An Efficient Tool-Set for Modeling Containment Ventilation in Very Complex Facilities-17215

Ryan Somero*, Lingaiah Mendu**, and Susan Brooks***,
* Newport News Shipbuilding – John.R.Somero@hii-nns.com
** Newport News Shipbuilding – Lingaiah.N.Mendu@hii-nns.com
*** HII - Stoller Newport News Nuclear - Susan.Brooks@hii-nns.com

ABSTRACT

The applicability of a multi-zone ventilation analysis program, CONTAM, to complex radioactive waste material handling or processing plants is evaluated. Validation results are based on simulations from three-dimensional multi-physics computational fluid dynamics. Insight into the application limits of CONTAM and conditions under which the use of higher fidelity tools are required are also discussed.

INTRODUCTION

Proper design of ventilation systems within facilities that handle radioactive material is critical to safe operation. This requires that the ventilation system be designed to maintain cascading, decreasing pressure from low to highly contaminated spaces over a wide range of operating scenarios and facility arrangements. For a structure of significant size and complexity, analysis of each scenario in order to challenge the plethora of operations and facility configurations would be time prohibitive, especially using traditional ventilation analysis techniques.

CONTAM [1] is an analytical tool developed, verified and validated by the National Institute of Standards and Technology (NIST) to analyze facility ventilation systems [2-4]. The application of CONTAM to large, complex structures, however was previously undocumented. CONTAM applies simplifying assumptions, which significantly reduce its computational burden, but potentially at the cost of accuracy and fidelity. For example, CONTAM assumes instantaneous and uniform distribution of properties such as pressure, temperature and contaminants within a space, ignoring inhomogeneities, gravitational affects and any time dependencies or transients. These assumptions are valid for many applications and facilities, where gases or vapors are uniform and short-lived excursions do not affect safety or performance. Whether CONTAM could be applied to multi-level, manned, contaminated processing facilities with simultaneous operations was not indicated in the literature. Moreover, the application of CONTAM to critical operating scenarios and highly inhomogeneous environments could not be substantiated.

DESCRIPTION of Methods

In order to confirm the appropriate use of CONTAM for complex radioactive material handling or processing plants, a range of validation simulations were conducted using the Computational Fluid Dynamics (CFD) code ANSYS Fluent. ANSYS Fluent is well validated for its applicability in simulating complex fluid flows, as it solves the full time-dependent Navier Stokes Equations [5], avoiding the simplifying assumptions of CONTAM.

A range of potentially challenging scenarios were selected from a CONTAM model that focused on a radioactive waste vitrification cave and modeled in ANSYS Fluent, Figure 1. The level of contamination within a space was designated by a contamination class, with C5 having the highest level of contamination. CFD models were built using the facility geometry for the same spaces for which CONTAM represented the physical structure as a single zone incorporating a uniform volume for a given floor area.

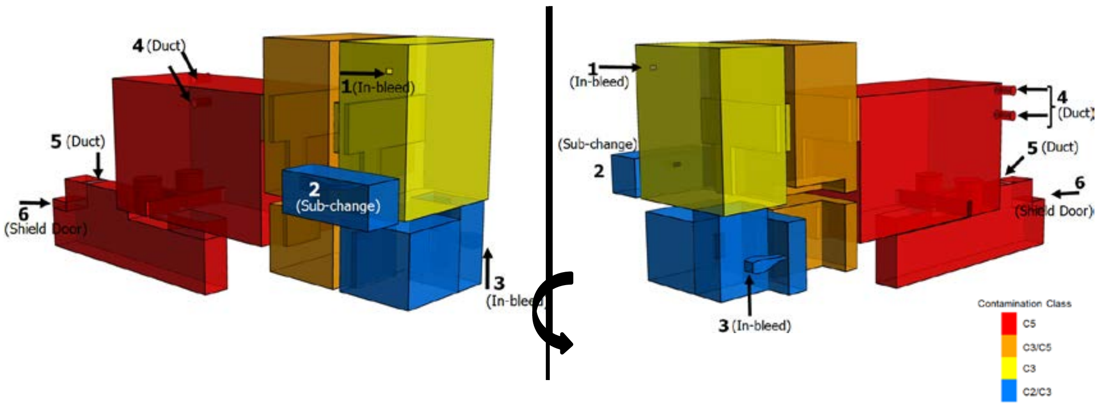


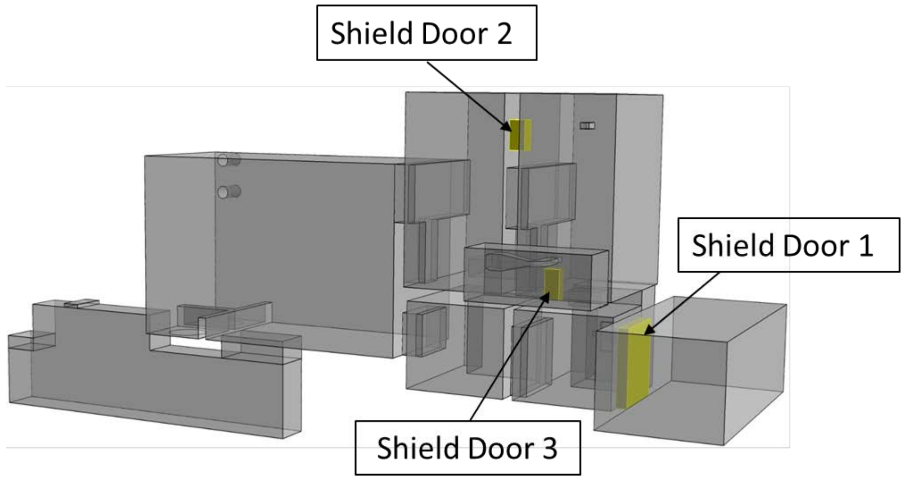
Fig. 1. Vitrification cave - ANSYS Fluent model

Scenario 1 was the baseline condition where all normally closed elements were shut and all normally open elements were open. Elements here refer to any variable flow path such as doors, hatches, dampers, etc. Boundary conditions applied to the baseline condition are shown in Table I.

TABLE I – Vitrification cave scenario 1 boundary conditions

Boundary Condition	Flow Rate (m^3/min) / (CFM)	Pressure (Pa)	Temperature (C)
1	39.7 (1400)	-	26.7
2	61.2 (2160)	-	26.7
3	76.7 (2707)	-	26.7
4	289.5 (10214)	-	20
5	-	-326.9	45
6	68.7 (2426)	-	35

Three additional scenarios were also selected, each having a single element that is normally closed in the open position. Figure 2 summarizes these conditions and the applicable element that was manipulated.



Scenario	Open Elements (Normally Closed)
1	None
2	Shield Door 1
3	Shield Door 2
4	Shield Door 3

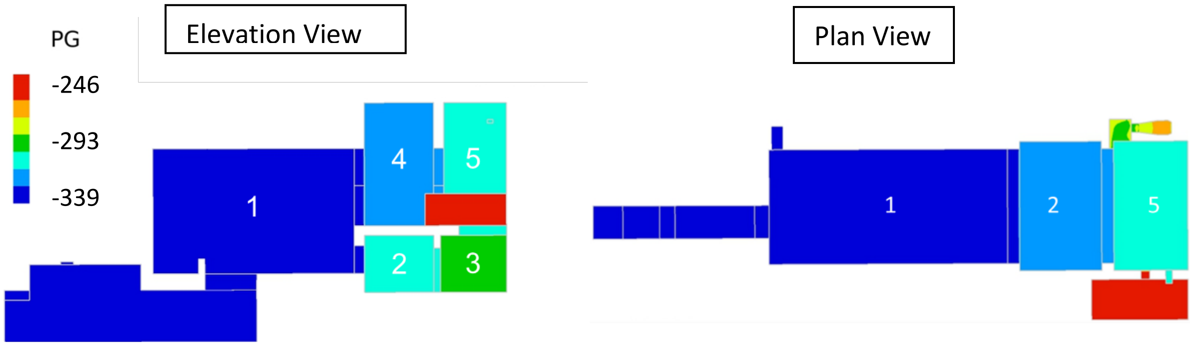
Fig. 2. Scenario operational configurations

The ANSYS Fluent model used ~18-million finite element cells with boundary layer spacing imposed around passageways and engineered gaps. The two equation k-omega turbulence model was used and pressure and momentum equations were solved with 2^{nd} order accuracy. Simulations were run until a steady state condition was reached with residuals less than e^{-6}.

DISCUSSION of Results

The volume weighted steady state pressure within each space was compared to the results from CONTAM, using CONTAM as the reference value. A positive percent difference reflects CFD analysis results in a lower volume weighted average pressure than the CONTAM single "node" pressure, and vice versa.

Results from the baseline condition, Scenario 1, are provided in Figure 3.

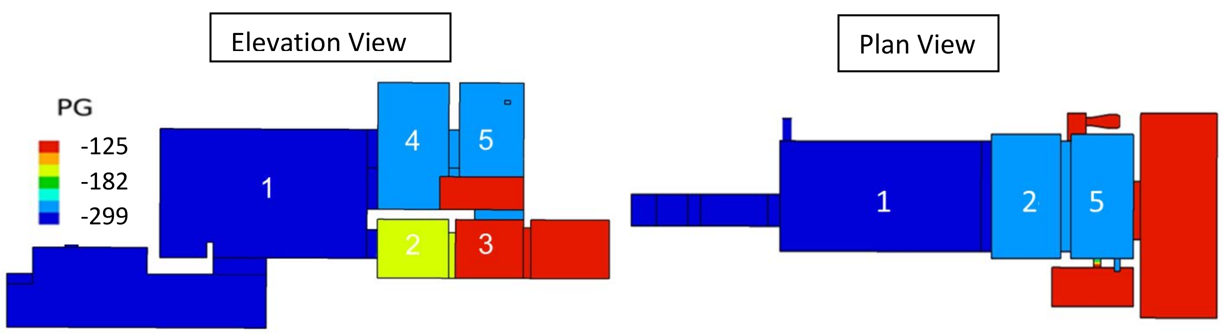


Room ID	Zone Design Pressure	Contamination Class	Zone Design Pressure Range	CONTAM Pressure	CFD Pressure	Difference
1	-348.6	C5	-249 to -349	-324.5	-327.0	0.77%
2	-298.8	C3/C5	-75 to -349	-312.3	-305.3	-2.24%
3	-249	C2/C3	-25 to -100	-302.7	-284.9	-5.88%
4	-298.8	C3/C5	-75 to -349	-315.3	-317.0	0.54%
5	-273.9	C3	-75 to -100	-309.0	-307.0	-0.65%

Fig. 3. Scenario 1 zone pressure results (Pa)

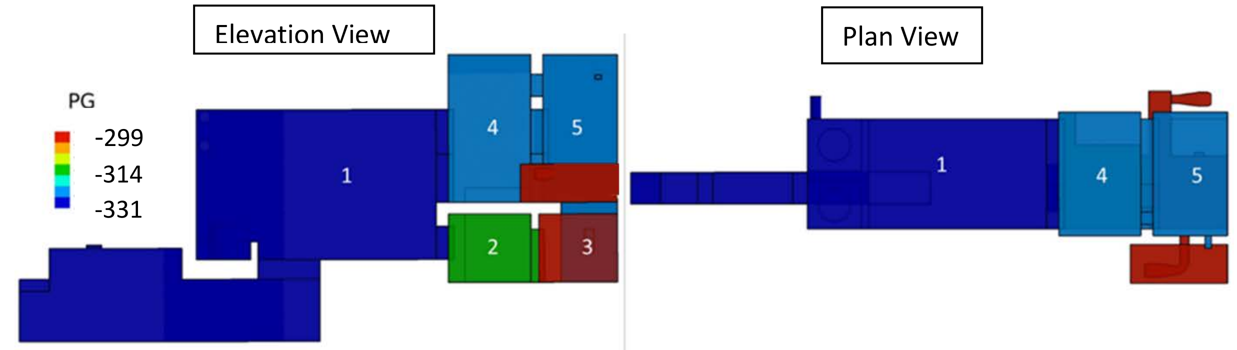
Simulated pressures are observed to be on the order of the zone design pressures. Variation in pressure between CFD and CONTAM is observed to be normally less than 1% with a maximum variation less than 6%. This result provides confidence that CONTAM is well suited for the prediction of the baseline design operating condition.

Figures 4-6 provide comparison between CFD and CONTAM for the off-design configurations.



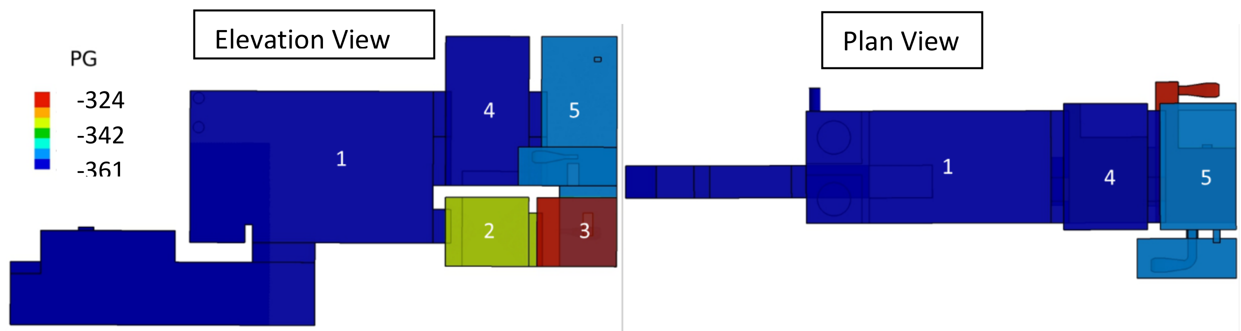
Room ID	Zone Design Pressure	Contamination Class	Zone Design Pressure Range	CONTAM Pressure	CFD Pressure	Difference
1	-348.6	C5	-249 to -349	-265.4	-278.8	5.05%
2	-298.8	C3/C5	-75 to -349	-168.3	-184.2	9.45%
3	-249	C2/C3	-25 to -100	-92.1	-92.2	0.01%
4	-298.8	C3/C5	-75 to -349	-256.2	-271.1	5.8%
5	-273.9	C3	-75 to -100	-250.0	-263.3	5.32%

Fig. 4. Scenario 2 zone pressure results (Pa)



Room ID	Zone Design Pressure	Contamination Class	Zone Design Pressure Range	CONTAM Pressure	CFD Pressure	Difference
1	-348.6	C5	-249 to -349	-329.1	-330.9	0.55%
2	-298.8	C3/C5	-75 to -349	-317.3	-315.6	-0.54%
3	-249	C2/C3	-25 to -100	-308.1	-301.0	-2.3%
4	-298.8	C3/C5	-75 to -349	-321.1	-324.6	1.09%
5	-273.9	C3	-75 to -100	-320.5	-324.3	1.18%

Fig. 5. Scenario 3 zone pressure results (Pa)



Room ID	Zone Design Pressure	Contamination Class	Zone Design Pressure Range	CONTAM Pressure	CFD Pressure	Difference
1	-348.6	C5	-249 to -349	-360.5	-359.9	-0.17%
2	-298.8	C3/C5	-75 to -349	-347.9	-340.3	-2.18%
3	-249	C2/C3	-25 to -100	-338.0	-321.6	-4.85%
4	-298.8	C3/C5	-75 to -349	-354.9	-357.6	0.76%
5	-273.9	C3	-75 to -100	-352.9	-355.4	0.70%

Fig. 6. Scenario 4 zone pressure results (Pa)

Agreement between the calculated CONTAM node zone pressure values and the CFD volume weighted average values is within 10% for all four scenarios, with the majority of results within 2%. Considering the range of scenarios, these results provide confidence that CONTAM can also well predict off-design conditions.

Velocity Profiles

In addition to verifying the pressure profiles established by CONTAM, ANSYS Fluent was used to investigate the velocity profiles in the areas of engineered gaps, gaps between major doorways designed to ensure cascading flow and required capture velocities are maintained. Figure 7 plots velocity contours for Scenario 2, highlighting the flow through the vitrification cave door engineered gap.

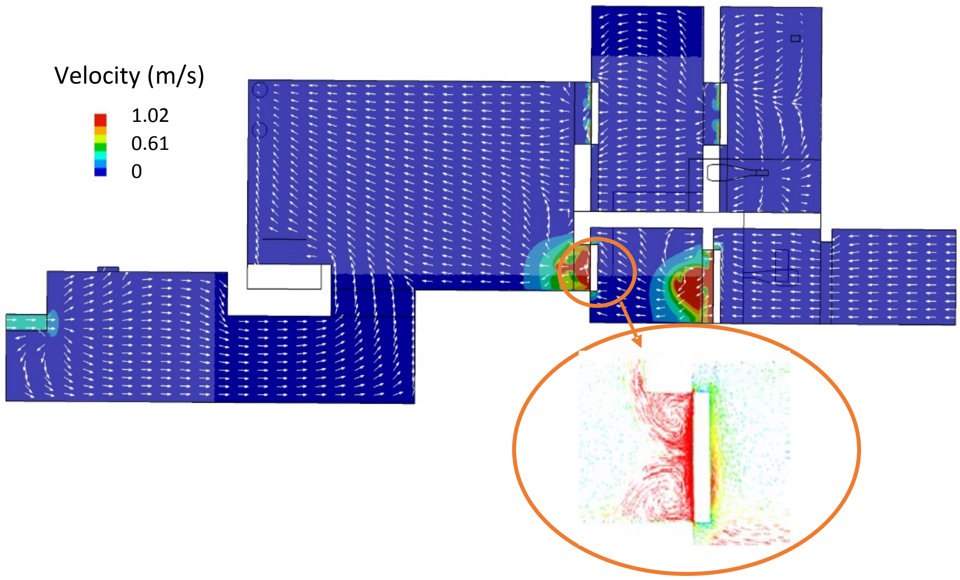


Fig. 7. Scenario 2 velocity profile

Capture velocities were observed to maintain above the required 1 m/s between the C5 and C3/C5 spaces, but also revealed large regions of recirculating flow on the downstream side of the gaps. While not itself a safety concern, these recirculation zones present the possibility of particulate/contaminate buildup in the recirculation zones.

It was also observed that again in these scenarios, pressures were relatively uniform throughout each space, save for the areas directly around engineered gaps. Pressure variations were observed on the leeward side of doorways due to the localized circulation that formed there, Figure 8.

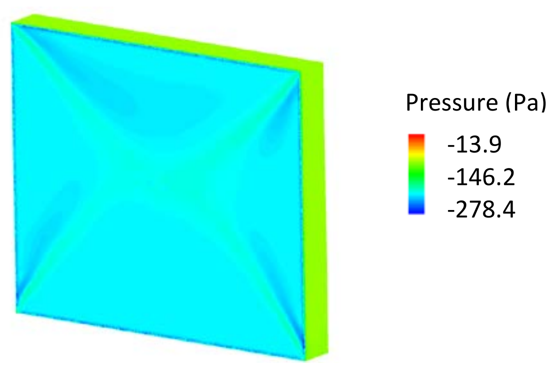


Fig. 8. Leeward door pressures

Transient Simulations

The fundamental assumption of the equal and instantaneous distribution of pressure within a space can be challenged by larger spaces with doors opening to large pressure deltas. One of the facility's largest spaces with also one of the largest pressure deltas across its door is at the vitrification cave/airlock transition. The transient impact of opening one of the main shield doors was thereby simulated using a sliding mesh in ANSYS Fluent, Figure 9.

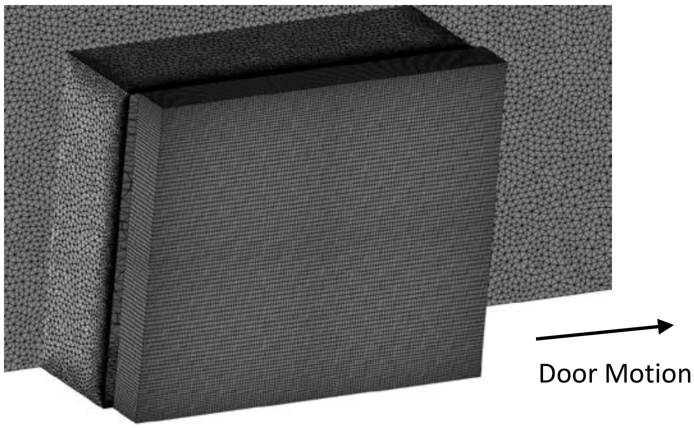


Fig. 9. Model mesh at door opening

The baseline balanced condition showed high velocity airflow through the engineered gaps. As the door is opened, the inlet velocity initially remains high and localized, but decreases in magnitude and increases in breadth as the gap widens, Figure 10.

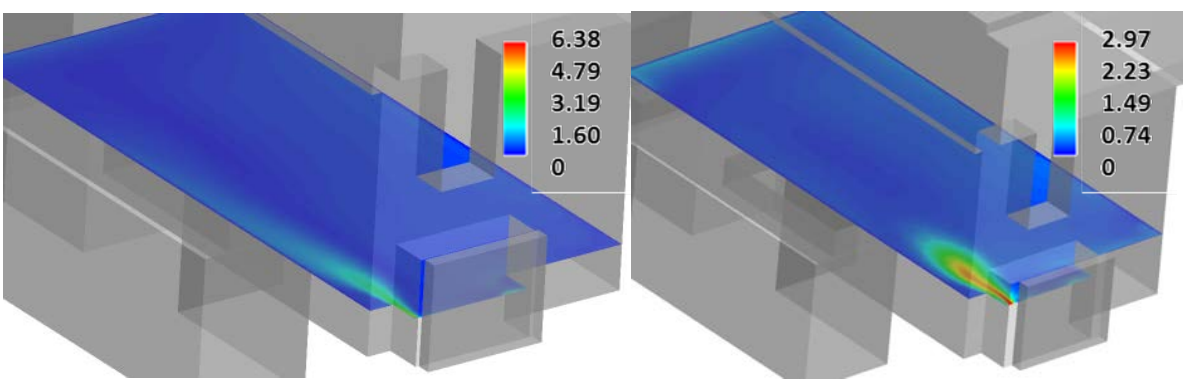


Fig. 10. Evolution of vitrification cave inlet velocities (m/s)

Flow velocities within the space modify the pressure distribution within the space and along the walls as shown in Figure 11. The pressures within the space are observed to be within 6 Pa, with the primary pressure deltas occurring at the passageways, providing confidence in the validity of the assumption to ignore flow momentum and pressure variations within spaces where door motions are relatively slow.

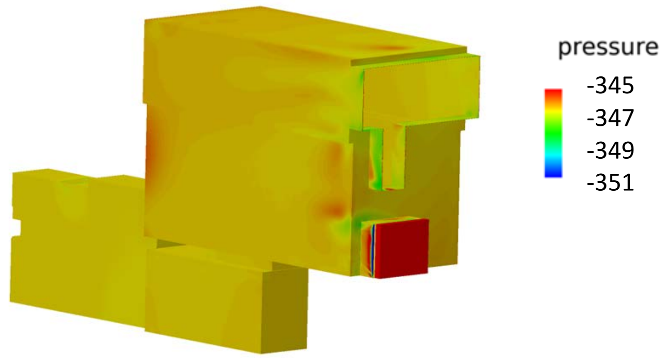


Fig. 11. Pressure distribution of vtrification cave and tunnels (Pa)

Thermal Analysis

A limitation of CONTAM is the ability to account for discrete heat sources and their impact on the flow field of a space. CONTAM accounts for the overall average temperature in a space, but not the distribution of heat sources. Significant heat loads, such as melters used for vitrification, have the ability to generate buoyancy driven flows. A configuration of the baseline condition was used containing a pair of melters at 982 °C degrees, Figure 12.

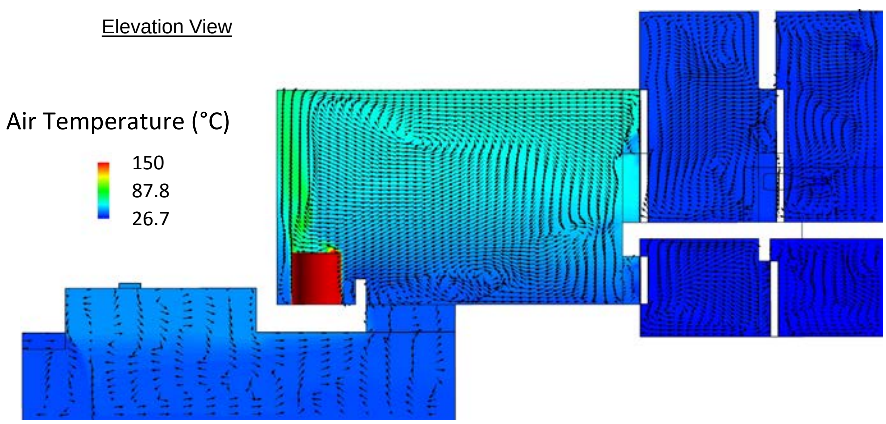


Fig. 12. Scenario A1 thermal results with flow velocity vectors

The pressure differential across the doors was extracted in Figure 13 with comparison to the CONTAM pressure deltas also provided in Table II. It should be noted that the CFD model included hydrostatic pressures which varied with height within each space, whereas CONTAM has a single pressure value for all locations within a space.

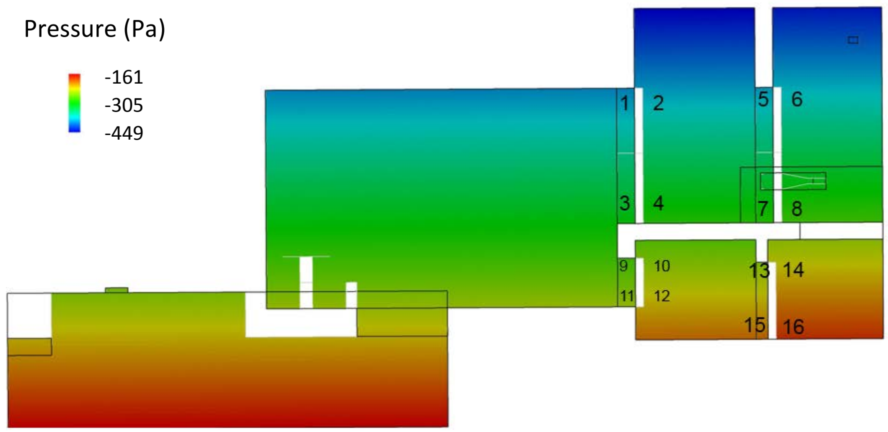


Fig. 13. Pressure profile & deltas with hot melters

Table II – Comparison of CONTAM & CFD pressure differentials (Pa)

Node #	Zone Design Pressure	Contamination Class	CONTAM Pressure	Fluent Pressure	DP CONTAM	DP Fluent
1	-349	C5	-324.5	396.7	-9.3	-6.4
2	-299	C3/C5	-315.3	390.3		
3	-349	C5	-324.5	306.7	-9.3	-14.2
4	-299	C3/C5	-315.3	292.6		
5	-299	C3/C5	-312.3	389.2	-3.3	-8.7
6	-274	C3	-309.0	380.5		
7	-299	C3/C5	-315.3	293.3	-6.2	-10.2
8	-274	C3	-309.0	283.1		
9	-349	C5	-324.5	281.9	-12.3	-21.4
10	-299	C3/C5	-312.3	260.5		
11	-349	C5	-324.5	248.7	-12.3	-23.3
12	-299	C3/C5	-312.3	225.4		
13	-299	C3/C5	-312.3	255.3	-9.6	-20.5
14	-249	C2/C3	-302.7	234.8		
15	-299	C3/C5	-312.3	201.7	-9.6	-21.8
16	-249	C2/C3	-302.7	180.0		

Significant variations were observed between the CONTAM and CFD pressures at individual locations due to the inclusion of the hydrostatic pressure. Comparison of the volume averaged values, though, showed relatively good agreement between CONTAM and CFD w/o hydrostatic pressure, considering CONTAM did not account for discrete thermal effects, Table III. In general, CFD predicted higher pressure deltas across the shield doors than CONTAM, other than the upper point of the upper level.

Table III – Zone pressure comparison with hot melters

Room ID	Zone Design Pressure	Contamination Class	Zone Design Pressure Range	CONTAM Pressure	CFD Pressure	Difference
1	-348.6	C5	-249 to -349	-324.5	-329.7	1.60%
2	-298.8	C3/C5	-75 to -349	-312.3	-305.8	-2.08%
3	-249	C2/C3	-25 to -100	-302.7	-284.4	-6.04%
4	-298.8	C3/C5	-75 to -349	-315.3	-312.1	-1.01%
5	-273.9	C3	-75 to -100	-309.0	-301.8	-2.33%

Three dimensional thermal evaluation may prove useful for evaluating potential contamination distributions in the event of vapor release, or to evaluate canister cleaning evolutions where air velocities will effect potential contamination settling behaviors and buildup.

CONCLUSIONS

The level of agreement observed in this study provides confirmation that CONTAM can satisfy its performance goal – to provide guidance in examining the HVAC system under steady state conditions using idealized components in time-independent evolutions.

To the extent that physical arrangement, lag-time and non-ideal performance can be neglected, and that the relevant scenarios are considered, CONTAM can provide system containment and component capacity guidance and insights. The results provided confidence that CONTAM produced reliable pressures and flow rates that could be used to determine where the required containment conditions could be challenged based on variable facility operations. In addition, the use of CONTAM should be considered for application in similar facility designs with a higher fidelity tool recommended for specific scenarios where transient effects or contaminant distribution could prove important. Additional study could also prove significant in examining where expensive or challenging conservative assumptions may be relaxed.

REFERENCES

1. W. S. Dols and B. Polidoro, NIST Technical Note 1887 – CONTAM User Guide and Program Documentation Version 3.2 (2016)
2. L. Wang, *Coupling of Multizone and CFD Programs for Building Airflow and Contaminant Transport Simulations.* PhD Dissertation. Purdue University, Lafayette (2007).
3. F. Haghighat and H. Li, “Building Airflow Movement – Validation of Three Airflow Models”, *Journal of Architectural and Planning Research Vol. 21* (2004)

4. L. Wang, *Using CFD Capabilities of CONTAM 3.0 for Simulating Airflow and Contaminant Transport in and around Buildings.* HVAC&R Research Journal 16 (2010).
5. *ANSYS Fluent Theory Guide*. ANSYS Inc. (2016)