions of gas and particle tracers, to assist in determining if the modified system will satisfy the ANSI/HPS N13.1-2011 standard. To provide accurate predictions of flow angle, velocity, tracer gas, and tracer particle distributions (at the sampling location) requires an accurate prediction of the turbulent air flow with transport and mixing of the tracer species within it. The geometry and flow field of the exhaust stack system is complex and highly three-dimensional (3-D). Therefore, a representative boundary-fitted, 3-D flow model also was required. The commercially available, computational fluid dynamics (CFD) flow simulation code, STAR–CCM+ (Siemens 2017) was selected for creation of the 3-D model domain and the flow simulations.

PNNL has been modeling stack designs for compliance for the past 15 years. During this time, CFD modeling has been shown to be useful both in the design process and as an effective predictor of flow angles and velocity and tracer COVs. The use of CFD modeling at PNNL to examine the flow and mixing performance in building filtered exhaust stacks and evaluate sampling point locations was first presented at the Annual HPS Meeting in San Diego, CA. (Barnett, Ballenger, and Recknagle 2003). Peer-reviewed publications authored by PNNL staff include: Modeling of the 325 Building exhaust stack system to evaluate relocation of the sampling point (Barnett et. al 2005, Recknagle et. al 2009), Modeling and testing to assess the 3410 Building exhaust stack sampling probe location (Yu et. al 2014), and modeling of a modified building stack for sampling compliance (Barnett et. al 2016). Internal reports include: sampling point compliance tests and modeling of the 325 Building at set-back flow conditions (Ballinger et. al 2011), sampling point compliance modeling of the 3410 Building with the addition of a third fan and the correct recommendation to add an air blender (Recknagle et. al 2013). Most recently, a presentation (and journal article to follow) on modeling building stack sampling points for qualification criteria (Recknagle et. al 2018) was presented by S.A. Suffield at the 1st International Symposium on Mechanics. Scotland, U.K. in July 2018. The present modeling for the 3420 Building stack was performed using the same modeling methodology applied in our previous work.

2.1 Flow Model

The stack sampling methodology assumes isothermal conditions exist within the stack; therefore, that assumption was adopted in the flow model. For isothermal flow solutions, STAR–CCM+ solves the Navier-Stokes conservation of mass and momentum equations, which for steady-state compressible and incompressible fluid flows are:

$$\frac{\partial}{\partial x_j}\left(\rho u_j\right)=0 \tag{1}$$

$$\frac{\partial}{\partial x_j}\left(\rho u_j u_i-\tau_{ij}\right)=-\frac{\partial p}{\partial x_i} \tag{2}$$

where the u_i are the absolute fluid velocity components in coordinate directions x_i (i = 1, 2, 3), ρ is the density, p is the pressure, and τ_{ij} is the fluid stress tensor, which for turbulent flows is represented by:

$$\tau_{ij} = 2\mu\sigma_{ij} - \frac{2}{3}\mu\frac{\partial u_k}{\partial x_k}\delta_{ij} - \bar{\rho}\overline{u_i'u_j'} \tag{3}$$

where μ is the dynamic viscosity, σ_{ij} is the rate of strain tensor, δ_{ij} is the Kronecker delta, u_i and u_j are fluctuations about the average velocity, and the overbar indicates the averaging of the fluctuations. The right-most term in Equation 3 represents the additional Reynolds stresses due to turbulent motion. These stresses are linked to the mean velocity via the turbulence model being used. In the simulations for this work, the generation and dissipation of turbulence is accounted for using a realizable κ-ε turbulence model, which is a widely tested and validated two-equation closure model for the Reynolds average Navier-Stokes equations, as described in the STAR-CMM+ User Guide (Siemens 2017). In past work by Recknagle et al. (2009), a turbulence model comparison found the Reynolds average Navier-Stokes κ-ε model to be the most suitable for simulating duct flow, a finding corroborated by Jensen (2007).

2.2 Gas Tracer Model

For the N_2O tracer gas simulations, the model assumes each species k of a gas mixture, with local mass fraction Y_k is governed by a species conservation equation of the form:

$$\frac{\partial}{\partial x_j}\left(\rho u_j Y_k + F_{k,j}\right) = S_k \tag{4}$$

where $F_{k,j}$ is the gas diffusional flux component and S_k is the gas species source term, which is non-zero at the injection location.

2.3 Particle Tracer Model

A Lagrangian dispersed two-phase flow model is used for the particle transport simulations. The Lagrangian methodology considers the interactions of mass, momentum, and energy between the continuum and dispersed phase. In general, motion of the dispersed phase is influenced by that of the continuous phase and vice versa. The strength of the phase interactions depend on concentration, size, and density of the dispersed particle. For the present work, particle concentrations are small, as is the nominal particle size, thus momentum transfer from particles to air is negligibly small. In the model, the momentum equation for a particle, given by Newton's second law, is:

$$m_d\frac{du_d}{dt} = F_{dr} + F_p + F_b \tag{5}$$

where m_d and u_d are the mass and velocity of the dispersed particle phase, F_{dr} is the drag force, F_p the pressure force, and F_b is body forces including effect of the gravity and angular velocity vectors. Surface vapor pressure and mass transfer between phases is not considered here. The problem is considered isothermal and does not involve electrically charged flow; therefore, thermophoresis and electrostatic effects are not included. Because of the low concentration of the particles, separation and coalescence models also were not considered.

2.4 Model Geometry and Computational Mesh

Design drawings or computer aided design software drawings of the stack system of interest were used to create 3-D geometry models of the system. The model geometry for the initial four-fan, 3420 Building system design is shown in Figure 2.1. Air flow upstream and through the fans is not included in the model domain but is accounted for as turbulence added at each fan duct. Thus, the model domain includes the ductwork from just downstream of the fans to the stack exit. Typical tracer injection locations are mid-duct, just downstream of the fans. The sampling point is located 79.6 ft downstream of Fan A, or about 15.4 diameters for the duct with a 62-inch diameter.

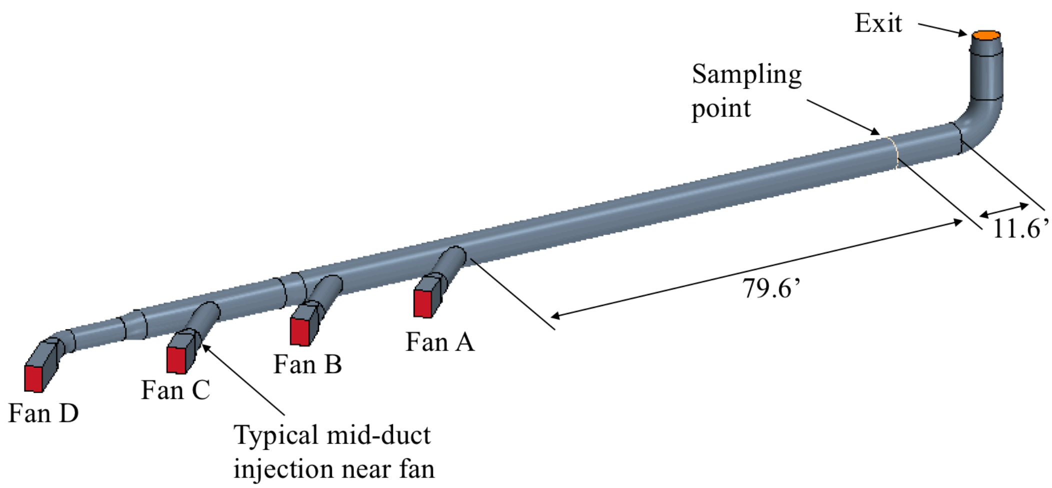
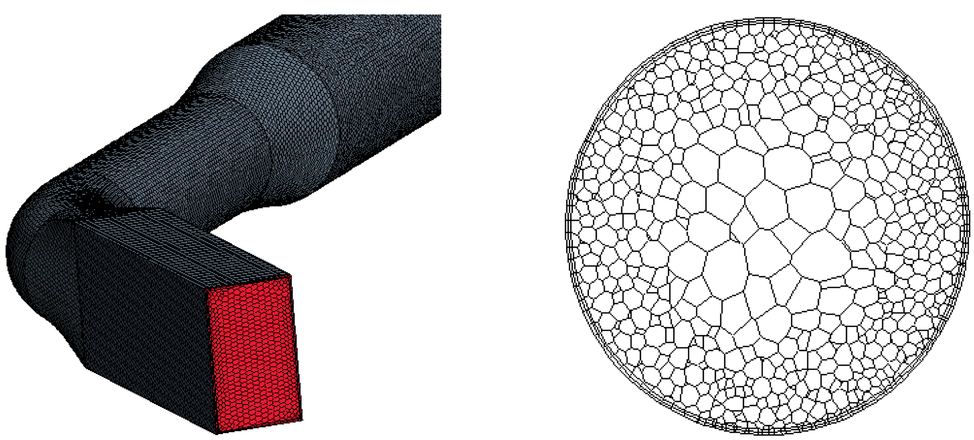


Figure 2.1. Model Geometry for the Initial Four-Fan 3420 Building Stack System Design.

The computational mesh is sufficiently refined to enable resolution of the turbulent flow field and provide accurate calculations of the gas and particle mixing throughout the system. The computational mesh used for the simulations contains approximately 1.2 million elements; Figure 2.2a provides a view of the mesh near Fan D. The typical resolution throughout the volume mesh is represented Figure 2.2b.



(a) (b)

Figure 2.2. Detail of Computational Mesh at the (a) Surface Near Fan D, and (b) Typical Cross-Section of the Volume Mesh in the Main Duct.

2.5 Boundary Conditions

Mass inflow boundaries were established at the duct inlets with turbulence intensity and length scale settings to account for upstream turbulence. A pressure boundary with 1 atmosphere absolute pressure was used at the stack exit. Duct walls were modeled as smooth surfaces with zero slip flow boundary conditions. The particle boundary condition at the walls was established so particles with trajectories that impact the duct walls would not stick to the surface.

3.0 Stack Model Benchmarking

The simulation cases presented in this section demonstrate the capability of the described CFD modeling methodology to suitably characterize the flow and sampling performance of an effluent stack. Validation of the methodology is achieved by simulations that provide a reasonable match of flow angle, velocity uniformity, gas tracer, and particle tracer data taken from actual stack performance testing.

The existing 3420 Building three-fan stack was tested for flow angle and velocity uniformity (Glissmeyer and Flaherty 2010) while inferring the results for tracer gas and tracer particle sampling efficiency from data collected during previous tests (Glisssmeyer and Droppo 2007) to determine if the stack meets the qualification criteria given in the ANSI/HPS N13.1-1999 standard. The inferred testing data was collected from a scaled physical model of the proposed design for the Hanford Waste Treatment Plant HV-C2 air exhaust stack, which is geometrically similar to the 3420 Building stack.

A 3-D CFD model (of the HV-C2 physical model) was created and set up to replicate the geometry and flow conditions tested. Figure 3.1 is a photograph of the assembled physical model. The locations of Fans A and B (and their injection ports) and Test Ports 1, 2, and 3 along the main duct are shown in the photograph. Figure 3.2 shows the geometry and computational domain of the associated 3-D CFD model. The CFD model domain included the full duct from immediately downstream of the fans to the duct exit, and with mesh resolution similar to that discussed in the previous section.

Figure 3.1. HV-C2 Physical Test Model

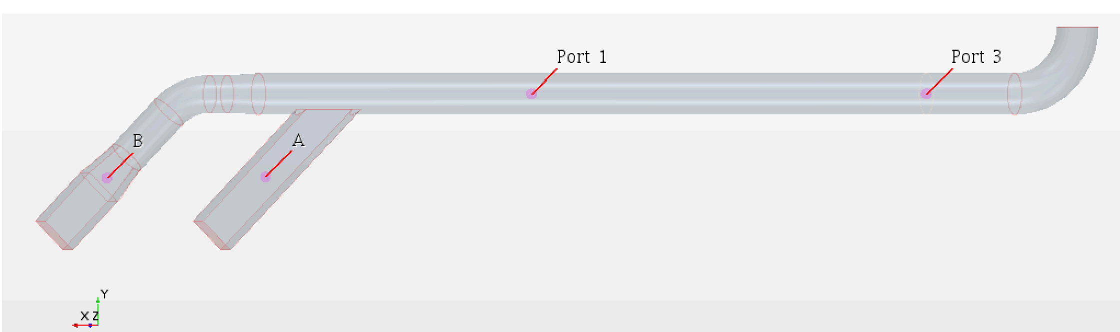


Figure 3.2. HV-C2 3-Dimensional Computational Fluid Dynamics Model

Stack testing included the collection of flow angle, velocity uniformity, gas tracers, and particle tracer data for operations of one or two fans with various data collected at Test Ports 1, 2, and 3. The model was run for comparison with the full suite of testing data at Test Port 1 to establish a benchmarking baseline. Additionally, the sampling location in the 3420 Building stack scales to most closely match Test Port 3. As such, gas and particle tracer data collected there was of interest when checking the results obtained from model benchmarking runs.

Table 3.1 summarizes the CFD modeling benchmark/testing data comparisons. The first two rows compare flow angle and velocity uniformity measured in testing and predicted by the model, showing acceptably similar values. Rows 3 and 4 compare testing data and CFD results for tracer gas and particles during operation of Fan A only. Note the large (tested and modeled) COVs for particle distributions at Port 1, which does not take advantage of the full stack length for mixing. When operating with multiple fans both gas and particle tracers are challenged to mix well as the air streams exiting each fan require many duct diameters of flow length to blend. Regardless, when operating both fans, gas and particle-tracer COVs were within the 20% COV limit for data collected at Port 3 (values shown in row 5 of Table 3.1). This data was sufficient for use in qualifying the original 3-fan 3420 Building stack sampling location. The CFD model predictions of gas and particle COVs (row 6) were similar to those measured during testing.

Table 3.1. Comparison of HV-C2 CFD Modeled Results with Data from Tests

	Type	**Fans Run**	**Test Port**	**Flow Angle**		**Velocity Uniformity**		**Gas Tracer**		**Particle Tracer**	
				Test	Angle	Test	COV	Test	COV	Test	COV
1	Test	A, B	1	FA-1	4.6°	VT-5	5.5	-	-	-	-
2	CFD	A, B	1	FA-1	3.8°	VT-5	4.6	-	-	-	-
3	Test	A	1	-	-	-	-	GT-13	2.9	PT-5	27.1
4	CFD	A	1	-	-	-	-	GT-13	3.4	PT-5	33.4
5	Test	A, B	3	-	-	-	-	GT-20	10.5	PT-9	17.6
6	CFD	A, B	3	-	-	-	-	GT-20	11.1	PT-9	19.5

Prior to modeling the performance of the initial four-fan design for the 3420 Building duct, the existing three-fan system was modeled as an additional benchmarking case for the CFD stack modeling methodology. The 3-D model was the same as described in Section 2 and shown in Figure 2.1 without the additional duct for the fourth fan, Fan D. Testing of this system is described in Glissmeyer and Flaherty (2010) in which the duct was operated at the nominal flow rate of 56,023 cfm to test flow angle (FA-1) and velocity uniformity (VT-1). The CFD model went beyond testing and included the injection of gas and particle tracers to check those COV values for the existing system.

Table 3.2 summarizes the flow angle and velocity uniformity measured at the 3420 Building duct and predicted by the CFD model. In this nominal flow rate case, all three fans were operating with open sashes. Agreement between the CFD model and testing data for flow angle and velocity uniformity is good. As expected, COVs for gas and particle tracers predicted by the CFD model also are acceptable.

Table 3.2. Comparison of 3420-Building Stack CFD Model Results and Data from Tests (sashes open, operating at 56,023 cfm running all three fans).

	Flow Angle		**Velocity Uniformity**		**Gas Tracer**	**Particle Tracer**
	Test	Angle	Test	COV	COV	COV
Test Results	FA-1	1.9°	VT-1	3.5	-	-
CFD Model Results	FA-1	2.1°	VT-1	3.3	7.47	16.0

These benchmarking exercises demonstrate the capability of the CFD modeling methodology to suitably simulate effluent stack operation and sampling location performance for a stack similar to that at the 3420 Building.

4.0 Stack Modeling Results

In this section, we discuss results from CFD simulations of the four-fan exhaust system. The simulations were undertaken to examine the mixing performance of the system when operating at design conditions. The simulation cases include three- and four-fan operations, and set back conditions involving two-fan and single-fan operations. Section 4.1 presents findings of the performance of the initial four-fan system design. Section 4.2 presents the work done to incorporate a 110-inch stationary air blender into the duct with minimal added pressure drop. Section 4.3 presents the resulting flow and mixing performance of the duct design including air blender.

4.1 Modeling the Performance of the Initial Four-Fan System Design

Operation of the modified exhaust system involved running in several different modes. All expected flow conditions must be examined to determine if any will fail to meet the ANSI/HPS N13.1-2011 standard. The flow conditions are described as follows:

- *Nominal Flow*: The 70-kcfm nominal flow is established by the operation of three fans at 93%, with one fan held in reserve. All three-fan combinations must be considered with the tracer injection location from the fan nearest the stack exit (i.e., the shortest duct length for mixing).
- *High Flow*: The 80-kcfm nominal high flow requires all four fans to operate at 80%. In this case, the tracers could be injected at any of the four fans, although the fan nearest the stack exit should present the greatest challenge for mixing.
- *Maximum Flow*: The 100-kcfm maximum flow operation requires all four fans to operate at 100%. Tracer injection is that described above for the High Flow Condition.
- *Set-Back Flow*: Minimal flow operations include a single fan operating at 66%, a flow rate of 16.5 kcfm, and tracer injection at the operating fan; or two fans operating at 66%, a flow rate of 33 kcfm, and tracer injection at either operating fan.

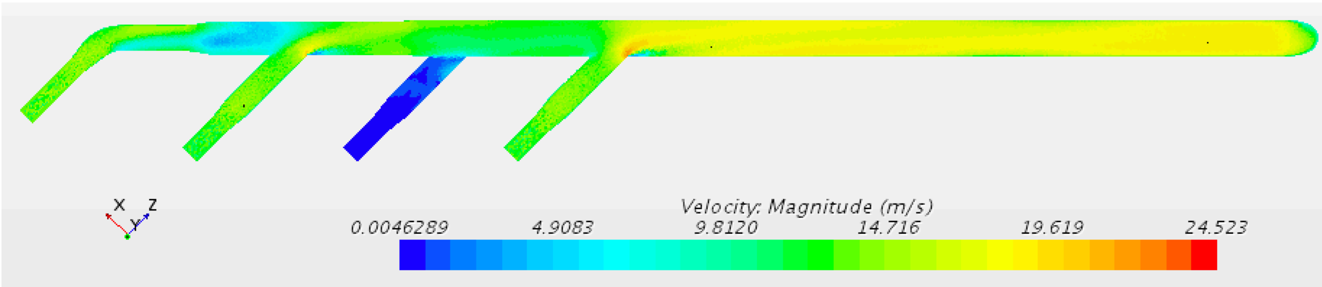
Simulation results for the initial four-fan system (shown in Figure 2.1, no air blender) are mixed. In all cases, the flow angle and velocity uniformity criteria are easily met, but tracer distributions at the sampling point result in elevated COVs, some of which fail to meet the standard mixing criteria. Results are shown in Table 4.1. Three of the nominal flow (70 kcfm) fan combinations passed the gas- and particle-tracer COV criteria, but the case in which Fans A, C, and D were operating with tracer injection near Fan A failed to sufficiently mix both tracer types. For High Flow (80 kcfm) four-fan operations, gas-tracer mixing failed at the three of the four injection locations, and particle-tracer mixing failed at all four injection locations. Only when tracers were injected near all four fans did the gas- and particle-tracer COVs meet the standard criteria. Because of the failures at High Flow operating conditions, a single Maximum Flow case in which the injection location was near Fan D was attempted. This should have been the most likely case to pass the mixing criteria, although it failed in the case of particle COV. A single-fan Set Back Flow case also was attempted for operation of Fan A, and it failed the particle COV.

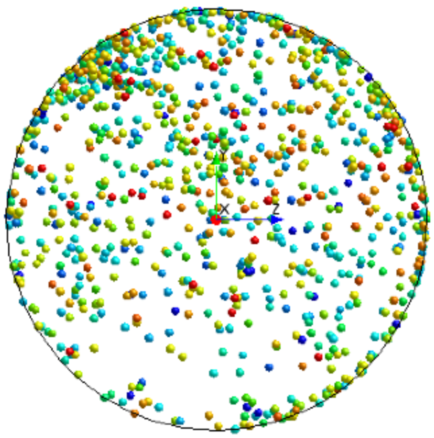
Figure 4.1 shows a plan view of the velocity magnitude at the duct mid-plane (top), and resultant particle distribution at the sampling point (bottom) for the 70 kcfm case running Fans A, C, and D. This case passed with respect to flow angle and velocity uniformity COV, but failed gas and particle tracer uniformity with COVs of over 38 and 30 respectively. Although a shrouded probe located at the duct centerline might detect the presence of particles, the variance of particle distribution does not meet the ANSI/HPS N13.1-2011 standard.

Table 4.1. Summary of Initial CFD Modeling Results for the 3420 Duct with Four Fans

Fans	Flow, kcfm	Inject location	Flow Angle	Velocity COV	Gas Tracer COV	Particle COV
Nominal Flow: Three-fan combinations (three fans operating at 93%, one fan in reserve)						
A, B, C	70	A	3.7	2.6	18.43	18.23
A, B, D	70	A	3.4	2.4	14.42	13.16
A, C, D	70	A	3.8	4.4	38.57	30.14[1]
B, C, D	70	B	3.2	6.7	6.32	19.73
High Flow: Four-fan operation (four fans operating at 80%)						
Fans	Flow, kcfm	Inject location	Flow Angle	Velocity COV	Gas Tracer COV	Particle COV
A, B, C, D	80	A	2.9	4.6	31.30	24.84
A, B, C, D	80	B	2.9	4.6	22.07	33.34
A, B, C, D	80	C	2.9	4.6	20.51	55.68
A, B, C, D	80	D	2.9	4.6	11.73	32.57
A, B, C, D	80	A B C D	2.9	4.6	5.9	18.35
Maximum Flow: Four-fan operation (four fans at operating 100%)						
Fans	Flow, kcfm	Inject location	Flow Angle	Velocity COV	Gas Tracer COV	Particle COV
A, B, C, D	100	A	-	-	-	-
A, B, C, D	100	B	-	-	-	-
A, B, C, D	100	C	-	-	-	-
A, B, C, D	100	D	3.0	4.2	11.37	31.91
Set Back Flow (Single-Fan): One-fan operation (one fan at operating at 66%)						
Fans	Flow, kcfm	Inject location	Flow Angle	Velocity COV	Gas Tracer COV	Particle COV
A	16.5	A	6.3	6.9	2.24	60.70
B	16.5	B	-	-	-	-
C	16.5	C	-	-	-	-
D	16.5	D	-	-	-	-
Set-Back Flow (Two-Fan): Two-fan operation (two fans operating at 66%)						
Fans	Flow, kcfm	Inject location	Flow Angle	Velocity COV	Gas Tracer COV	Particle COV
A, B	33	A	-	-	-	-
A, C	33	A	-	-	-	-
A, D	33	A	-	-	-	-
B, C	33	B	-	-	-	-
B, D	33	B	-	-	-	-
C, D	33	C	-	-	-	-

[1] See also Figure 4.1.





Color scale: BLUE = 0 m/s to RED = 24 m/s

Figure 4.1. Velocity Magnitude at Mid-Duct. Plan view (top) and particle distributions at the sampling point (bottom) for the 70 kcfm case operating Fans A, C, and D. Particle uniformity COV = 30.

4.2 Design Development to Include Stationary Blender

Because of the insufficient mixing and elevated COVs of tracers in the initial flow cases and the subsequent uncertainty that the stack will qualify with the addition of the fourth fan, a stationary air blender from Blender Products, Inc. was considered for inclusion in the stack system.

Preferably, addition of the air blender would not increase the system pressure drop, relative to the initial four-fan design, by more than ~0.25-inch water column for a flow rate of 75 kcfm. This pressure drop benchmark requires that the diameter of the air blender be 110-inch An air blender with a 62-inch diameter to match that of the main duct can be produced, but the pressure drop would either be too large or the mixing insufficient (with a low blade angle to limit the pressure drop).

CFD models of straight ducts including the air blender were created to analyze the mixing performance. We found that the 110-inch blender installed in an equal diameter duct and operated at 75 kcfm has a well-established counter-rotating flow through the separate central and annular portions of the device (see Figure 4.2a), and smaller pressure drop than the 62-inch duct without a blender (at the same flow rate). However, when the blender was added to the duct with short (4-feet long) expansion and contraction regions, the flow was focused mostly through the center of the device so counter-rotating flow was not established and the pressure drop was greater than 1-inch water column.

(a) (b)

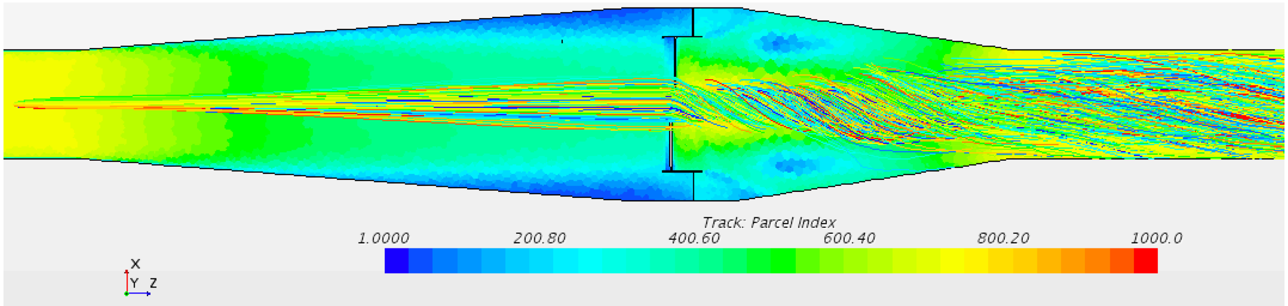
Figure 4.2. Typical Static Air Blender. (a) Photograph of sample device. (b) CFD Model representation.

Additional models were created to test long expansion/contraction regions that transition from the 62-inch duct to the 110-inch air blender, and various lengths of 110-inch straight duct between expansion/contraction, and adjacent to the blender. We found that long, low-angle expansion and contraction regions (27-feet and 14-feet respectively), and short (10-inch long) 110-inch ducts adjacent to the blender helped minimize flow separation along the expansion wall and provided 1) counter-rotating flow through the blender, 2) good mixing of particles, and 3) a low pressure drop of 0.28-inch water column. Figure 4.3 shows the shape of the final CFD-designed expansion/blender/contraction duct region. Figure 4.3 (a) and (c) show the flow velocity magnitude and particle tracks through the mid-plane for a flow rate of 75 kcfm. Figure 4.3 (b) shows the particle distribution at the sampling point due to particle-tracer releases along the main duct centerline, and Figure 4.3 (d) shows the particle distribution at the sampling point due to particle-tracer releases 5 inches from the main duct wall. This portion of the duct with air blender was added to the 3420 Building system model for testing the flow and mixing performance.

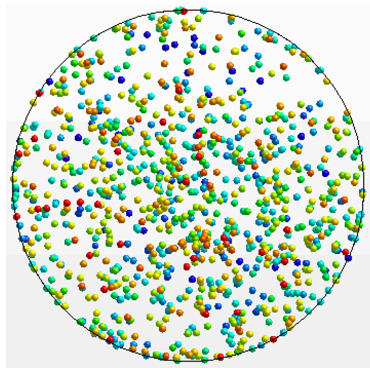
4.3 Modeling the Performance of the 3420 Duct with Blender

Figure 4.4 shows the four-fan system adapted to include the air blender. The fans, sampling point, and stack exit locations are all unchanged. The air blender is incorporated into the system with the 27-ft expansion region located just downstream of Fan A to allow the greatest mixing distance from the blender to the sampling point.

A test simulation of the modified four-fan design shown in Figure 4.4, operating at the 80-kcfm High Flow condition, was performed to check the system mixing performance and sampling point efficiency. Figure 4.5 shows the mid-duct velocity magnitude profile in the plan view, and the velocity distribution looking downstream to the sampling point. Flows from each fan entering the main duct at an angle creates a swirling flow upstream of the air blender. The swirl is noticeable entering the expansion. The air blender establishes a counter-rotating flow. Once through the contraction, most mixing is complete, and the flow travels towards the sampling point with minimal swirl. For this test case, the maximum flow angle at the sampling point is 6.8°, and the COV of velocity uniformity is 2.1. Data collection locations at the sampling point are shown as red dots.

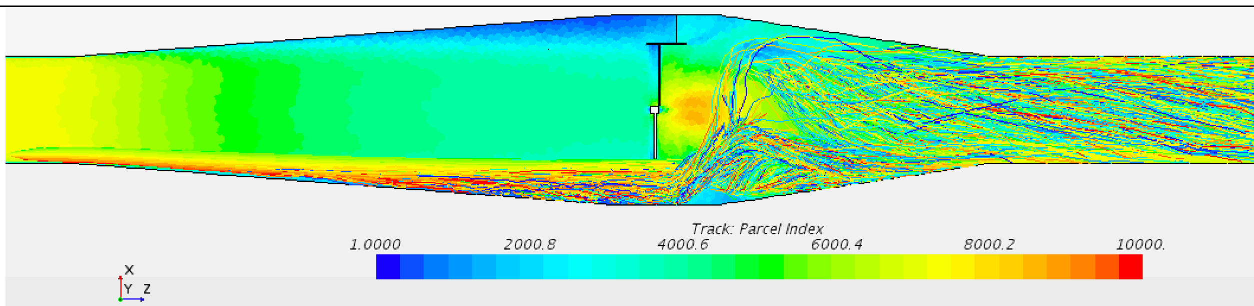


(a)

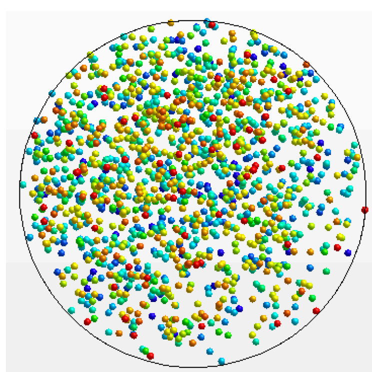


(b)

Velocity magnitude at the mid-plane of the duct due to 75 kcfm flow with particle tracks (a) and particle distribution at the sampling point (b) for tracers released at the duct centerline.



(c)



(d)

Velocity magnitude at the mid-plane of the duct due to 75 kcfm flow with particle tracks (c) and particle distribution at the sampling point (d) for tracers released 5 inch from the main duct wall.

Figure 4.3. Design of Air Blender in Duct

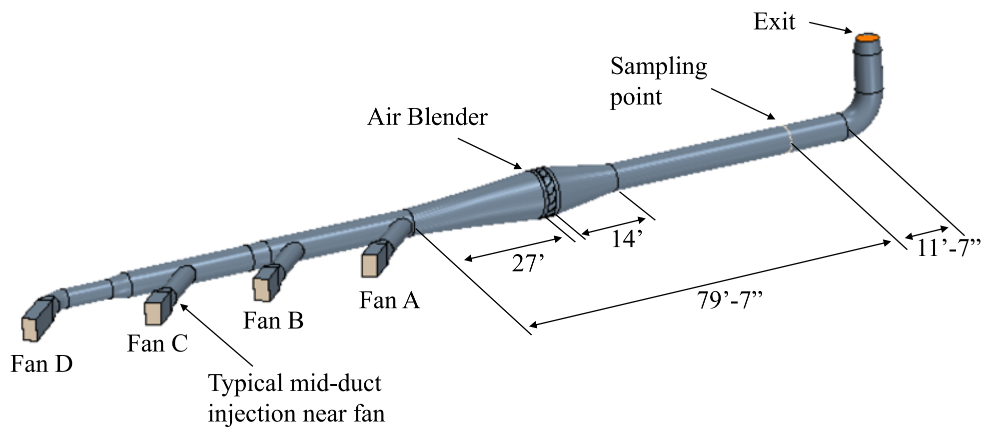


Figure 4.4. 3420 Building Exhaust System with Static Air Blender.

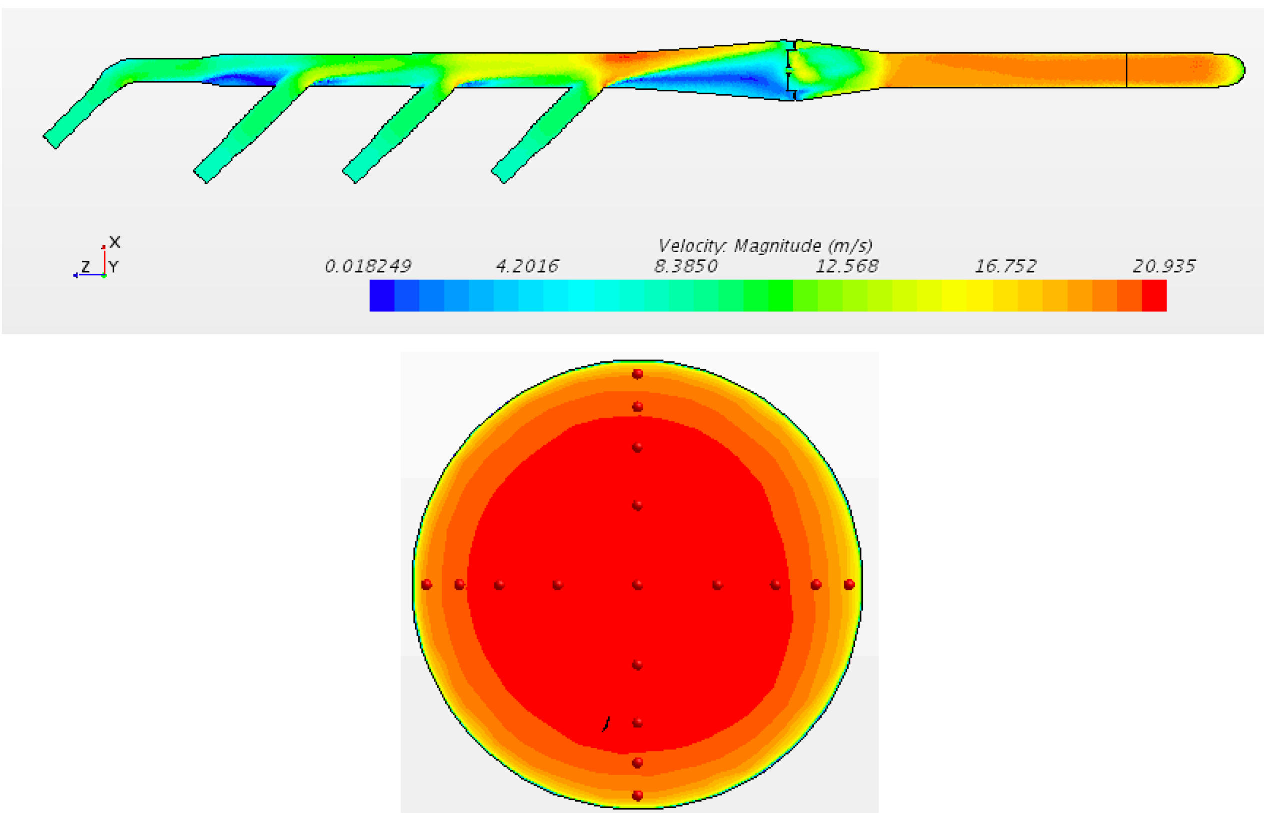


Figure 4.5. Contours of Velocity Magnitude in the Duct with the Air Blender Installed in Plan View (top), and at the Sampling Point (bottom), with all Four Fans Operating in the High-Flow Condition with a Flow Rate of 80 kcfm

For the test case, N_2O tracer gas was injected near Fan A. Figure 4.6 shows the mid-duct N_2O mass fraction in the plan view, and looking downstream at the sampling point. Mixing appears to be mostly complete within a few diameters downstream of the contraction. The gas tracer COV for this case is 2.23.

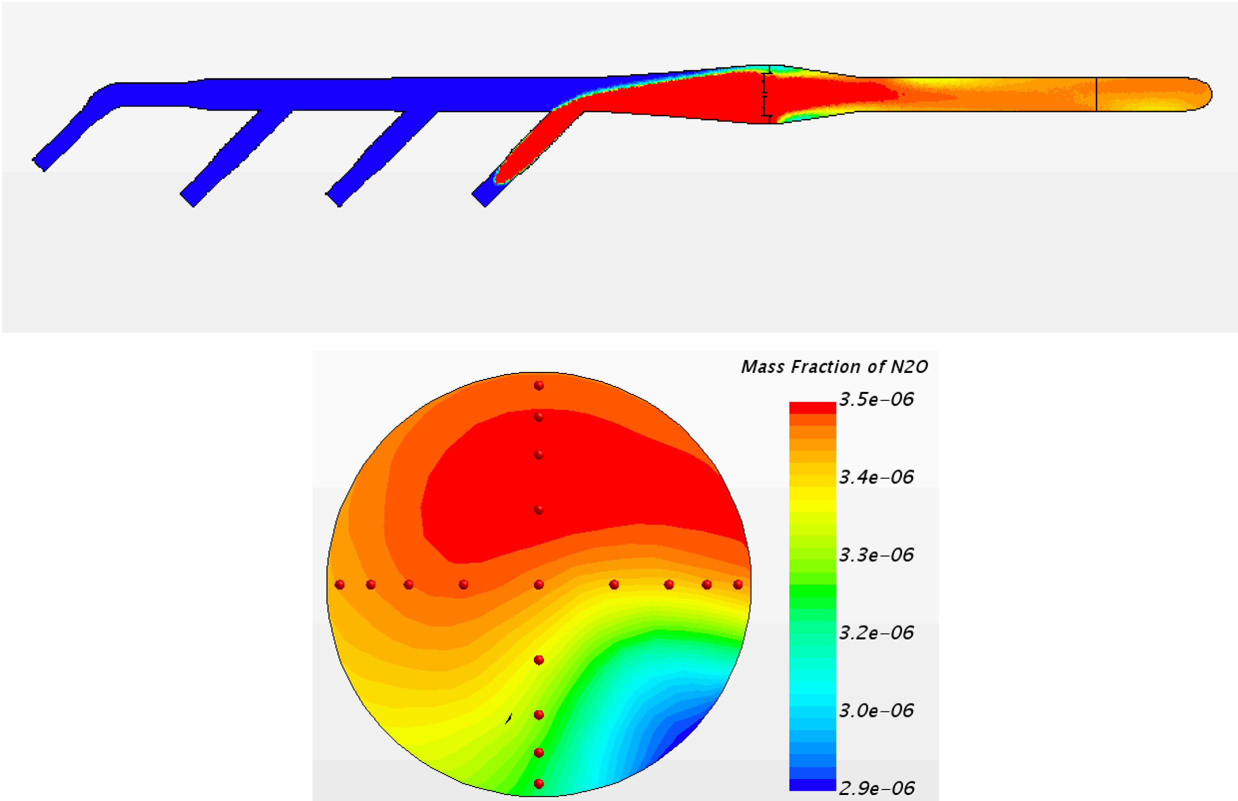


Figure 4.6. Contours of N_2O mass fraction in the Duct with the Air Blender Installed, in the Plan View (top), and at the Sampling Point (bottom), with all Four Fans Operating in the High-Flow Condition with a Flow Rate of 80 kcfm

For this 80 kcfm, 4-fan case, 10-micron aerodynamic diameter tracer particles are released near Fan A. The swirling flow entering the expansion distributes the particles across the duct, and the counter-rotating inner and outer flows through the blender serve to mix the flow within a few duct diameters downstream of the contraction. Particle-tracer paths from the Lagrangian solution are shown in Figure 4.7 in plan view and at the sampling point. The COV for this particle distribution is 16.2. This figure illustrates the quality of distribution required such that particle uniformity COV is less than 20.

The full array of Standard Flow condition cases (introduced in 4.1) were run using the updated four-fan system with the air blender. The results of these runs are summarized in Table 4.2 which shows maximum average flow angles ranging from 0.8 to 6.9°, well within the standard limit of 20°. The velocity uniformity COV values range from 1.58 to 4.12. Gas tracer COV values range from 0.11 to 3.45, and particle tracer COV values range from 11.4 to 19.8. All resulting COV values were below the limit of 20. And in no case was the maximum gas tracer concentration 30% greater than the mean. Thus, the modeling results predict that flow angle, velocity, gas tracer, and particle tracer criteria established by the ANSI/HPS N13.1-2011 standard will be met with the addition of the air blender installed using the expansion/blender/contraction design.

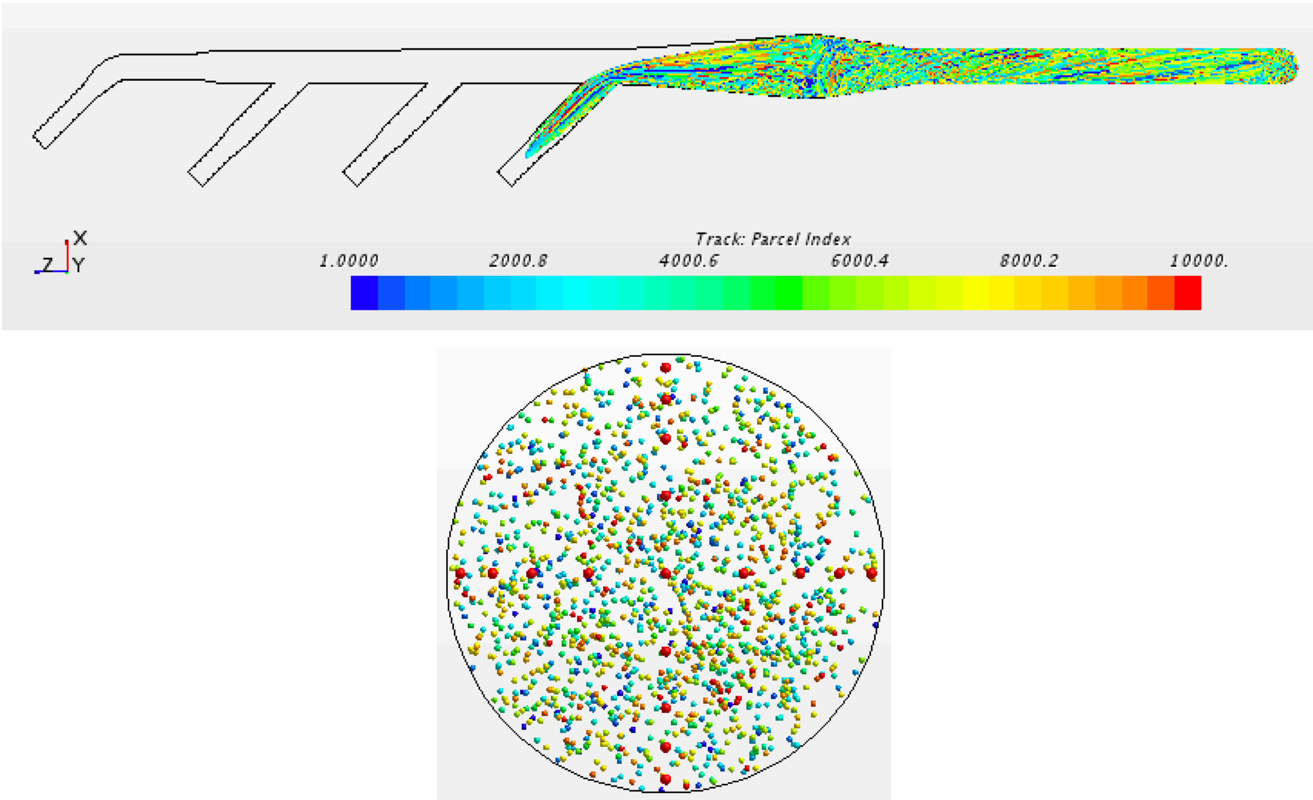


Figure 4.7. Tracer Particle Paths through the Air Blender with All Four Fans Operating in the High-Flow Condition with a Flow Rate of 80 kcfm. Plan View (top) and at the Sampling Point (bottom).

Table 4.2. Summary of CFD Modeling Results for the 3420 Building Duct with an Air Blender

Nominal Operation: Three-fan combinations (three fans operating at 93%, one fan in reserve)						
Fans	Flow, kcfm	Inject location	Flow Angle	Velocity COV	Gas Tracer COV	Particle COV
A, B, C	70	A	5.3	1.62	2.71	18.4
A, B, D	70	A	5.1	1.59	2.68	19.4
A, C, D	70	A	5.2	1.61	3.45	19.3
B, C, D	70	B	1.8	1.58	1.17	13.8
Nominal High: Four-fan operation (four fans operating at 80%)						
Fans	Flow, kcfm	Inject location	Flow Angle	Velocity COV	Gas Tracer COV	Particle COV
A, B, C, D	80	A	6.8	2.08	2.23	16.2
A, B, C, D	80	B	"	"	3.22	17.5
A, B, C, D	80	C	"	"	0.94	18.9
A, B, C, D	80	D	"	"	2.62	18.0

Maximum Operation: Four-fan operation (four fans operating at 100%)						
Fans	Flow, kcfm	Inject location	Flow Angle	Velocity COV	Gas Tracer COV	Particle COV
A, B, C, D	100	A	6.9	1.97	2.11	17.7
A, B, C, D	100	B	“	“	3.04	15.6
A, B, C, D	100	C	“	“	0.91	19.0
A, B, C, D	100	D	“	“	2.57	19.8
Set Back Flow (single fan): One-fan operation (one fan operating at 66%)						
Fans	Flow, kcfm	Inject location	Flow Angle	Velocity COV	Gas Tracer COV	Particle COV
A	16.5	A	6.6	3.03	0.23	17.4
B	16.5	B	2.7	1.90	0.11	19.0
C	16.5	C	1.3	2.69	0.30	18.8
D	16.5	D	0.8	4.12	0.47	15.9
Set Back Flow (two fans): Two-fan operation (two fans operating at 66%)						
Fans	Flow, kcfm	Inject location	Flow Angle	Velocity COV	Gas Tracer COV	Particle COV
A, B	33	A	1.0	1.99	1.70	12.4
A, C	33	A	1.9	1.64	1.91	17.9
A, D	33	A	1.7	1.65	1.02	14.7
B, C	33	B	2.6	1.67	1.28	19.4
B, D	33	B	0.8	1.81	0.26	11.4
C, D	33	C	1.0	2.46	1.21	19.7

5.0 Conclusions

Based on CFD modeling of the 3420 Building filtered exhaust stack system, the following conclusions are drawn:

- CFD model benchmarking showed the modeling methodology provides flow angle, velocity uniformity COV, gas tracer uniformity COV, and particle-tracer uniformity COV values that are in good agreement with those derived from testing of the original stack configuration with three fans and tests of the HV-C2 physical test model used to help qualify the 3420 Building stack.
- Modeling results for expected nominal, high, maximum, and set-back flows in the initial four-fan system design predict that flow angles and velocity uniformity COV values should remain well within compliance of the ANSI/HPS N13.1-2011 standard.
- Modeling results for expected nominal, high, maximum, and set-back flows in the initial four-fan system design predict that gas tracer and particle-tracer uniformity COV values may not remain within compliance, suggesting the addition of an air blender.
- CFD modeling produced a duct design that allowed an oversized (large diameter) air blender to be installed in the 3420 Building exhaust stack system while providing for the necessary counter-rotating inner and outer flows, increased mixing, and low pressure drop.
- Modeling results of the four-fan duct including an air blender operating at all expected flow conditions predict that flow angle, velocity uniformity, and tracer concentration criteria established by the ANSI/HPS N13.1-2011 standard will be met with the addition of the air blender installed using the expansion/blender/contraction design.

6.0 References

Incropera FP and DP DeWitt. 1985. *Introduction to Heat Transfer*. Second edition, John Wiley and Sons, New York.

ANSI/HPS N13.1-1999. 1999. *Sampling and Monitoring Releases of Airborne Radioactive Substances from the Stack and Ducts of Nuclear Facilities*. American National Standards Institute/Health Physics Society, McLean, Virginia.

Barnett JM, MY Ballinger, KP Recknagle, and ST Yokuda. 2005. *Computational Modeling of a Stack Sampling Location for Radioactive Air Emissions*. Health Physics 89(1):S39-S40.

Jensen BBB. 2007. *Numerical Study of Influence of Inlet Turbulence Parameters on Turbulence Intensity in the Flow Domain: Incompressible Flow in Pipe System*. Proceedings of the Institution of Mechanical Engineers, Part E: Journal of Process Mechanical Engineering 221(4):177-186.

Glissmeyer JA and JG Droppo. 2007. *Assessment of the HV-C2 Stack Sampling Probe Location*. PNNL-16611, Pacific Northwest National Laboratory, Richland, Washington.

Recknagle KP, ST Yokuda, MY Ballinger, and JM Barnett. 2009. *Scaled Tests and Modeling of Effluent Stack Sampling Location Mixing*. Health Physics 96(2):164-174.

Glissmeyer JA and JE Flaherty. 2010. *Assessment of the 3420 Building Filtered Exhaust Stack Sampling Probe Location*. PNNL-19563, Pacific Northwest National Laboratory, Richland, Washington.

ANSI/HPS N13.1-2011. 2011. *Sampling and Monitoring Releases of Airborne Radioactive Substances from the Stack and Ducts of Nuclear Facilities*. American National Standards Institute/Health Physics Society, McLean, Virginia.

Ballinger MY, JA Glissmeyer, JM Barnett, KP Recknagle, and ST Yokuda. 2011. *Sampling Point Compliance Tests for 325 Building at Set-Back Flow Conditions*. PNNL-20397. Richland, WA: Pacific Northwest National Laboratory.

Recknagle KP, JM Barnett, and SR Suffield. 2013. *Modeling the Air Flow in the 3410 Building Filtered Exhaust Stack System*. PNNL-22185. Richland, WA: Pacific Northwest National Laboratory.

Yu X-Y, KP Recknagle, JA Glissmeyer, and JM Barnett. 2014. *Integrating Modeling and Physical Testing for Assessing Filtered Exhaust Stack Sampling Probe Location*. Health Physics 107(1):S30.

Barnett, JM, X-Y Yu, KP Recknagle, and JA Glissmeyer. 2016. *Modeling and Qualification of a Modified Emission Unit for Radioactive Air Emissions Stack Modeling Compliance*. Health Physics 111(5):432-441. DOI: 10.1097/HP.0000000000000557.

Recknagle KP, JM Barnett, X-Y Yu, SR Suffield. 2018. *Modeling Filtered Building Effluent Stack Sampling Points for Qualification Criteria*. 1st International Symposium on Mechanics. Scotland, U.K.

Siemens PLM Software. 2017. STAR-CCM+12.06.011 *User Guide*. Siemens PLM Software, Plano,TX.

Pacific Northwest
NATIONAL LABORATORY

www.pnnl.gov

902 Battelle Boulevard
P.O. Box 999
Richland, WA 99352
1-888-375-PNNL (7665)

U.S. DEPARTMENT OF
ENERGY