

CFD-Based Assessment of Indoor Air Quality and Ventilation Optimization in a Printing and Dyeing Industry

Gaurav Arvind Shukla¹, Namrata D. Jariwala²

¹ Former Post Graduate student, Department of Civil Engineering, S V National Institute of Technology, Surat, Gujrat, India

² Associate Professor, Department of Civil Engineering, S V National Institute of Technology, Surat, Gujrat, India

Abstract

The dyeing or printing industry in surat is largely established before the Factories Act, 1948; hence, there is limited provision of safe and healthy working environments, proper indoor environment quality. Additionally, these industries are often constrained by budgets, so they rarely look for betterment of the work environment. Existing ventilation systems do not allow for proper air flow, resulting in poor thermal comfort for workers and a physically harsh conditions, This leads to lower work efficiency, productivity and ultimately economic and social inequality. This study is on dispersion of indoor air with the current ventilation strategy, and possible system improvements with mechanical ventilation enhancement methods. The numerical method is used to study pollutant diffusion in printing and dyeing industry. The CFD allow the simulation of two different scenarios to obtain a quantitative assessment of the effects of the proposed improvement measures.

Numerical modelling with CFD is performed using ANSYS FLUENT in order to analyze the air flow in a printing industry facility, specifically monitoring the mass fraction of Carbon dioxide, Sulphur dioxide, Nitrogen monoxide, and variations in temperature. The simulation results were analysed with both scenarios one with naturally existing ventilation system and other with proposed forced ventilation (mechanical exhaust). The best case scenario in terms of indoor air quality with improved air dispersion, minimum mass fraction of pollutants and the lowest temperature gradient, consisted in when mechanical exhaust are considered for ventilation which cause an inlet velocity of 1.75 m/sec from emission source, and a temperature of 400°K with one window and door only thus improved better thermal comfort employing better work efficiency and production.

Keywords: ANSYS Fluent, Computational Fluid Dynamics (CFD), Indoor air quality, Modelling, Numerical assessment, dyeing & printing industry.

1. Introduction

The production of printed textiles in India dates to the fourth century BC[1]. Today the textile industry is the single largest organized sector in the country, earning one-third of total national export profits[2]. In India printing industry forms a major component of manufacturing sector with a share of 2.42% by value of output. Indian printing industry employs 336857 skilled and semi-skilled workers thereby enhancing

employability for both the working class and the middle-class population. Its year on year profit grown by an average of 78.24% in the past two decades. With 10926 factories with a fixed capital of Rs 50 million lakhs is an important component of manufacturing sector hence contributes to growth in GDP of India([3].

Surat, often referred to as the "Textile City," is located in the western region of India, within the state of Gujarat. This study focuses on the small-scale dyeing and printing industry in Surat, a significant hub for synthetic fabric production, where approximately 20-25% of the city's population is employed in this sector. The city hosts around 400 dyeing and printing processing units distributed across various industrial estates.[4].Dyeing and printing involves various steps to go through employing different emissions at different steps. Typically Dyeing and printing is the batch process carried out, which depends upon the quality of raw fabrics, customer requirement and design to be printed on fabrics. In the textile processing house from the raw fabric known as "grey cloth" turn into the nal product known as "sarees" through wet processing technology, comprises various stages such as sizing, desizing, scouring, bleaching, mercerizing, dyeing, printing and finishing. All of the said operations are carried out at the temperature ranges from 100°C-180°C, so large variations in the humidity and temperature can be observed throughout the plant[5]. This industry is characterized by its labor-intensive nature and predominantly utilizes low-quality fuels, such as coal and Light Diesel Oil (LDO) , for steam generation. The combustion of coal in boilers results in the emission of various pollutants, including particulate matter (PM), Sulfur oxides (SOx), and Nitrogen oxides (NOx). Additionally, specific adhesives are employed in the dyeing process to enhance the efficacy of dyeing and printing operations. The limited space and inadequate ventilation facilities within these units further complicate the dispersion of pollutants, ultimately degrading both indoor and outdoor air quality[4]. Small scale industries often fail to provide safe and efficient design, leading to inadequate ventilation and HVAC systems that do not provide the required comfort. The industries work in restricted spaces which are not planned and not designed effectively, where natural ventilation is poor, and where construction is haphazard and full of people and buildings that do not allow expansion. In these cases, the need for CFD simulations and planning is essential. Such industries are well-established in our study area, specifically in the industrial zone of Surat. We believe that our analysis could be of use by many other industries as there is no second options left to improve pre-establish small scale industries constructed haphazardly, Although this method is somewhat costly, it is the most suitable option given the current site conditions.

Dyeing and printing industry utilises lot of resources such as manpower, water, energy specifically generated from coal and LDO. So its one of high carbon emitting industries thus needs special attention even after plant is established.

Controlling the quality of the indoor air in occupied buildings requires an understanding of the ventilation air change rate. The amount of natural air leakage into buildings was restricted by extremely tight energy-saving requirements for buildings. In many naturally ventilated buildings, decreased ventilation led to higher concentrations of indoor air pollution and extremely poor indoor air quality[6].

Most Europeans and Americans spend up to 90% of their time indoors and many spend most of their working hours in an office environment. Studies conducted by the U.S. Environmental Protection Agency (USA/EPA, 1997) and others conclude that indoor environments may sometimes show levels of pollutants that are actually higher than levels found outside (USA/ EPA, 1997). Pollutants in an indoor environment

can increase the risk of illness. Several studies by USA/EPA, states, and independent scientific panels have consistently ranked indoor air pollution as an important environmental health problem quality.[7]

In an industrial setup, it is perhaps easier to monitor and control air quality because emissions can be attributed to specific processes taking place, and the sources can be identified[8]. People who live in inadequately ventilated buildings may have a variety of health issues, which are more common in youngsters[9]. In order to estimate the indoor air exchange rate and evaluate the quality of the indoor air, these cause ventilation measurements are crucial. Computational Fluid Dynamics (CFD) is one method for simulating a building's ventilation system, which is one of many used to assess building ventilation and indoor air quality. More thorough research on local and urban air pollution over extended periods of time is necessary for air quality management with the aid of a computational fluid dynamics (CFD) model [10].

The study focuses on understanding and addressing critical aspects of indoor air quality in dyeing and printing facilities, where pollutant dispersion is a significant concern. Such knowledge is needed by engineers and architects for selecting appropriate ventilation systems and control strategies to minimize indoor contaminant exposures.[11]. By employing Computational Fluid Dynamics (CFD) for simulation and modeling, the research aims to analyze the flow patterns of emitted pollutants within the workspace and CFD model integrated the effects of airflow and turbulent characteristics associated CO₂, NO₂ and SO₂ emission sources to yield the detailed distributions of airflow, temperature, and contaminant in buildings. A particular emphasis is placed on identifying "hot spots"—areas with higher concentrations of pollutants—which can pose health risks to workers.

Additionally, the study explores how air dispersion is happening in existing ventilation scenario, and secondly how it can be improved in simulation with improvisation in existing ventilation without expanding workspace as its constraint. The simulation developed in this study can be used to understand the flow patterns of air pollutants in a room, and areas with higher concentrations of pollutants and hotspots can be predicted. Based on contours and flow streamlines created proper ventilation can be optimized and ventilation ducts and exhaust are provided at the most suitable places and thermal comfort and clean air are achieved at a minimal cost. The only limitation of the CFD approach is its complexity. As the complexity of a problem statement increases CFD results become less viable. Also, it only gives the flow pattern of a pollutant based on the boundary conditions and initialization stated which may vary with respect to time here we can only model the flow contours not the actual concentration changing after ventilation so variation in the result and actual application is highly aspected and need thorough validation of the model. Due to the extraordinary rise in computer hardware capacity over the past few years, all of this is now feasible.[12] Indoor Air Quality Computer models (IAQ computer models) have been developed and increasingly used for predicting indoor air pollutant concentrations [11]. This will help in understanding the flow pattern of air or any fluid or their mixture of different concentrations around any object and gives an approximate idea of flow around that can be validated easily by experiment basis. It reduces cost as well as time is taken for trial and error in designing. The CFD-based IAQ models solve a set of conservation equations describing the flow, energy, and contaminants in the system.

2. Materials and Methodology

In this study, we employed a comparative method to evaluate the performance of a naturally existing ventilation system. The initial simulations revealed that pollutant dispersion was inadequate, leading to a decline in indoor air quality. The indoor space was found to be constrained and congested, with no possibility for workplace expansion. Consequently, as natural ventilation is poor emissions from process machinery exhibited low velocity because of natural ventilation is insufficient as Velocity inlet at input that is stenter machine and pressure outlet at output that is windows and doors are considered, resulting in limited dispersion and adversely affecting both ventilation and indoor air quality.

In the second simulation scenario, mechanical exhausts were assumed to be integrated into the existing doors and windows (pressure outlets) for simulation based on simulation results actual execution will be initiated. This modification facilitated the extraction of emissions from the printing machines (stenter machines), thereby enhancing air dispersion within the printing hall. This improvement significantly reduced the formation of hotspots, thereby enhancing indoor air quality and contributing to a more comfortable and breathable environment by lowering the temperature.

physical problem considered

Here in this study one of the dyeing and printing factories in the Surat, Gujarat, India, industrial estate, a study on the indoor work environment was conducted. Pollutants like CO, CO₂, O₃, HCHO, Methane, Particulate matter PM₁, PM_{2.5}, PM₁₀, and Dry Bulb Temperature are monitored and their seasonal variation is measured inside a dyeing and printing industry of the Panesar area situated in surat, Gujarat, India is considered for sampling and data collection.

Based on data collected a simulation model is created using Computational Fluid Dynamics (CFD) in ANSYS Fluent software. The general flow of the CFD modelling process is represented in Fig. 1

Surveys were conducted in the industry to determine the impact of the production process on the indoor air quality on both working and non-working days. Depending on the process used, four areas within the processing unit were chosen, including the printing machine area, the stenter machine area, the jet dyeing machine area, and the looping machine region. All the data is collected by PhD scholar of SVNIT Department of civil engineering.

As all three chosen Modelling areas have identical type of planning and designing so one single representative dimension area is created and different variational parameters at inlet is applied and output is observed in this model.

Fig.1 represents a hall of dimension 3.66 m × 3.66 m × 3.66 m (In Ansys software dimensions were used in feet) is created in the Design modular of Ansys package with a single door and one window. a stenter machine is located in the middle of the hall which acts as an inlet of pollutants. dimensions of created geometry are as follows.

Table 1 Dimensions of Geometry

NAME	SIZE (meter)
Hall room	3.66 × 3.66 × 3.66
Door	2.13 × 1.22

Window	1.22×1.22
Stenter machine	0.61×0.61×0.61
Volume	48.93

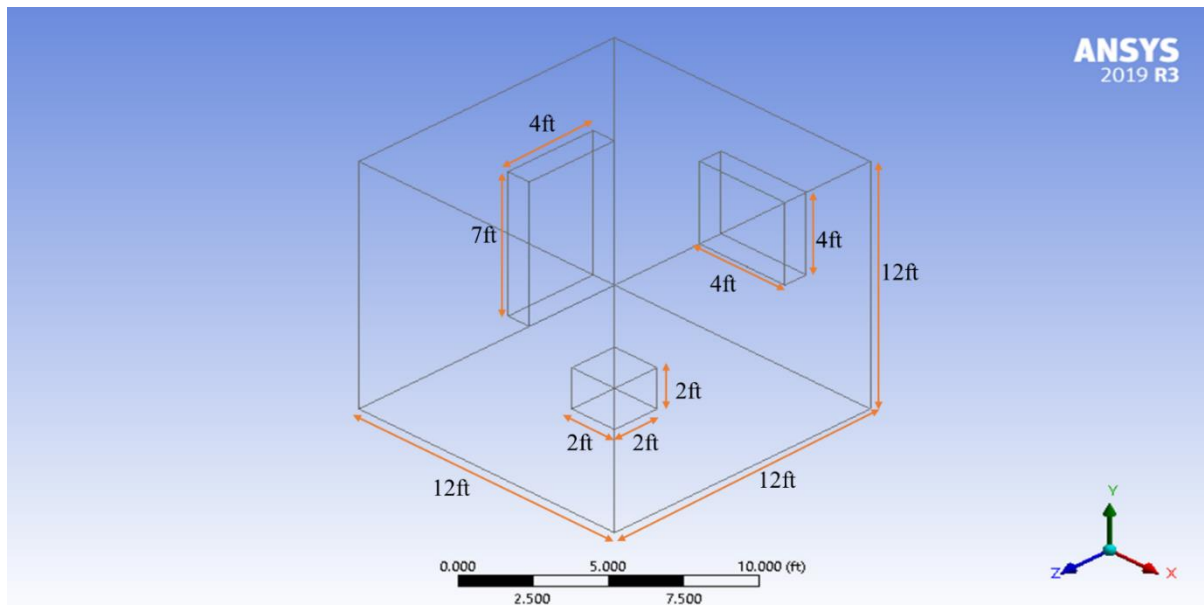


Fig 1. 3D Design of modelling area

calculations performed

People have known for a very long time how crucial fluid flow is to the sustainable use of water, the control of floodwater, agriculture irrigation, and public health. Modern society has incorporated fluid mechanics into practically every element of our daily life. Fluid flow has long been systematically studied. The Navier Stokes 19 equations, put forth by the French Claude Louis Marie Henry Navier (1785-1836) and the Irish George Gabriel Stokes (1819-1903), are the ones that are most pertinent to this primer. The conservation laws for mass and momentum are mathematically described by these equations, which extend the Euler equations for inviscid flows to include the viscous effect.[13]

$$\nabla \cdot \mathbf{u} = 0 \quad (1)$$

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = -\frac{1}{\rho} \nabla p + \nu \nabla^2 \mathbf{u} \quad (2)$$

Where

$\mathbf{u}$ = Velocity vector,

p = Pressure,

ρ = Density,

t = Time, and

ν = Kinematic viscosity of fluid.[13]

The equations describing a fluid's motion in three dimensions are solved using numerical techniques by CFD. The mathematical formulations of the physics conservation rules are shown in the partial differential equations below.[14]

Mass conservation (or continuity) equation: the mass of fluid is conserved that is

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(\rho u) = 0 \quad (3)$$

Momentum conservation equation: the rate of change of momentum equals the sum of the forces on a fluid particle (Newton's second law) that is

$$\frac{\partial \rho}{\partial t}(\rho u) + \frac{\partial}{\partial x}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + \rho f_i \quad (4)$$

Energy conservation equation: the rate of change of energy is equivalent to the sum of the rate of heat added to and the rate of work done on a fluid particle (First law of thermodynamics) that is

$$\frac{\partial}{\partial t}(\rho e) + \frac{\partial}{\partial x_j}(\rho e u_j) = -\rho \frac{\partial u_j}{\partial x_j} + \frac{\partial y}{\partial x_j} \left(k \frac{\partial T}{\partial x_j} \right) + S_e \quad (5)$$

These equations can be generalized to:

$$\frac{\partial \rho \phi}{\partial t} + \nabla \cdot (\rho \phi \tilde{V}) = \nabla \cdot (\nabla \phi) + S_\phi \quad (6)$$

or in Cartesian tensor form rather than the above vector form:

$$\frac{\partial}{\partial t}(\rho \phi) + \frac{\partial}{\partial x_j}(\rho \phi u_j) = \frac{\partial}{\partial x_j} \left(\nabla \frac{\partial \phi}{\partial x_j} \right) + S \quad (7)$$

Where,

$\frac{\partial}{\partial t}(\rho \phi)$ = Accumulation term

$\frac{\partial}{\partial x_j}(\rho \phi u_j)$ = convective term

$\frac{\partial}{\partial x_j} \left(\nabla \frac{\partial \phi}{\partial x_j} \right)$ = Diffusive term

S = source term

ϕ = flow variable

x_j = length in dimension j

t = time (s)

ρ = Fluid density (kg/m³)

u_j =Velocity (m/s) in j^{th} direction

α = Diffusive co-efficient(m^2/s)

S= Source or sink term of flow variable

Note that the mass conservation (or continuity) equation is equivalent to the general conservation equation with ϕ assigned to be unity. These equations are non-linear and cannot be solved analytically in the vast majority of practical cases [14]. The equations need to be linearized and solved over a large number of tiny control volumes (the computational mesh). The input of the geometry, boundary conditions, and fluid properties is necessary for CFD simulations to determine the flow field.

physics and boundary conditions: The physics and boundary conditions were set in ANSYS Fluent 2019R3 in a stepwise manner. The flow considered here is compressible.

In this step, general units are set up and a pressure-based solver type is selected with a steady-state condition considered and acceleration due to gravity -9.8 m/s^2 is given in y-direction.

viscous model: flow is considered to be single-phase, that is, gaseous; therefore, the multiphase is off, and the energy equation is considered a gaseous temperature variation that needs to be studied as per the viscosity, which is a concern that the $k-\epsilon$ model is selected with the standard $k-\epsilon$ model standard wall function.

k =turbulent kinetic energy=0.8

ϵ =dissipation rate=0.8[12]

species Transport model: The species transport model is selected for NO_x and SO_x monitoring in which methane-air is selected with 5 fuel species O_2 , CO_2 , H_2O , N_2 , and CH_4 . And volumetric reactions as given below.

material selection: In the Dyeing industry Indian coal is used as a fuel so thus based on data collected and raw material used Inserting material taken into consideration for our analysis namely. Materials include methane air mixture containing methane, nitrogen, oxygen, carbon dioxide and water vapours fluid material as air.

boundary conditions (bc):

input parameters: The velocity inlet boundary condition is applied at the stenter machine inlet, while two pressure outlets are defined at the openings—a door and a window. At these inlets, the specified velocity magnitude is 1.75 m/s with a turbulent intensity of 5%. where as at pressure outlet considered backflow turbulent intensity of 5%.

2 different boundary conditions are considered that is for a given inlet velocity temperature is used one at ambient temperature and one at a higher temperature, V_1 at lower inlet velocity being velocity when natural ventilation conditions are considered and V_2 , when modified ventilation (mechanical Exhaust) was assumed the same pair of temperature is used as shown in Table 1 And based on their results comparisons and error possibility conclusions are drawn.

Table 2. Velocity and Temperature combinations

Velocity(m/s)	Ambient temperature(K)	Higher temperature(K)
V1=0.82	300	400
V2=1.75	300	400

Output solver: Parameters for solution methods, the pressure-velocity coupling is used with first-order turbulent kinetic upwind and Pressure second order, velocity second order. The table 3 concludes detailed method adopted for solving the equations.

Table 2 Details of schemes and their solutions

SCHEME	SOLUTION METHOD
Pressure-Velocity coupling Scheme	Coupled
Pressure	Second order
Momentum	Second-order upwind
Turbulent kinetic energy(k)	First-order upwind
Turbulent dissipation rate(ϵ)	First-order upwind
Energy equation	Second-order upwind

After finalizing, the solution method initialization has to be provided with hybrid initialization, calculations are performed, and iterations for calculations are performed until the result converges.

computational domain and computational details

An unstructured tetrahedral grid was chosen for mesh generation to ensure a balance between computational efficiency and accuracy. The grid was configured with a maximum element size of 257.87 mm, which facilitated the creation of a high-quality mesh with excellent orthogonality. The minimum orthogonal quality of the mesh was measured at 0.13487 mm, demonstrating its precision and suitability for accurate numerical simulations. The generated mesh comprised a total of 586,543 nodes and 425,454 elements, providing a robust framework for detailed analysis and reliable computational results. This configuration supports enhanced spatial resolution and the ability to capture intricate geometrical features and complex flow dynamics effectively.

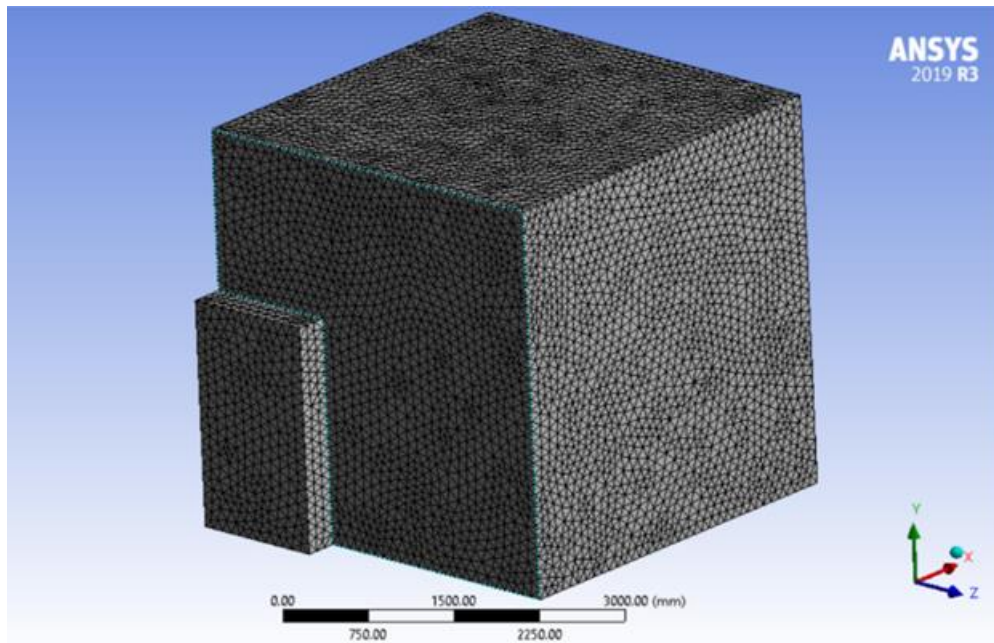


Fig 2. Mesh Statistics

3. Results and Discussion

Here obtained results from fluent solver is being thoroughly discussed and changes in mass fraction of different pollutants and temperature variations are discussed for both scenarios. Every parameter is discussed and analysed for natural as well as improve version bifurcated by four different cases with various temperature velocity combinations. Table 4 lists the two different scenarios.

Table 4. Different combinations of cases.

CASE	VELOCITY(m/sec)	TEMPERATURE (K)
Natural ventilation without improvement		
1	0.82	400
Forced ventilation with mechanical exhaust		
2	1.75	400

4. Discussion

temperature dispersion: In fig 4 simulation modelling from fluent solver clearly indicates that, the heat map for case 2 showed the better performance when forced ventilation is applied through. There is significant reduction in temperature in this case which is obtained through the use of mechanical exhaust as opposed to case 1, which only relies on natural ventilation and shows significant drop in temperature at outlet which also leads towards increasing thermal comfort at indoor environment.

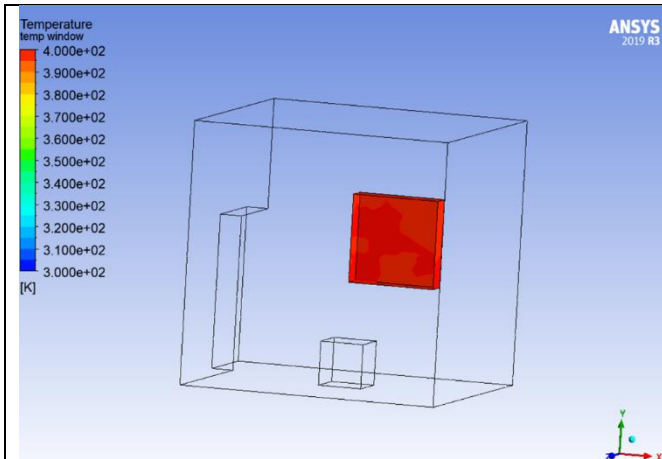


Fig 3. Case 1 Natural ventilation with original temperature

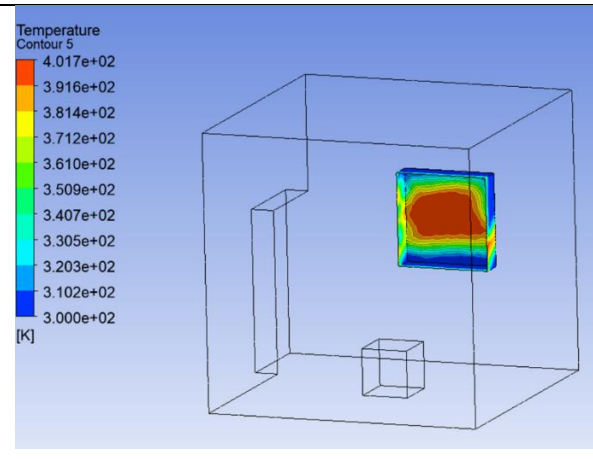


Fig 4. Case 2 Natural ventilation with ambient temperature

no2 dispersion:

The best results were observed in Case 2 for Forced ventilation scenario, where mechanical ventilation is employed on doors and windows. In this scenario, the NO₂ mass fraction showed a significant reduction, decreasing from 0.017 to 8.07×10^{-25} . Whereas when compare with existing ventilation pollutant dispersion is almost negligible as shown in fig 5. Figure 5 illustrates the volume rendering of the mass fraction distribution within a room with forced ventilation.

In contrast, the case 1 did not exhibit a significant reduction in NO₂ mass fraction, as depicted in fig. 6.

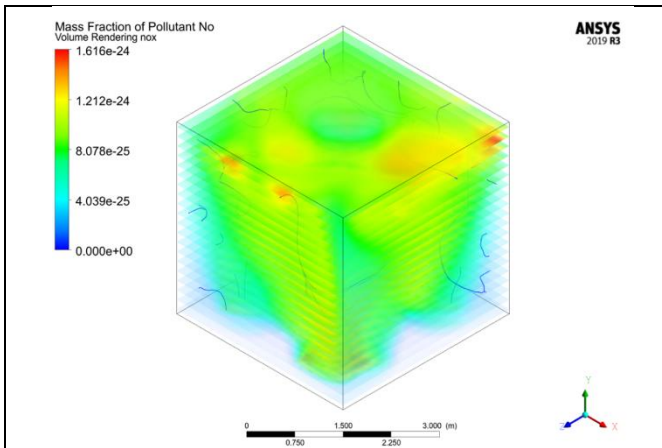


Fig 5. Case 2 NO₂ volume rendering with forced ventilation

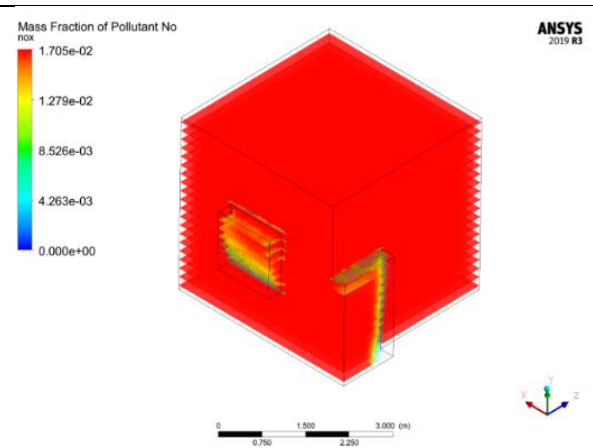


Fig 6. Case 1 NO₂ volume rendering with natural ventilation

so₂ dispersion: The dispersion of sulphur dioxide is analysed through the pollutant dispersion streamline pattern of the particles, revealing distinct patterns for each case. From figure 7 indicates that in Case 1 when only natural ventilation is available, the movement was limited to the corners and edges of the room causing poor ventilation. Conversely with forced ventilation pollutant dispersion is predominantly directed toward the window and door with, indicating efficient dispersion pathways as depicted in figure 8 thus indoor air quality by improving pollutant dispersion movement.

Velocity Gradient increased from 0.1524 m^{-1} to 0.1596 m^{-1} represented in Fig 7 and Fig 8 respectively.

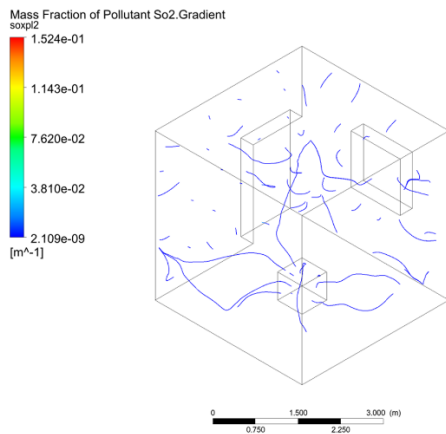


Figure 7 Case 1 SO₂ pollutant dispersion streamline with natural ventilation

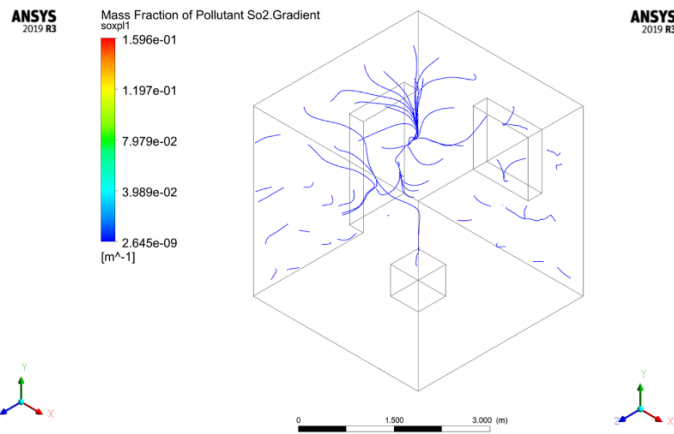


Figure 8 Case 2 SO₂ pollutant dispersion streamline with forced ventilation

co₂ dispersion: A similar pattern was observed for the carbon dioxide dispersion. With existing ventilation system the particles tended to stagnate within the room, as illustrated in Figure 9 However, when improvisation in ventilation applied, the particle movement was more pronounced near the outlets, facilitating effective dispersion as depicted in figure 10.

By representing increased ventilation gradient from 6.366 m^{-1} to 7.516 m^{-1} . Represented in Fig 9 and Fig 10 Respectively.

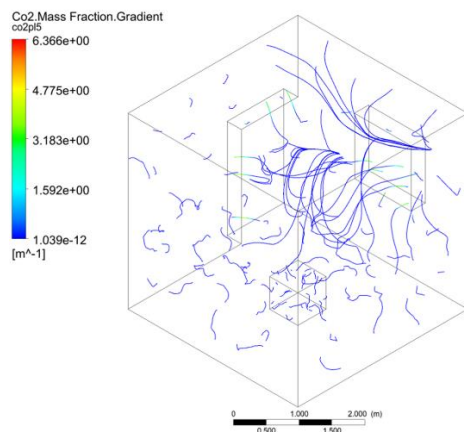


Figure 9 case 1 CO₂ pollutant dispersion streamline with natural ventilation

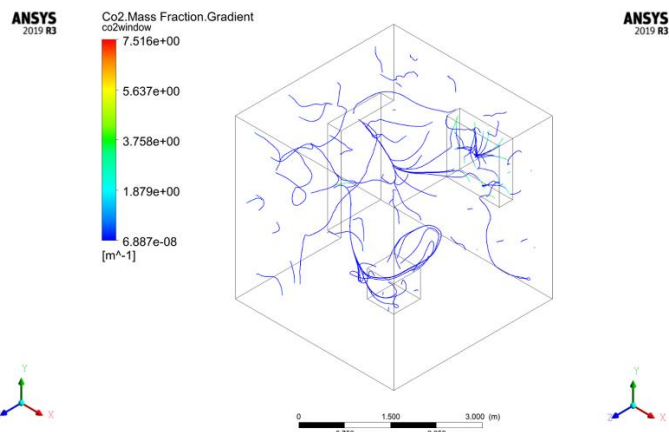


Figure 10 case 2 CO₂ pollutant dispersion streamline with forced ventilation

spatial scaling of results

For the CFD simulations, a representative domain of $3.66 \text{ m} \times 3.66 \text{ m} \times 3.66 \text{ m}$ was selected. Since the characteristics did not vary significantly across the space, results from the representative domain were spatially interpolated to estimate values in the any space. The interpolated values for the entire space are shown in Table 5 and Table 6, respectively as an example.

Table 5 Simulated parameters on representative space

Representative space	$3.66 \text{ m} \times 3.66 \text{ m} \times 3.66 \text{ m}$	
	Simulated Parameters	
Parameter	Initial	Final
Temperature	400 k	300 k
NO ₂ (Volumetric Rendering-Unit less)	0.017	8.017E-25
SO ₂ (streamlined flow representation-velocity gradient)	0.1524 m^{-1}	0.1596 m^{-1}
CO ₂ (streamlined flow representation-velocity gradient)	6.366 m^{-1}	7.516 m^{-1}

Table 6 Parameters calculated on actual space

Actual space	$15.24 \text{ m} \times 15.24 \text{ m} \times 15.24 \text{ m}$	
	Calculated Parameters	
Parameter	Initial	Final
Temperature	1667 K	1250 K
NO ₂ (Volumetric Rendering-Unit less)	0.00708	3.340E-26
SO ₂ (streamlined flow representation-velocity gradient)	0.635 m^{-1}	0.665 m^{-1}
CO ₂ (streamlined flow representation-velocity gradient)	26.525 m^{-1}	31.3166 m^{-1}

5. Conclusion

The goal of this study was to examine the Computational Fluid Dynamics modelling of the dispersion of a range of pollutants from the printing and dyeing industry. This study sought to extrapolate the benefits of improving ventilation without actually applying this in real space, by observing how indoor air quality IAQ, specifically CO₂, SO₂, NO₂ and temperature, responded. This is efficient and much less costly than trial and error. We feel the results provide a substantial improvement in Indoor Air Quality and thus a more comfortable environment for employees. It is demonstrated how a mechanical exit allows for the dispersion of air in tighter settings.

Given that expansion is not possible, our conclusion is that Indoor Air Quality is severely compromised in buildings with insufficient indoor space and with windows that do not allow for adequate ventilation. Better exhaust via mechanical pumps or exhausters is an option to improve ventilation. These can be tested dynamically, prior to the actual implementation of any improvement, through the CFD methodology, preventing the trial-and error approach which can be costly and time consuming. This means the best solution can be chosen.

By identifying hot spots and points of pollutant accumulation, we can implement mechanical ventilation systems to enhance air dispersion effectively.

Future Scope

- For future research, the most relevant course of action would be to broaden this study to encompass additional industrial sites with diverse conditions and air circulation systems. A simulation of an enhanced ventilation system should be conducted, and Computational Fluid Dynamics (CFD) could be combined with Machine Learning (ML) or Deep Learning (DL) models to facilitate more data-driven decision-making.

Additionally, there is curiosity about the impact of indoor activities on PM_1 , $PM_{2.5}$, and PM_{10} emissions, an aspect not addressed in the current study. This is particularly significant due to the considerable effect of particulate matter on human health.

Nomenclature

u = Velocity vector

p = Pressure

ρ = Density

t = Time

ν = Kinematic viscosity of fluid

ϕ = flow variable

x_j = length in dimension j

t = time (s)

ρ = Fluid density (kg/m^3)

u_j = Velocity (m/s) in j^{th} direction

Γ = Diffusive co-efficient (m^2/s)

ϵ = dissipation rate = 0.8

S = Source or sink term of flow variable

k = turbulent kinetic energy = 0.8

References

1. "Goetz P.W. (Ed.). 1987. The New Encyclopaedia Britannica. 15th ed. Encyclopaedia Britannica, Inc., Chicago, IL, U.S.A."
2. "Husain I, Vaidya V. K, Husain J, Vaidya R, Sharma C. S. Groundwater Pollution by Textile Industry. Orient J Chem 2003;19(3)."

3. “Annual Survey of Industries. (2018, October 9). ASI Summary results. (M. o. Implementation, Producer) Retrieved October 9, 2018, from Ministry of Statistics and Programme Implementation: <http://mospi.nic.in/asi-summary-results>”.
4. “Jariwala D, Christian R. Development of indoor work environmental air quality index for a dyeing and printing industry. *Journal of Air Pollution and Health*. 2023;8(2): 205-226.”.
5. D. Jariwala, R. A. Christian, and N. D. Jariwala, “Prediction of Health Risk Potential Using Fuzzy Rule Base System for Indoor Work Environmental Condition in Dyeing and Printing House,” Jun. 10, 2020. doi: 10.21203/rs.3.rs-33758/v1.
6. “Dimitroulopoulou, 2012a, 2012b; Griffiths & Eftekhari, 2008; Santamouris et al., 2008”.
7. “**U.S. Environmental Protection Agency**, An Office Building Occupant's Guide to Indoor Air Quality, EPA-402-K-97-003. Washington, DC, USA: Office of Air and Radiation, Indoor Environments Division, Oct. 1997.”.
- Fanger P.O. 2006. What is IAQ? Indoor Air, 16(5): 328–334**
8. Bekö G., Lund T., Nors F., Toftum J. and Clausen G. 2010. Ventilation rates in the bedrooms of 500 Danish children. *Build. Environ.*, 45(10): 2289–2295., doi: 10.1016/j.buildenv.2010.04.014.
9. Sanchez B., et al. 2017. Modelling NOx concentrations through CFD-RANS in an urban hot-spot using high resolution traffic emissions and meteorology from a mesoscale model. *Atmos. Environ.*, 163: 155–165., doi: 10.1016/j.atmosenv.2017.05.022.
10. Panagopoulos I.K., Karayannis A.N., Kassomenos P. and Aravossis K. 2011. A CFD simulation study of VOC and formaldehyde indoor air pollution dispersion in an apartment as part of an indoor pollution management plan. *Aerosol Air Qual. Res.*, 11(6): 758–762., Nov. 2011, doi: 10.4209/aaqr.2010.11.0092.
11. Teodosiu C., Ilie V. and Teodosiu R. 2014. Appropriate CFD turbulence model for improving indoor air quality of ventilated spaces. *Math. Model. Civ. Eng.*, 10(4): 28–42., doi: 10.2478/mmce-2014-0020.
12. **X. Liu and Z. J. Zhai**, Applications in Water, Wastewater, and Environmental Engineering Modeling. 2012..”.
13. Y. Wang, T. D. Waite, and G. L. Leslie, “Computational fluid dynamics (CFD) analysis of membrane reactors: modelling of membrane bioreactors for municipal wastewater treatment,” in *Handbook of Membrane Reactors*, Elsevier, 2013, pp. 532–568. doi: 10.1533/9780857097330.3.532.