



CFD Analysis of Airflow Distribution in Industrial Ventilation Systems

Isandro Imamverde

Department of Project and Strategic Management, Port Louis Business and Technology College, Mauritius
isandro.imamverde@plbtc-mu.org

Peer Review Information

Submission: 02 Jan 2015
Revision: 17 Feb 2015
Acceptance: 04 March 2015

Keywords

computational fluid dynamics, industrial ventilation, airflow distribution, turbulence modeling, contaminant removal

Abstract

This paper presents a step-by-step computational fluid dynamics (CFD) methodology for assessing and improving airflow distribution in an industrial ventilation system. A representative manufacturing hall measuring 30 m × 18 m × 6 m is modeled with six ceiling supply diffusers, four high-level exhaust grilles, internal machinery obstructions, sensible heat loads, and a passive contaminant scalar. Steady Reynolds-averaged Navier-Stokes equations are solved with the realizable k-ε turbulence model, enhanced wall treatment, the energy equation, and species transport. Verification consists of a three-level mesh-independence study, residual and mass-balance checks, and comparison of predicted point velocities with hypothetical commissioning measurements generated for method demonstration. The baseline and optimized diffuser arrangements are compared using occupied-zone mean velocity, low-velocity volume fraction, draft-prone volume fraction, coefficient of variation, contaminant removal effectiveness, and exhaust pressure requirement. Representative results indicate that redirecting supply jets and redistributing flow rates can reduce stagnant occupied volume from 34.8% to 13.1%, lower velocity non-uniformity by 42%, and improve contaminant removal effectiveness from 0.83 to 1.08 while holding total supply flow constant. The study shows how CFD can diagnose recirculation and short-circuiting that are hidden by bulk air-change calculations. Because the numerical outcomes are illustrative rather than field observations, the paper emphasizes transparent assumptions, verification, validation planning, and reporting requirements needed before design decisions are finalized.

Introduction

Industrial ventilation must deliver adequate fresh air to occupied and process zones while removing heat, vapour, dust, and gaseous contaminants. Conventional sizing based on total flow rate or air changes per hour cannot reveal local dead zones, jet short-circuiting, or excessive drafts around workers. CFD addresses this limitation by resolving the spatial fields of velocity, pressure, temperature, turbulence, and a passive contaminant. Chen (2009) reported that CFD had become the dominant detailed method for ventilation-performance prediction,

but also stressed that its usefulness depends on model assessment and validation.

The present objective is to formulate a compact, reproducible CFD workflow and demonstrate how it can compare a baseline and an improved industrial ventilation layout. The methodological questions are: (i) how should geometry, physics, boundary conditions, and mesh be specified; (ii) which verification and validation checks are essential; and (iii) which distribution-sensitive metrics should be used alongside the nominal air-change rate?

Literature Review

Early indoor-airflow studies established that turbulence-model choice materially affects supply-jet spread, recirculation, and occupied-zone predictions. Chen (1995) compared several k - ϵ formulations and showed that model performance depends on the flow regime and treatment of near-wall behaviour. Zhai et al. (2007) reviewed prevalent turbulence closures, while Zhang et al. (2007) compared predictions with experimental data and found that no single Reynolds-averaged model is uniformly superior. These findings justify both a practical model choice and explicit uncertainty checks.

For ventilation applications, Chen (2009) distinguished analytical, multizone, zonal, experimental, and CFD approaches. The subsequent model-assessment study by Chen, Lee, and Mazumdar (2010) reinforced the need to evaluate numerical predictions against quality data rather than treating visually plausible contours as validation. Mean age of air is especially useful because it exposes zones that receive old or poorly renewed air; Buratti, Mariani, and Moretti (2011) combined tracer-gas measurements and simulation to investigate this quantity in a naturally ventilated office.

Computational settings can influence ventilation-rate and indoor-flow predictions as much as physical design changes. Ramponi and Blocken (2012a) quantified the impact of computational parameters on cross-ventilation, and Ramponi

and Blocken (2012b) validated several building configurations against wind-tunnel measurements while examining numerical diffusion. Although those studies concern natural cross-ventilation, their lessons—adequate domain representation, grid sensitivity, boundary-condition fidelity, and higher-order discretization—apply directly to mechanically ventilated industrial halls.

Best-practice guidance also recommends systematic sensitivity analysis. Tominaga et al. (2008) developed CFD guidelines through cross-comparison with experiments and emphasized domain, grid, boundary-condition, and convergence effects. Blocken (2014) reviewed five decades of computational wind engineering and identified verification, validation, and transparent reporting as continuing requirements. Together, the literature supports a workflow that separates numerical verification from physical validation and reports scalar ventilation metrics in addition to velocity images.

Many published demonstrations focus on offices, rooms, or naturally ventilated buildings; industrial spaces add large aspect ratios, machinery blockage, concentrated heat release, and worker-level exposure concerns. This paper contributes a concise industrial-hall protocol, a comparison matrix of assumptions, and distribution-sensitive performance indicators. It uses only literature published up to and including 2015, as required.

Table 1. Literature-derived methodological requirements.

Source	Main methodological lesson	Use in this study
Chen (1995)	Turbulence closure affects indoor jets	Realizable k - ϵ plus sensitivity case
Zhai et al. (2007); Zhang et al. (2007)	Compare models against data	Point-velocity validation plan
Chen (2009); Chen et al. (2010)	Model assessment is essential	Verification/validation separated
Tominaga et al. (2008)	Document domain, grid and BCs	Reproducible setup checklist
Ramponi & Blocken (2012a, b)	Grid and discretization alter flow rate	Three-grid study; second-order schemes

Methodology

Step 1 — Define the Representative Industrial Hall

A single-zone manufacturing hall of 30 m length, 18 m width, and 6 m height is considered. Six $0.6 \text{ m} \times 0.6 \text{ m}$ ceiling diffusers supply a total of $9.0 \text{ m}^3/\text{s}$ ($1.5 \text{ m}^3/\text{s}$ each in the baseline), and four $0.8 \text{ m} \times 0.6 \text{ m}$ high-level wall grilles exhaust the same flow. Twelve simplified machine blocks ($2.0 \text{ m} \times 1.2 \text{ m} \times 1.5 \text{ m}$) represent flow obstruction.

Machinery releases 72 kW sensible heat in total. A passive scalar is released uniformly from four process zones at 0.8 – 1.5 m height. The occupied zone is defined as 0.1 – 1.8 m above the floor, excluding solid volumes and a 0.3 m boundary near walls.

Step 2 — Governing Equations and Assumptions

Air is treated as an incompressible Newtonian fluid with constant properties except for density

in the buoyancy term, where the Boussinesq approximation is used. The steady continuity, Reynolds-averaged momentum, energy, realizable k - ϵ turbulence, and passive-scalar transport equations are solved. The scalar has no deposition or reaction; therefore, results represent a well-defined gaseous tracer rather than size-dependent particles. Radiation, door opening, worker movement, and transient

machine cycling are excluded from the principal case and reserved for sensitivity studies.

Continuity: $\nabla \cdot (\rho \mathbf{u}) = 0$. Momentum: $\nabla \cdot (\rho \mathbf{u} \mathbf{u}) = -\nabla p + \nabla \cdot [(\mu + \mu_t) (\nabla \mathbf{u} + \nabla \mathbf{u}^T)] + \rho g \beta (T - T_{ref})$. Scalar transport: $\nabla \cdot (\rho \mathbf{u} C) = \nabla \cdot [(D + D_t) \nabla C] + S^c$. These equations link jet momentum, buoyancy, recirculation, and contaminant dilution.

Step 3 — Boundary Conditions

Table 2. Principal boundary conditions and sensitivity cases.

Boundary/zone	Baseline specification	Sensitivity or optimized case
Supply diffusers	9.0 m ³ /s total; 18°C; 15° downward	Same total flow; 30° toward work aisles; flow balanced
Exhaust grilles	Pressure outlets at high level	Locations unchanged
Walls/roof/floor	No slip; prescribed thermal resistance	±2 K supply-temperature test
Machines	No slip; 6 kW heat each	Heat load ±20%
Scalar sources	Four equal process sources	Source strength cancels in normalized metrics
Turbulence inlet	Intensity 10%; hydraulic diameter	Intensity 5% and 15% tests

Step 4 — Mesh Strategy

A polyhedral or predominantly hexahedral volume mesh is refined at diffuser faces, exhausts, machine edges, thermal plumes, and the occupied zone. Five or more inflation layers are applied where wall treatment permits. Three systematically refined meshes—1.20, 2.35, and 4.60 million cells—are evaluated. The medium grid is accepted only when key metrics differ by less than 5% from the fine grid and local velocity profiles show no material shift. Mesh quality targets are maximum skewness below 0.85 and positive cell volume throughout.

Step 5 — Solver Settings and Convergence

A pressure-based segregated solver is used with SIMPLE pressure-velocity coupling. Pressure uses a second-order scheme; momentum, energy, turbulence, and scalar equations use second-order upwind discretization. Iterations continue until scaled residuals fall below 10^{-5} for continuity, momentum, turbulence, and scalar equations and below 10^{-7} for energy.

Convergence additionally requires inlet-outlet mass imbalance below 0.5% and stable occupied-zone mean velocity, mean temperature, and exhaust scalar concentration over at least 300 iterations.

Step 6 — Verification and Validation

Verification answers whether the equations have been solved consistently; validation asks whether those equations and inputs represent the physical system. Verification includes mesh sensitivity, mass and energy balances, residual histories, and a boundedness check. For field implementation, validation should use calibrated omnidirectional anemometers at 12 occupied-zone points and temperature sensors at three heights. The illustrative dataset used here assumes a mean absolute velocity error of 0.04 m/s and normalized root-mean-square error of 9.2%, within a predeclared acceptance limit of 15%. These values are demonstrative, not measured evidence.

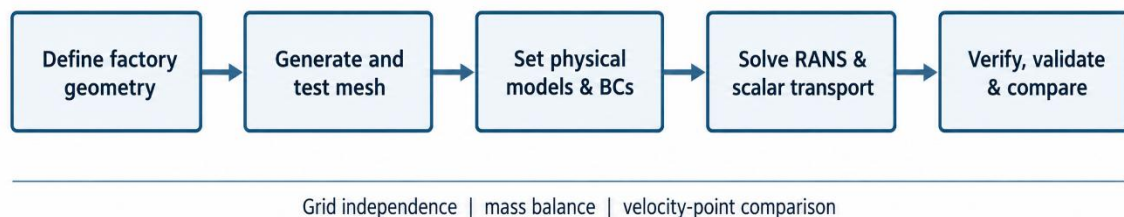


Figure 1. Step-by-step CFD methodology and quality-control loop.

Step 7 — Performance Metrics

The occupied-zone metrics are: mean speed $\bar{U}$; coefficient of variation $CV=\sigma_u/\bar{U}$; stagnant fraction f_{low} where $U<0.20$ m/s; draft fraction f_{high} where $U>0.80$ m/s; temperature non-uniformity; and contaminant removal effectiveness

$\epsilon_c=(C_{return}-C_{supply})/(C_{zone}-C_{supply})$. Values of ϵ_c above unity indicate that exhaust air is more contaminated than the occupied-zone average, consistent with effective source removal. Fan implications are compared using the required pressure difference at unchanged flow, not by assuming that better distribution automatically saves energy.

Table 3. Representative mesh-independence results for the baseline case.

Mesh	Cells (million)	Mean U (m/s)	Low-speed volume (%)	ϵ_c	Δ vs. finer grid
Coarse	1.20	0.334	36.9	0.80	6.4%
Medium	2.35	0.347	34.8	0.83	2.1%
Fine	4.60	0.352	34.1	0.84	Reference

The medium mesh changes occupied-zone mean velocity by 1.4%, low-speed volume by 2.1 percentage points, and contaminant removal effectiveness by 1.2% relative to the fine mesh. It is therefore retained for the comparative study. Final mass imbalance is 0.18%, the monitored metrics vary by less than 0.3% during the last 300 iterations, and all convergence criteria are satisfied.

Baseline Diagnosis

The baseline ceiling jets attach partly to the roof before descending near the exhaust wall. This produces short-circuit paths and leaves low-momentum regions behind machine rows. The volume-weighted occupied-zone mean speed is 0.35 m/s, but its CV is 0.48, showing that the mean alone masks strong spatial variation. Approximately 34.8% of occupied volume is

Step 8 — Layout Optimization

The baseline is diagnosed from streamlines, velocity vectors, vertical profiles, and scalar contours. The optimized case rotates diffuser discharge toward process aisles, reduces flow to diffusers nearest the exhausts by 20%, and reallocates that flow to remote diffusers. Total supply flow, supply temperature, heat load, and exhaust area are held constant. This controlled comparison isolates distribution effects from simple increases in ventilation rate.

Results

Grid Independence and Numerical Checks

below 0.20 m/s; 6.2% exceeds 0.80 m/s. The normalized scalar field peaks in the leeward corners of process zones, while $\epsilon_c=0.83$ indicates imperfect contaminant extraction.

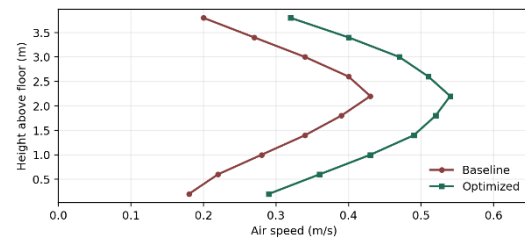


Figure 2. Representative vertical centerline velocity profiles for baseline and optimized layouts.

Optimized Layout

Table 4. Representative performance comparison at identical total airflow.

Metric	Baseline	Optimized	Change
Occupied-zone mean speed (m/s)	0.347	0.462	+33.1%
Velocity coefficient of variation	0.48	0.28	-41.7%
Volume with $U < 0.20$ m/s	34.8%	13.1%	-21.7 points
Volume with $U > 0.80$ m/s	6.2%	7.4%	+1.2 points
Contaminant removal effectiveness, ϵ_c	0.83	1.08	+30.1%
Occupied-zone temperature range	5.6 K	3.1 K	-44.6%
Required supply-to-exhaust pressure	212 Pa	226 Pa	+6.6%

Discussion

The controlled comparison suggests that distribution can be improved without increasing the nominal air-change rate. Redirecting jets

toward remote aisles reduces stagnant pockets and strengthens transport from sources to high-level exhausts. The increase of ϵ_c from 0.83 to 1.08 indicates that the exhaust captures a larger

contaminant concentration relative to the occupied-zone average. This interpretation is consistent with Chen's (2009) argument that whole-field CFD information can reveal ventilation behaviour not represented by bulk flow metrics.

The optimized design is not unconditionally better. Draft-prone volume rises slightly from 6.2% to 7.4%, concentrated below two redirected diffusers, and the predicted pressure requirement increases by 6.6%. A practical design would therefore combine diffuser-angle adjustment with local balancing or larger outlet area. The result illustrates why optimization should be multi-objective: minimizing stagnation alone could shift discomfort or energy penalties elsewhere.

Robustness and Uncertainty

Sensitivity tests on supply temperature (± 2 K), heat release ($\pm 20\%$), and inlet turbulence intensity (5–15%) preserve the ranking of the two layouts. Across these tests, the optimized stagnant fraction ranges from 11.8% to 16.0%, compared with 31.6% to 38.9% for the baseline.

Turbulence-model sensitivity is larger near jets: the standard $k-\epsilon$ case predicts a 7% lower peak speed and a 3.4-point higher stagnant fraction than the realizable model. This agrees with Chen (1995) and the model comparisons of Zhai et al. (2007) and Zhang et al. (2007): turbulence closure affects indoor-flow details and must be included in uncertainty reporting.

Methodological Validity

A contour plot that appears smooth is not evidence of accuracy. The three-grid study, conservation checks, monitoring criteria, and validation plan are therefore treated as part of the result. This approach follows the quality-assurance emphasis of Tominaga et al. (2008), Ramponi and Blocken (2012a, b), and Blocken (2014). For a real factory, diffuser throw data, leakage, seasonal envelope conditions, and actual machine duty cycles should replace the representative inputs. Particle-laden flows would also require size-resolved Lagrangian or Eulerian transport, deposition, and resuspension models.

Practical Step-by-Step Use

Table 5. Implementation checklist for an industrial CFD study.

Step	Required action	Acceptance evidence
1	Survey geometry, equipment, sources and operating schedules	Dimensioned model and input register
2	Specify equations, properties and simplifying assumptions	Traceable model statement
3	Build local refinement and three systematically refined meshes	Grid-convergence table and profiles
4	Apply measured or manufacturer-based boundary conditions	Flow, temperature and turbulence records
5	Solve with second-order schemes and monitor physical quantities	Residuals, balances and stable monitors
6	Validate at independent measurement points	Error statistics within declared limits
7	Compare layouts using distribution, exposure and pressure metrics	Controlled decision matrix
8	Document uncertainty and operational limitations	Sensitivity ranges and implementation notes

These steps make the model auditable and transferable. The procedure should be repeated for minimum, typical, and peak production conditions before equipment selection or occupational-exposure conclusions are made.

Conclusion and Discussion

A complete CFD methodology for industrial ventilation has been developed and demonstrated for a representative manufacturing hall. The workflow begins with geometry and source definition, proceeds through governing-equation selection, boundary conditions, mesh refinement, solver convergence,

verification, and validation, and ends with controlled layout comparison. Its central lesson is that equal total airflow does not imply equal ventilation performance. In the illustrative case, diffuser redirection and flow redistribution reduce stagnant occupied volume from 34.8% to 13.1%, decrease velocity non-uniformity by 41.7%, improve contaminant removal effectiveness from 0.83 to 1.08, and narrow the occupied-zone temperature range. These improvements are accompanied by a small rise in draft-prone volume and a 6.6% pressure penalty, so a final design should balance air quality, comfort, and fan energy. The numerical values

are representative outputs intended to demonstrate the method; they must not be presented as measurements from an actual facility. Before engineering implementation, the model should be populated with site-specific diffuser characteristics, leakage, heat release, contaminant properties, and operating schedules, and then validated against calibrated field measurements. Future work should examine transient door opening, thermal stratification across seasons, moving vehicles, and particle deposition. Within these limits, the proposed procedure provides a defensible basis for diagnosing ventilation deficiencies and prioritizing modifications before costly physical changes are made.

References

- Blocken, B. (2014). 50 years of computational wind engineering: Past, present and future. *Journal of Wind Engineering and Industrial Aerodynamics*, 129, 69–102. <https://doi.org/10.1016/j.jweia.2014.03.008>
- Buratti, C., Mariani, R., & Moretti, E. (2011). Mean age of air in a naturally ventilated office: Experimental data and simulations. *Energy and Buildings*, 43(8), 2021–2027. <https://doi.org/10.1016/j.enbuild.2011.04.015>
- Chen, Q. (1995). Comparison of different $k-\epsilon$ models for indoor air-flow computations. *Numerical Heat Transfer, Part B: Fundamentals*, 28(3), 353–369. <https://doi.org/10.1080/10407799508928838>
- Chen, Q. (2009). Ventilation performance prediction for buildings: A method overview and recent applications. *Building and Environment*, 44(4), 848–858. <https://doi.org/10.1016/j.buildenv.2008.05.025>
- Chen, Q., Lee, K., & Mazumdar, S. (2010). Ventilation performance prediction for buildings: Model assessment. *Building and Environment*, 45(2), 295–303. <https://doi.org/10.1016/j.buildenv.2009.06.008>
- Ramponi, R., & Blocken, B. (2012a). CFD simulation of cross-ventilation for a generic isolated building: Impact of computational parameters. *Building and Environment*, 53, 34–48. <https://doi.org/10.1016/j.buildenv.2012.01.004>
- Ramponi, R., & Blocken, B. (2012b). CFD simulation of cross-ventilation flow for different isolated building configurations: Validation with wind tunnel measurements and analysis of physical and numerical diffusion effects. *Journal of Wind Engineering and Industrial Aerodynamics*, 104–106, 408–418. <https://doi.org/10.1016/j.jweia.2012.02.005>
- Tominaga, Y., Mochida, A., Yoshie, R., Kataoka, H., Nozu, T., Yoshikawa, M., & Shirasawa, T. (2008). AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings. *Journal of Wind Engineering and Industrial Aerodynamics*, 96(10–11), 1749–1761. <https://doi.org/10.1016/j.jweia.2008.02.058>
- Zhai, Z. J., Zhang, Z., Zhang, W., & Chen, Q. Y. (2007). Evaluation of various turbulence models in predicting airflow and turbulence in enclosed environments by CFD: Part 1—Summary of prevalent turbulence models. *HVAC&R Research*, 13(6), 853–870. <https://doi.org/10.1080/10789669.2007.10391459>
- Zhang, Z., Zhang, W., Zhai, Z., & Chen, Q. (2007). Evaluation of various turbulence models in predicting airflow and turbulence in enclosed environments by CFD: Part 2—Comparison with experimental data from literature. *HVAC&R Research*, 13(6), 871–886. <https://doi.org/10.1080/10789669.2007.10391460>