

PNNL-32393

Qualification of ANSI/HPS N13.1- 2011 Mixing Criteria by Computational Fluid Dynamics Modeling for the 3430 Building Fan Addition and Increased Ventilation Capacity

January 2022

SR Suffield
JM Barnett
SE Gourley

DISCLAIMER

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

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
www.osti.gov
ph: (865) 576-8401
fox: (865) 576-5728
email: reports@osti.gov

Available to the public from the National Technical Information Service
5301 Shawnee Rd., Alexandria, VA 22312
ph: (800) 553-NTIS (6847)
or (703) 605-6000
email: info@ntis.gov
Online ordering: <http://www.ntis.gov>

Qualification of ANSI/HPS N13.1-2011 Mixing Criteria by Computational Fluid Dynamics Modeling for the 3430 Building Fan Addition and Increased Ventilation Capacity

January 2022

SR Suffield
JM Barnett
SE Gourley

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

Pacific Northwest National Laboratory
Richland, Washington 99354

Summary

Additional ventilation capacity has been designed for the 3430 Building filtered exhaust stack system. The updated system will increase the number of fans from two to three and include new ductwork with a larger diameter 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 two-fan stack previously was found to comply with the American National Standards Institute/Health Physics Society (ANSI/HPS) N13.1-1999 standard. Full-scale, three-dimensional computational fluid dynamics (CFD) modeling was used to evaluate the modified three-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 two-fan system showed good agreement with test data collected from the 3430 Building stack. Modeling was performed to develop a suitable three-fan design. Initial modeling of the three-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.

Acronyms and Abbreviations

acfm	actual cubic feet per minute
3-D	three-dimensional
ANSI	American National Standards Institute
cfm	cubic feet per minute
HPS	Health Physics Society
HVAC	heating, ventilation, and air conditioning
CFD	computational fluid dynamics
COV	coefficient of variation
PNNL	Pacific Northwest National Laboratory
scfm	standard cubic feet per minute

Contents

Summary	ii
Acronyms and Abbreviations.....	iii
1.0 Introduction	1
2.0 Modeling Methodology	3
2.1 Flow Model	4
2.2 Gas Tracer Model	4
2.3 Particle Tracer Model.....	5
2.4 Model Geometry and Computational Mesh	5
2.5 Boundary Conditions.....	6
3.0 Stack Model Benchmarking	7
4.0 Stack Modeling Results	10
4.1 Modeling the Performance of the Initial Three-Fan System Design.....	10
4.2 Design Development to Include Air Blender	12
4.3 Modeling the Performance of the 3430 Building Duct with the Air Blender	13
5.0 Conclusions.....	16
6.0 References	17

Figures

Figure 1.	Model Geometry for the Initial Three-Fan 3430 Building Stack System Design	5
Figure 2.	Detail of Computational Mesh at the (a) Surface Near Fan 2, and (b) Typical Cross-Section of the Volume Mesh in the Main Duct	6
Figure 3.	HV-C2 Physical Test Model	7
Figure 4.	3430 Existing Two-Fan System – Dimensional CFD Model	8
Figure 5.	Velocity Magnitude at Mid-Duct. Plan view (top) and particle and tracer gas distributions at the sampling point (bottom) for the 114,000-cfm case operating Fans 1, 2, and 3 with the injection point at the junction.....	12
Figure 6.	Static Air Blender CAD Geometry.	13
Figure 7.	3430 Building Exhaust System with Static Air Blender	13
Figure 8.	Velocity Magnitude at Mid-Duct. Plan view (top) and particle and tracer gas distributions at the sampling point (bottom) for the 114,000-cfm case operating Fans 1, 2, and 3 with the injection point at the junction and the inclusion of an air blender in the duct system.....	15

Tables

Table 1.	Velocity Uniformity Comparison of CFD Modeled Results with Data from 3430 Tests	9
Table 2.	Flow Angle Comparison of CFD Modeled Results with Data from 3430 Tests.....	9
Table 3.	Gas-Tracer Comparison of CFD Modeled Results with Data from HV-C2 Tests.....	9
Table 4.	Particle-Tracer Comparison of CFD Modeled Results with Data from HV-C2 Tests	9
Table 5.	Summary of Initial CFD Modeling Results for the 3430 Building Duct with Three Fans	11
Table 6.	Summary of CFD Modeling Results for the 3430 Building Duct with an Air Blender	14

1.0 Introduction

The 3430 Building at Pacific Northwest National Laboratory (PNNL) houses radiological capabilities. Permit conditions require that 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 American National Standards Institute/Health Physics Society (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. Uniformity is expressed by the coefficient of variation (COV), which is defined as the standard deviation divided by the mean, reported as a percentage. 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, and COV values 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 two-fan 3430 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 3430 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 3430 Building sampling location. Thus, it is reasonable to expect that all of the main duct length of the 3430 Building exhaust system will be needed to provide sufficient mixing of tracer gas and tracer particles.

The 3430 Building exhaust stack system will be updated with additional ventilation capacity. The updated system will incorporate a third fan and associated ductwork to integrate the new fan into the stack. The existing stack will be demolished, and a new larger diameter stack that considers both the upstream and downstream number of duct diameters in relation to the sampling location will be constructed. As a result, the stack configuration will be changed substantially. The nominal operating condition will have two fans operating with one fan in standby. The average overall flow rate has been 35,500 cfm for the past 5 years and as high as about 37,500 cfm for a single year (Barnett and Snyder 2021). The new anticipated flow rate is expected to increase significantly to 76,000 cfm. 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. In the new ANSI/HPS N13.1-2021, computational fluid dynamics (CFD) modeling is discussed as an option for optimizing and upgrading an existing system and indicates the same requirements for qualifying the sample extraction system must be met (i.e., those methods similar to either a similar exhaust system or a scale model system [HPS 2021]). The final 3430-Building-stack modeled design with three fans includes an oversize air blender to improve gas and particle mixing. The modeled design effectively acts as a similar stack design.

2.0 Modeling Methodology

The purpose of modeling the 3430 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 CFD flow simulation code, STAR-CCM+ (Siemens 2021) was selected for creation of the 3-D model domain and the flow simulations.

PNNL has extensive experience in modeling stack designs for compliance. Past 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, California (Ballinger, Recknagle, and Barnett 2003). Peer-reviewed publications authored by PNNL staff include papers that addressed the following relevant topics:

- 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),
- Modeling of a modified building stack for sampling compliance (Barnett et. al 2016), and
- Modeling of filtered building effluent stack sampling points for qualification criteria (Barnett et. al 2020).

Relevant internal reports include those that address:

- 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), and
- Compliance modeling of the 3420 Building with the addition of a fourth fan (Recknagle et. al 2018a, Flaherty et. al 2020).

In addition, a presentation on modeling building stack sampling points for qualification criteria (Recknagle et. al 2018b) was presented by S.R. Suffield at the 1st International Symposium on Mechanics, Scotland, United Kingdom in July 2018. The present modeling for the 3430 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:

$$\frac{\partial}{\partial x_j}(\rho u_j) = 0 \quad (1)$$

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

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

$$\tau_{ij} = 2\mu\sigma_{ij} - \frac{2}{3}\mu\frac{\partial u_k}{\partial x_k}\delta_{ij} - \overline{\rho u'_i u'_j} \quad (3)$$

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 standard 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 2021). 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). To capture strong secondary flows, which are frequently seen in heating, ventilation, and air conditioning (HVAC) systems, non-linear terms were added to the stress-strain relationship for the κ - ϵ model by selecting a cubic constitutive relationship. This modified the Boussinesq approximation with cubic terms (Siemens 2021).

These equations (Eqs. 1, 2, and 3) are independent of units. That is, the user can select the units for length, density, and velocity and make any necessary conversions to ensure consistent units.

2.2 Gas Tracer Model

For the 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:

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

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 depends 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:

$$m_d \frac{du_d}{dt} = F_{dr} + F_p + F_b \quad (5)$$

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 are 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 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 three-fan, 3430 Building system design is shown in Figure 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. The geometry also includes the dampers just downstream of the fans. The dampers are in an open configuration. Typical tracer injection locations are mid-duct, just downstream of the fans, and near the junction where all three fan streams converge. The sampling point is located 72.5 ft downstream of Fan 2, or 15 diameters for the duct with a 58-inch diameter.

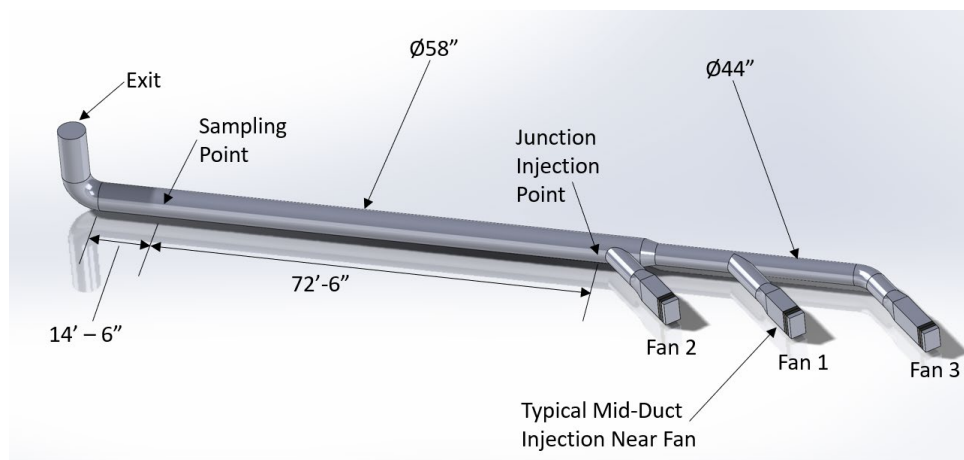


Figure 1. Model Geometry for the Initial Three-Fan 3430 Building Stack System Design

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 2 million elements. Figure 2a provides a view of the mesh near Fan 2. The typical resolution throughout the volume mesh is represented in Figure 2b.

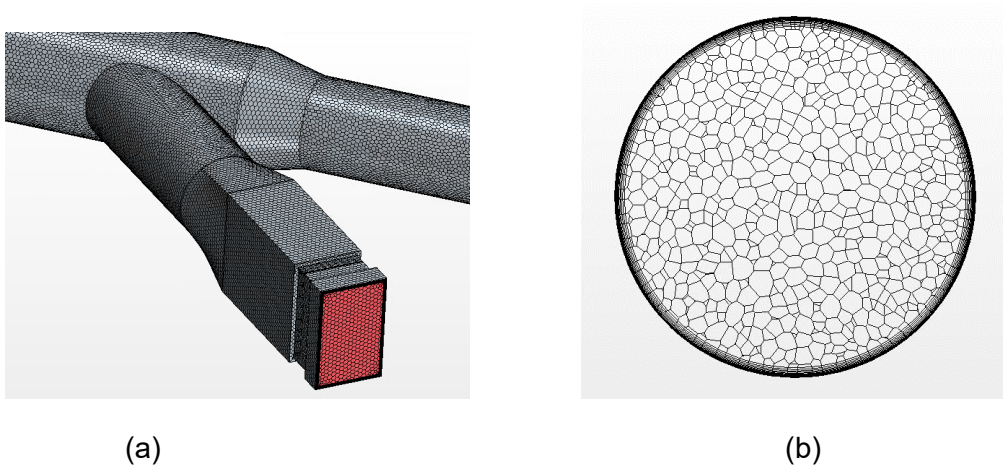


Figure 2. Detail of Computational Mesh at the (a) Surface Near Fan 2, and (b) Typical Cross-Section of the Volume Mesh in the Main Duct

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 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 3430 Building two-fan stack was tested for flow angle and velocity uniformity (Glissmeyer and Flaherty 2010). Results for tracer gas and tracer particle sampling efficiency were inferred from data collected during previous tests (Glissmeyer 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 were 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 3430 Building stack.

A 3-D CFD model of the existing 3430 Building two-fan stack was created and set up to replicate the geometry and flow conditions tested. The 3430 Building two-fan system is a similar configuration to the HV-C2 test model, except that the fans enter the main duct at a 90° angle to the outlet in the 3430 Building system instead of a 180° angle to the outlet in the HV-C2 test system. Figure 3 is a photograph of the assembled HV-C2 physical test 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 4 shows the geometry and computational domain of the associated 3-D CFD model of the existing 3430 two-fan system. 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. HV-C2 Physical Test Model

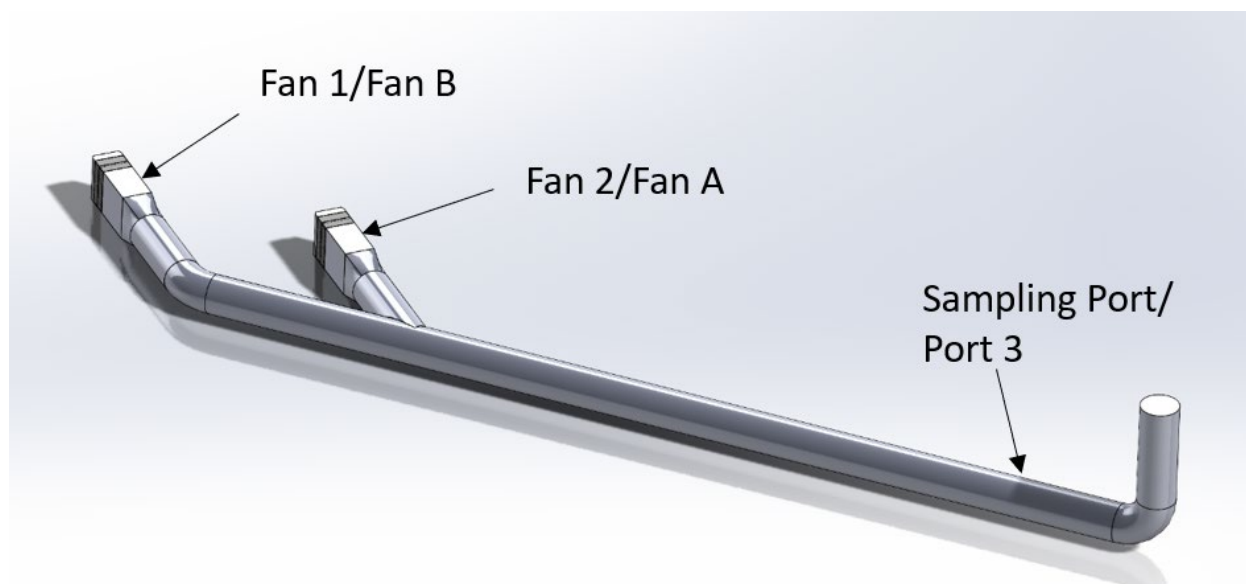


Figure 4. 3430 Building Stack Existing Two-Fan System – Dimensional CFD Model

Stack testing of the existing 3430 Building two-fan system included velocity and flow angle measurements, which were used to benchmark the CFD model. Stack testing of the HV-C2 system 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 sampling location in the 3430 Building stack scales to match the testing results of the HV-C2 system most closely at Test Port 3. As such, gas and particle tracer data collected there were of interest when checking the results obtained from model benchmarking runs.

Table 1 through Table 4 summarize the CFD modeling benchmark/testing data comparisons. Table 1 compares velocity uniformity measured in testing of the existing 3430 Building stack and predicted by the model. The flow angle comparisons with the measured testing of the 3430 Building system are shown in Table 2. Gas-tracer uniformity and particle-tracer uniformity comparisons from testing of the HV-C2 test system are shown in Table 3 and Table 4, respectively. The CFD model gave consistently conservative particle uniformity results compared to the measured data. With the exception of one of the single fan cases, the CFD model tended to give conservative tracer uniformity results compared to the measured data. The CFD model also gave higher COV values for velocity and flow angle for the one operating fan configuration. With two fans in operation, the results are very similar between the CFD predictions and measured results for velocity and flow angle. Overall, both the measured data and CFD predictions values are below the COV and variation limits specified in the ANSI/HPS N13.1-2011 standard.

This benchmarking exercise demonstrates 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 3430 Building.

Table 1. Velocity Uniformity Comparison of CFD Modeled Results with Data from 3430 Tests

Table 4.2 from PNNL-19262, Rev. 1 (Glissmeyer and Flaherty 2010)				CFD Model
Fan Operating Configuration	Run Nos.	Measured Airflow (acfm)	% COV	% COV
Maximum (2 Fans, Sashes Open)	VT-4	34,252	3.8	3.3
Maximum (2 Fans, Sashes Open)	VT-5	34,352	3.8	
Minimum (2 Fans, Sashes Closed)	VT-6	24,634	3.2	3.2
Minimum (1 Fan, Sashes Closed)	VT-7	24,613	2.6	6.4

acfm = actual cubic feet per minute

Table 2. Flow Angle Comparison of CFD Modeled Results with Data from 3430 Tests

Table 4.3 from PNNL-19262, Rev. 1 (Glissmeyer and Flaherty 2010)				CFD Model Mean Absolute Flow Angle (Degrees)
Fan Operating Configuration	Run Nos.	Measured Airflow (scfm)	Mean Absolute Flow Angle (Degrees)	(Degrees)
Maximum (2 Fans, Sashes Open)	FA-4	31,650	1.8	1.8
Maximum (2 Fans, Sashes Open)	FA-5	31,250	2.6	
Minimum (2 Fans, Sashes Closed)	FA-6	23,150	2.9	2.2
Minimum (1 Fans, Sashes Closed)	FA-7	23,300	3.0	9.6

scfm = standard cubic feet per minute

Table 3. Gas-Tracer Comparison of CFD Modeled Results with Data from HV-C2 Tests

Table B.1 from PNNL-19262, Rev. 1 (Glissmeyer and Flaherty 2010)						CFD Model	
Injection Port	Operating Fans	Test Port	Run No.	% COV	% Deviation from Mean	% COV	% Deviation from Mean
A Center	A	3	GT-37	2.3	5.3	1.7	3.3
A Center	A&B	3	GT-34	3.2	7.9	8.8	18.6
B Center	B	3	GT-47	1.7	2.9	2.2	5.2
B Center	A&B	3	GT-54	3.9	9.1	6.3	15.8

Table 4. Particle-Tracer Comparison of CFD Modeled Results with Data from HV-C2 Tests

Table B.2 from PNNL-19262, Rev. 1 (Glissmeyer and Flaherty 2010)					CFD Model
Injection Port	Operating Fans	Test Port	Run No.	% COV	% COV
A	A&B	3	PT-20	8.1	12.9
A	A	3	PT-14	3.7	18.7
B	B	3	PT-19	3.6	12.0

4.0 Stack Modeling Results

In this section, we discuss results from CFD simulations of the three-fan exhaust system. The simulations were undertaken to examine the mixing performance of the system when operating at design conditions. The simulation cases include one, two, and three-fan operations. Section 4.1 presents findings of the performance of the initial three-fan system design. Section 4.2 presents the work done to incorporate a 104-inch stationary air blender into the duct. 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 Three-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 airflow conditions for each fan are described as follows:

- *Minimum Airflow per Fan:* The minimum flow rate per fan is 11,400 cfm.
- *Maximum Airflow per Fan:* The maximum flow rate per fan is 38,000 cfm.

The minimum and maximum airflow conditions per fan results in a minimum airflow of 11,400 cfm for a single fan in operation at the minimum airflow rate per fan and a maximum airflow of 114,000 cfm for all three fans in operation at the maximum airflow per fan.

Simulation results for the initial three-fan system are mixed. In all cases, the flow angle and velocity uniformity criteria are 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 5. The results that failed to meet the standard mixing criteria are shown in red. Most of the failed cases occurred when the source term injection point was at the junction where the three-fan air streams converge (“Junction” in Table 5). For all three fans in operation, the injection at Fan 2 also failed at both the minimum and maximum flow rates.

Figure 5 shows a plan view (no air blender) of the velocity magnitude at the duct mid-plane (top), and resultant particle and tracer gas distribution at the sampling point (bottom) for the 114,000-cfm case running Fans 1, 2, and 3 with the injection point at the junction. This case passed with respect to flow angle and velocity uniformity COV but failed for gas and particle tracer uniformity with COVs over limits at both the junction and Fan 2 injection points.

Table 5. Summary of Initial CFD Modeling Results for the 3430 Building Duct with Three Fans

Operating Fans	Injection Points	Airflow per Fan [cfm]	Total Airflow [cfm]	Velocity Uniformity	Mean Absolute Flow Angle	Gas Tracer Uniformity		Particle Tracer Uniformity
				% COV	Degrees	% COV	% Deviation from Mean	% COV
Fan1	Fan1	11,400	11,400	3.91	16.36	1.74	3.34	6.32
Fan1	Junction	11,400	11,400	3.76	16.20	13.76	32.57	15.68
Fan2	Fan2	11,400	11,400	3.46	17.60	1.76	4.22	18.00
Fan2	Junction	11,400	11,400	3.69	17.58	1.66	4.78	12.50
Fan3	Fan3	11,400	11,400	1.69	12.26	2.30	5.27	12.39
Fan3	Junction	11,400	11,400	1.67	12.25	15.36	26.89	19.85
Fan1	Fan1	38,000	38,000	4.67	17.61	1.59	5.03	16.39
Fan1	Junction	38,000	38,000	4.82	17.81	10.93	35.87	11.13
Fan2	Fan2	38,000	38,000	4.26	19.20	0.29	0.54	18.09
Fan2	Junction	38,000	38,000	4.73	18.66	3.33	9.58	12.36
Fan3	Fan3	38,000	38,000	2.29	13.26	1.99	3.77	19.36
Fan3	Junction	38,000	38,000	2.26	13.25	14.05	27.70	15.37
Fan1,2,&3	Fan1,2,&3	11,400	34,200	3.01	9.18	7.84	12.12	19.68
Fan1,2,&3	Fan1,2,&3	38,000	114,000	2.40	9.77	10.86	22.63	16.98
Fan1,2,&3	Junction	11,400	34,200	3.00	9.20	24.29	34.57	53.36
Fan1,2,&3	Junction	38,000	114,000	2.52	9.71	29.16	60.62	72.97
Fan1,2,&3	Fan2	11,400	34,200	3.00	9.21	15.37	21.75	26.78
Fan1,2,&3	Fan2	38,000	114,000	2.41	9.75	45.93	76.62	43.66
Fan1&2	Fan1&2	11,400	22,800	2.79	12.70	0.83	3.13	18.48
Fan1&2	Fan2	11,400	22,800	2.79	12.72	2.79	4.64	14.98
Fan1&2	Junction	11,400	22,800	2.77	12.71	8.27	11.21	12.58
Fan1&2	Fan1&2	38,000	76,000	3.26	13.61	0.53	1.64	17.22
Fan1&2	Fan2	38,000	76,000	3.23	13.53	5.40	10.79	18.00
Fan1&2	Junction	38,000	76,000	3.35	13.67	5.03	10.54	6.74
Fan2&3	Fan2&3	11,400	22,800	3.59	10.44	2.92	3.84	12.29
Fan2&3	Fan2	11,400	22,800	3.44	10.51	8.75	12.29	16.84
Fan2&3	Junction	11,400	22,800	3.46	10.57	22.34	56.33	24.83
Fan2&3	Fan2&3	38,000	76,000	3.21	11.49	4.23	6.97	15.73
Fan2&3	Fan2	38,000	76,000	3.19	11.48	11.48	26.22	16.93
Fan2&3	Junction	38,000	76,000	3.19	11.48	21.22	47.38	36.16
Fan1&3	Fan1&3	11,400	22,800	3.99	8.22	3.43	4.79	15.29
Fan1&3	Fan1	11,400	22,800	4.04	8.21	8.75	13.21	10.63
Fan1&3	Junction	11,400	22,800	3.95	8.25	24.62	60.85	23.41
Fan1&3	Fan1&3	38,000	76,000	3.15	9.32	2.82	6.23	14.61
Fan1&3	Fan1	38,000	76,000	3.38	9.18	7.72	14.82	19.49
Fan1&3	Junction	38,000	76,000	3.23	9.36	20.00	48.92	31.44

Junction: The location where the airstreams of the three fans converge.

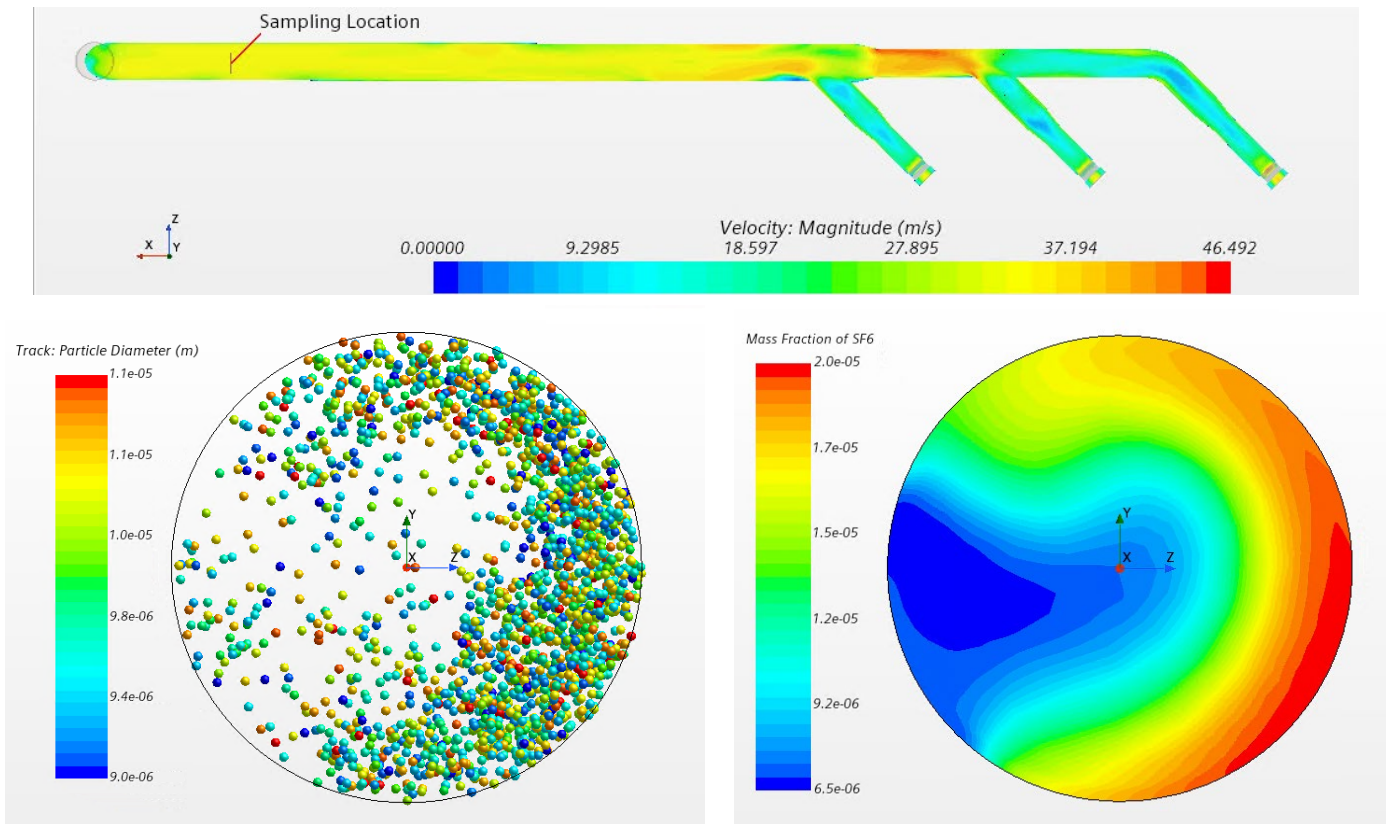


Figure 5. Velocity Magnitude at Mid-Duct. Plan view (top) and particle and tracer gas distributions at the sampling point (bottom) for the 114,000-cfm case operating Fans 1, 2, and 3 with the injection point at the junction.

4.2 Design Development to Include Air Blender

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 third fan resulted in the conclusion to add a stationary air blender in the stack system. A 104-inch diameter air blender from Blender Products, Inc. was integrated into the system. The design of the air blender into the duct system, including the length of the upstream and downstream reducers, is based on previous work reported in Recknagle et. al (2018b) that would minimize the system pressure drop due to the addition of the air blender. The length of the upstream and downstream reducers was 23 feet and 14 feet, respectively. Figure 6 shows the CAD geometry for the air blender.

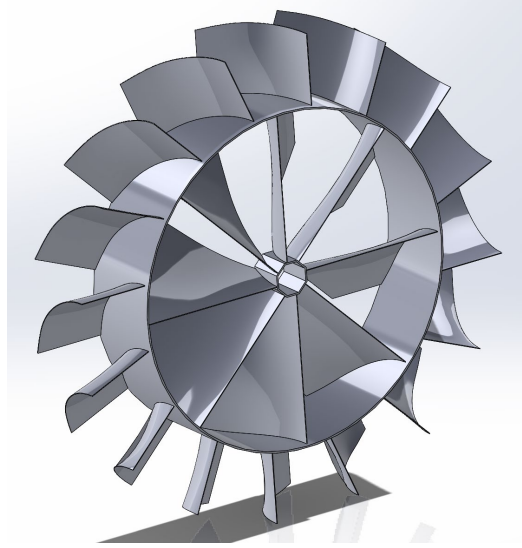


Figure 6. Static Air Blender CAD Geometry.

4.3 Modeling the Performance of the 3430 Building Duct with the Air Blender

The three-fan system was adapted to include the 104-inch air blender and 46-inch stack exit cone. The fans and sampling point locations are unchanged. The air blender is incorporated into the system with the 23-foot expansion region located just downstream of Fan 1 to allow the greatest mixing distance from the blender to the sampling point. Figure 7 shows the air blender integrated into the 3430 Building model geometry.

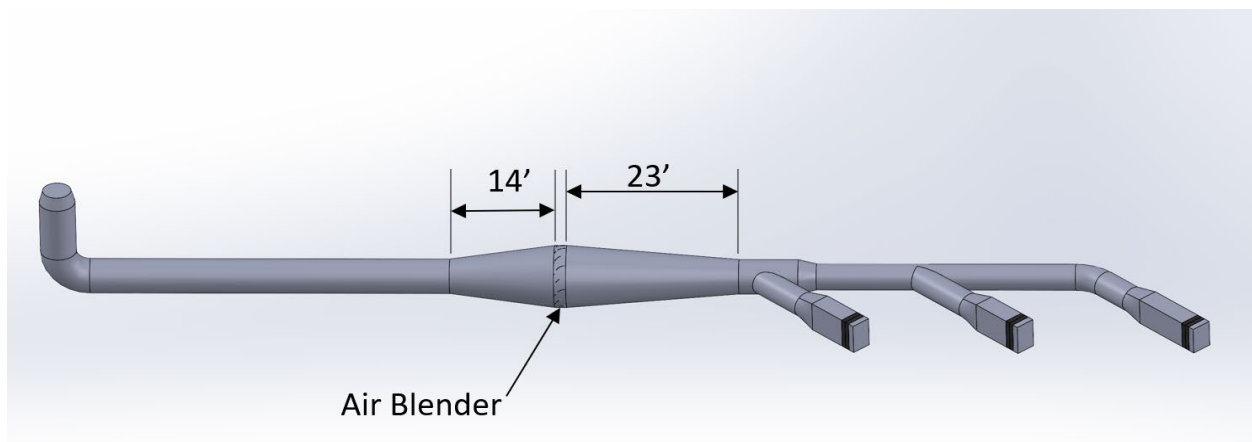


Figure 7. 3430 Building Exhaust System with Static Air Blender

All cases were re-run with the air blender included in the system model. The results of these runs are summarized in Table 6. Results show maximum average flow angles ranging from 5.55° to 17.34° , which is within the ANSI/HPS N13.1-2011 standard limit of 20° . The velocity uniformity COV values range from 0.98% to 3.90%. Gas tracer COV values range from 0.34% to 5.21%, and particle tracer COV values range from 5.16% to 18.11%. All resulting COV values were below the limit of 20%. And in no case was the maximum gas tracer concentration deviation from the mean greater than 30%. 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 for this 3430 Building Exhaust system design with the addition of the air blender installed using the expansion/blender/contraction configuration.

Table 6. Summary of CFD Modeling Results for the 3430 Building Duct with an Air Blender

Operating Fans	Injection Points	Airflow per Fan [cfm]	Total Airflow [cfm]	Velocity Uniformity	Mean Absolute Flow Angle	Gas Tracer Uniformity	Particle Tracer Uniformity	
				% COV	Degrees	% COV	% Deviation from Mean	%COV
Fan1	Junction	11,400	11,400	2.72	8.84	2.34	3.17	15.32
Fan2	Junction	11,400	11,400	3.67	17.34	0.84	1.85	13.02
Fan3	Junction	11,400	11,400	2.89	7.37	0.82	1.74	9.19
Fan1	Junction	38,000	38,000	3.49	11.58	0.84	1.44	14.20
Fan2	Junction	38,000	38,000	2.43	15.68	0.34	2.43	10.24
Fan3	Junction	38,000	38,000	1.46	8.43	0.38	0.48	17.84
Fan1,2,&3	Junction	11,400	34,200	2.11	8.15	3.17	4.92	16.88
Fan1,2,&3	Junction	38,000	114,000	0.98	5.55	2.98	5.20	14.38
Fan1&2	Junction	11,400	22,800	2.77	12.56	4.22	9.59	10.64
Fan1&2	Junction	38,000	76,000	3.40	14.73	2.40	7.97	9.13
Fan2&3	Junction	11,400	22,800	3.90	13.99	3.40	9.12	9.76
Fan2&3	Junction	38,000	76,000	3.06	14.57	2.84	6.45	17.78
Fan1&3	Junction	11,400	22,800	2.05	11.25	1.84	1.97	16.56
Fan1&3	Junction	38,000	76,000	3.18	11.05	0.82	2.83	17.13
Fan1	Fan 1	11,400	11,400	1.94	9.34	0.41	1.02	11.43
Fan2	Fan 2	11,400	11,400	3.70	17.30	0.61	0.68	15.71
Fan3	Fan 3	11,400	11,400	2.38	6.55	1.05	2.26	14.66
Fan1	Fan 1	38,000	38,000	2.41	11.53	0.52	1.12	5.16
Fan2	Fan 2	38,000	38,000	2.63	15.33	0.80	1.15	5.55
Fan3	Fan 3	38,000	38,000	2.03	9.04	1.07	2.24	18.11
Fan1,2,&3	Fan 2	11,400	34,200	2.15	9.90	5.21	7.20	7.05
Fan1,2,&3	Fan 2	38,000	114,000	1.72	6.45	3.48	9.71	15.02
Fan1&2	Fan 2	11,400	22,800	2.38	12.02	1.85	4.44	16.15
Fan1&2	Fan 2	38,000	76,000	2.71	14.67	2.01	7.09	8.83
Fan2&3	Fan 2	11,400	22,800	3.70	13.61	1.25	3.58	12.10
Fan2&3	Fan 2	38,000	76,000	2.63	13.69	1.95	2.91	6.29
Fan1&3	Fan 1	11,400	22,800	2.15	10.88	0.36	0.84	11.51
Fan1&3	Fan 1	38,000	76,000	3.35	11.29	1.19	1.63	6.54

Figure 8 shows a plan view of the velocity magnitude at the duct mid-plane (top) and resultant particle and tracer gas distributions at the sampling point (bottom) for the 114,000-cfm case running Fans 1, 2, and 3 with the injection point at the junction. With the addition of the air blender, tracer uniformity mixing is acceptable.

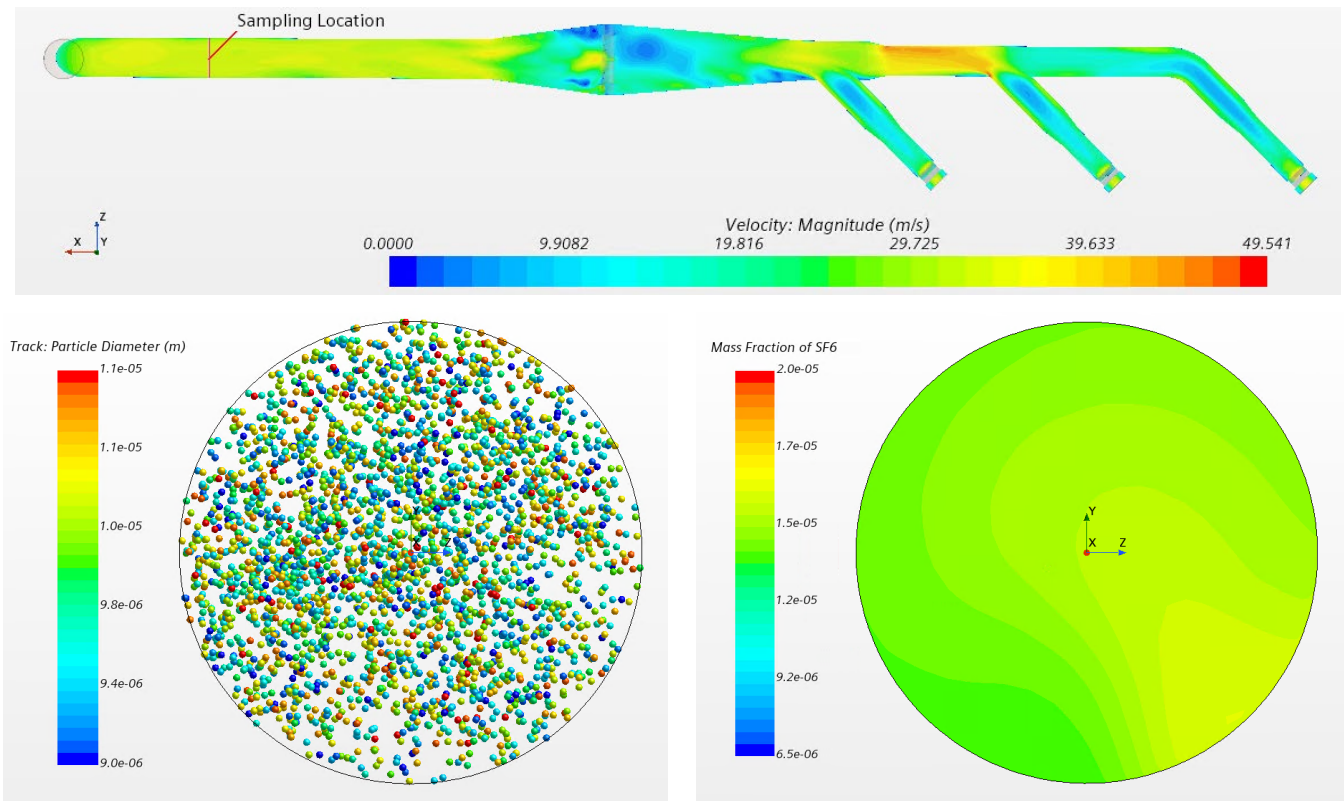


Figure 8. Velocity Magnitude at Mid-Duct. Plan view (top) and particle and tracer gas distributions at the sampling point (bottom) for the 114,000-cfm case operating Fans 1, 2, and 3 with the injection point at the junction and the inclusion of an air blender in the exhaust system design.

5.0 Conclusions

Based on CFD modeling of the 3430 Building filtered exhaust stack system, the following conclusions are drawn:

- CFD model benchmarking showed that modeling results of flow angle, velocity uniformity COV, gas tracer uniformity COV, and particle-tracer uniformity COV values are in good agreement with those derived from testing of the original stack configuration with two fans and tests of the HV-C2 physical test model used to help qualify the 3430 Building stack.
- Modeling results for the initial three-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 the initial three-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.
- Modeling results of the three-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 when the air blender is installed using the expansion/blender/contraction design. This design also includes an exhaust exit cone.
- The process of CFD modeling meets the intent of optimizing and upgrading a new or existing system as described in Section 6.9 of ANSI/HPS N13.1-2021. Specifically, the CFD modeling of the re-designed exhaust system shows that without an air blender, gas uniformity and particle tracer uniformity conditions could not be met.

6.0 References

Ballinger MY, KP Recknagle, and JM Barnett. 2003. *Comparison of a Computational Fluid Dynamics Model with Exhaust Flow Data from a Scale Model Stack*. Health Physics 84(6):S242-S243.

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. Pacific Northwest National Laboratory, Richland, Washington.

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.

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

Barnett, JM, X-Y Yu, SR Suffield, KP Recknagle. 2020. *Modeling Filtered Building Effluent Stack Sampling Points for Qualification Criteria*. Progress in Nuclear Energy 124:103338. DOI: 10.1016/j.pnucene.2020.103338

Barnett JM, and SF Snyder. 2021. *Pacific Northwest National Laboratory Facility Radionuclide Emission Points and Sampling Systems*. PNNL-15992, Revision 5. Pacific Northwest National Laboratory, Richland, Washington.

Flaherty, JE, JP Rishel, SR Suffield, JM Barnett. 2020. *Assessment of the 3420 Building Filtered Exhaust Stack Sampling Probe Location – Stack Verification Following Fan and Air Blender Additions*. PNNL-30747. Pacific Northwest National Laboratory, Richland, Washington.

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

Glissmeyer JA and JE Flaherty. 2010. *Assessment of the 3430 Building Filtered Exhaust Stack Sampling Probe Location*. PNNL-19262, Revision 1. Pacific Northwest National Laboratory, Richland, Washington.

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

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

Health Physics Society (HPS). 2021. *Sampling and Monitoring Releases of Airborne Radioactive Substances from the Stack and Ducts of Nuclear Facilities*. ANSI/HPS N13.1-2021. Health Physics Society, Herndon, Virginia.

Incropera FP and DP DeWitt. 1985. *Introduction to Heat Transfer*. Second edition, John Wiley and Sons, New York. ISBN-13: 978-0471612476

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.

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.

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

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

Recknagle KP, SR Suffield, JM Barnett. 2018b. *Qualification of ANSI/HPS N13.1-2011 Mixing Criteria by Computation Fluid Dynamics Modeling for the 3420 Building Fan Addition and Increased Ventilation Capacity*. PNNL-27826. Pacific Northwest National Laboratory, Richland, Washington.

Siemens PLM Software. 2021. *STAR-CCM+16.02.009 User Guide*. Siemens PLM Software, Plano, Texas.

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.

Pacific Northwest National Laboratory

902 Battelle Boulevard
P.O. Box 999
Richland, WA 99354
1-888-375-PNNL (7665)

www.pnnl.gov