Ventilation Assessments of e-Waste Recycling Facilities during Normal and Abnormal Operating Conditions

NEIHAD HUSSEN AL-KHALIDY
CFD, Wind and Energy,
SLR Consulting,
Tenancy 202 Submarine School, Sub Base Platypus,
120 High Street, North Sydney 2060,
AUSTRALIA

Abstract: - Waste facilities deal with dust, odours, and harmful gases. Designing an effective ventilation system for e-waste recycling facilities presents unique challenges due to the complexity of waste materials and potential emissions of hazardous air pollutants. This paper explores key considerations in designing an effective ventilation solution for e-waste facilities featured in an industrial showcase project in NSW, Australia. The facility comprises a Batch Rotary Pyrolysis (BRP) plant area to process waste electrical and electronic equipment, generating a flammable syngas during thermal treatment. Under abnormal conditions, this syngas may be released into the building, posing a fire hazard if ignited by unprotected electrical equipment near BRP machines or attached gas burners. To mitigate this risk, ventilation serves as the first line of defence against the accumulation of explosive gas concentrations, in alignment with AS1482-2013 standards. This study employs Computational Fluid Dynamics (CFD) analysis integrated with the site-specific wind rose to evaluate ventilation effectiveness, creating a detailed 3D model of the proposed facility, its surrounding structures, and natural ventilation components. A localised 10m-height reference wind rose was developed using The Air Pollution Model (TAPM) and CALMET diagnostic meteorological modelling software to provide accurate wind conditions for the project site. The CFD analysis delivers detailed insights, including airspeed distributions and predicted air exchange rates, highlighting hotspot areas with inadequate ventilation. Furthermore, the model facilitates comparative evaluations against regulatory standards by simulating concentrations of key gases (CO, SO_2 , and NO_2) under abnormal gas leak conditions, ensuring that both safety and compliance are thoroughly upheld. The study offers comprehensive design input to refine natural ventilation systems, optimise opening sizes, and integrate mechanical ventilation where natural airflow is insufficient due to site orientation. This research demonstrates that an effective ventilation strategy for e-waste facilities can be developed when CFD modelling is incorporated early in the concept design stage. The proposed tool also has multiple downstream applications, including air quality and odour examination, thermal comfort assessment, fire risk evaluation, and smoke dispersion analysis.

Key-Words: - e-Waste Recycling Facility, Natural and Mixed Mode Ventilation, CFD, Gas Leak, Batch Rotary Pyrolysis.

Received: April 2, 2025. Revised: July 13, 2025. Accepted: August 28, 2025. Published: October 3, 2025.

1 Introduction

Managing the Work Environment and Facilities, [1], and the Workplace Health, Safety and Welfare (WHSW) Regulations, [2], require that the employer do what is needed to make sure that every enclosed workplace is adequately ventilated. This includes providing a sufficient supply of fresh or purified air to maintain a safe, healthy, and comfortable environment for all occupants.

General ventilation, or dilution ventilation, refers to the movement of air into and out of a workspace. It can be achieved through:

- Natural ventilation which relies on wind pressure and temperature differences to circulate fresh air throughout a building. However, this method is typically not fully controllable.
- Forced or mechanical ventilation, which utilizes mechanical supply and/or extraction systems to regulate airflow, ensuring a controlled and consistent exchange of fresh air.

When implementing general ventilation to manage exposure to contaminants generated by industrial processes, it is essential to ensure the following conditions are met:

- The contamination rate must be low enough to be effectively diluted by the available airflow.
- The contaminant must have a low toxicity.
- The contaminant must be produced at a consistent rate.
- Workers should be positioned perpendicular (90°) to the airflow to ensure contaminated air is not directed toward their faces, minimizing exposure risks.
- The contaminant must be generated in low concentrations and effectively controlled within the assigned Occupational Exposure Limit (OEL), [3].
- Airflow must not interfere with the operation of other extraction systems, such as fume cupboards or local exhaust ventilation.

A natural ventilation system is recommended to harness wind-induced airflow throughout the year, thereby minimising energy costs. The National Construction Code (NCC) of Australia, [4], mandates that natural ventilation openings must constitute at least 5% of the floor area of a ventilated space. This ensures that habitable rooms receive sufficient fresh air to maintain indoor air quality and comfort without relying on mechanical systems. In practical terms, this requirement means that for every 100 m^2 of floor area, there must be at least 5 m^2 of openable windows or other openings to facilitate adequate air circulation. However, meeting this 5% threshold can be challenging for industrial buildings due to factors such as large floor areas, internal layouts, and structural constraints.

The WHSW-1992 Guidance, [2], states that the fresh air supply rate to the workplace should not normally fall below 5 to 8 litres per second, per occupant. The guidance provides no criteria for the required ventilation rates in industrial and process buildings. The guidance advises the reader to consult an occupational hygienist and/or a ventilation engineer or other competent person to decide on the correct ventilation rate and design.

In the design of natural ventilation systems, several factors must be carefully evaluated to ensure optimal performance, [5]:

- Local wind climate exposure.
- Topographical Wind Amplification
- Built Environment Interactions
- Facade-Induced Pressure Gradients

Designing an effective ventilation system for e-waste recycling facilities presents unique challenges due to the complexity of waste materials and potential emissions of hazardous air pollutants. Under abnormal operating conditions, syngas may be released into the facility, increasing the risk of fire and explosion. To mitigate this hazard, ventilation serves as the first line of defence against the accumulation of explosive gas concentrations. The ventilation design must align with AS1482-2013 standards, ensuring a safe and controlled indoor environment, [6].

With today's computer capabilities, ventilation simulations in industrial facilities can be effectively assessed utilizing Computational Fluid Dynamics (CFD). Its applications range from optimising natural ventilation and thermal comfort to solving complex environmental challenges in the building sector. By integrating CFD simulations, architects and engineers can enhance energy efficiency, improve air circulation, and minimise environmental impacts—all while ensuring regulatory compliance and sustainability, [7].

CFD has been extensively employed to evaluate the performance of various ventilation systems, [8]. This study explored a range of operating conditions using a simplified three-dimensional model to assess system effectiveness.

A series of CFD simulations and physical model experiments were conducted to identify the optimal ventilation strategy for an isolated refuse disposal facility, [9]. In this investigation, the ventilation system was represented as a one-dimensional model. Findings indicated that both air curtains and wind shields enhance airflow, offering improved protection against dust and odors within the workspace.

A systematic review encompassing 173 studies on CFD and ventilation has been presented in [10]. This review highlights key challenges in the application of CFD to building ventilation, particularly in the areas of turbulence modelling, numerical approximation techniques, and the treatment of boundary conditions.

CFD has revolutionized the design and optimization of waste management facilities in numerous ways. Through CFD analysis, natural ventilation in aerobic composting systems has been significantly improved over conventional designs, [11]. This powerful tool has also been applied to model the hydrodynamics of chemical reactors used to dissolve metals from electronic waste, offering enhanced efficiency in resource recovery, [12]. Furthermore, selected case studies have demonstrated CFD's impact on optimising wastewater treatment plant performance, [13].

CFD has been widely utilized in the analysis of plastic waste, biomass, and mixed solid waste

pyrolysis, helping refine reactor designs for better process efficiency, [14], [15], [16].

Several case studies are showcased, including a CFD analysis of a local fume capture hood to validate design modifications, an evaluation of a canopy hood to identify optimal ventilation improvements, and an in-depth investigation into troubleshooting and redesigning the ventilation system of a large furnace building, [17].

An analysis of thermal comfort and enhanced ventilation strategies aimed at improving the working environment within a waste incineration facility is presented in [18].

The ventilation design for e-waste facilities, especially considering both normal and abnormal operating conditions, is a complex challenge that hasn't been thoroughly explored in published literature to our knowledge. In contributing to existing literature, the current study provides valuable insights into optimising airflow distribution, contamination control, and emergency ventilation strategies for abnormal operating conditions and explores key considerations in designing an effective ventilation solution for e-waste facilities featured in an industrial showcase project in NSW, Australia.

This study recommends 3 to 6 Air Changes per Hour (ACPH) for warehouses and stores. The recommended number of ACPH is 8 to 10 for warehouses with heat and fume build-up issues.

2 Problem Formulation

The CFD model addresses the fundamental governing equations of fluid flow, namely the continuity equation, the momentum equations, and the species concentration equation. For a steady-state scenario, these equations are expressed as follows:

$$\frac{\partial}{\partial x_i}(\rho u_i) = 0 \tag{1}$$

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

$$\frac{\partial}{\partial x_j}(C u_i) = \frac{\partial}{\partial x_i}\left(\frac{v_t}{\sigma}\frac{\partial C}{\partial x_j}\right) \tag{3}$$

In the above, ρ is the density, u is the velocity, p is the static pressure, ρg and F are the gravitational body and external body forces, ι_{ij} is the stress tensor. C is the pollutant concentration, and v is the kinematic viscosity.

A Realizable k-ε model is adopted to estimate the wind velocity in terms of three directions, pressure profile, and turbulence parameters. The modelled steady state transport equations in the Realizable k-ε, [19], [20], are:

$$\frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j}\left[(\mu + \frac{\mu_t}{\sigma_k})\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\varepsilon - Y_M + S_k$$

and

$$\frac{\partial}{\partial x_j}(\rho\varepsilon u_j) = \frac{\partial}{\partial x_j}\left[(\mu + \frac{\mu_t}{\sigma_\varepsilon})\frac{\partial \varepsilon}{\partial x_j}\right] + \rho C_1 S\varepsilon - \rho C^2 \frac{\varepsilon^2}{k+\sqrt{v\varepsilon}} + C_{1\varepsilon}\frac{\varepsilon}{k}C_{3\varepsilon}G_b + S_\varepsilon$$

In these equations, G_K denotes the production of turbulence kinetic energy arising from mean velocity gradients, G_b signifies the generation of turbulence kinetic energy driven by buoyancy, and Y_M reflects the impact of fluctuating dilatation in compressible turbulence on the total dissipation rate, C_2 and $C_1\varepsilon$ are constants, and σ_k and $\sigma\varepsilon$ are the turbulent Prandtl numbers for K and ε, respectively. S are user-defined source terms if required.

The energy model is solved when the temperature distribution inside the facility is required.

It is also worth noting that the model transitions to a transient simulation when predicting flue gas concentrations under abnormal operating conditions. Refer to Section 4.6 for further details.

3 Methodology

Ventilation studies integrate statistical meteorological data with airflow speed and building-specific parameters to assess wind modelling impacts. The proposed methodology for quantitative assessment follows these key steps:

- Develop a localised weather windrose tailored to the site using The Air Pollution Model (TAPM) and CALMET diagnostic meteorological modelling software
- Construct a discrete model of the proposed development, incorporating relevant openings to optimise airflow distribution within the building.
- Account for environmental factors, including surrounding buildings, terrain, and topography, that influence ventilation dynamics.
- Predict wind speeds and air exchange rates

across key areas, such as material sorting zones, BRA rooms, and designated points of interest.
- Benchmark results against industry-accepted design criteria and applicable ventilation standards specific to the project site.
- Identify ventilation inefficiencies, addressing ventilation gaps where natural airflow is inadequate, and propose solutions to meet design requirements.
- Assess risk scenarios, particularly abnormal leak conditions relevant to the e-waste facility.
- Explore mechanical ventilation alternatives if natural airflow improvements are constrained by site limitations.
- Refine recommendations by re-modelling key adjustments and verifying compliance with established design criteria.

This structured methodology ensures the assessed site meets ventilation requirements and maintains appropriate air dilution, even under abnormal leak conditions.

4 Case Study

The showcase project is located in New South Wales (NSW), Australia. It comprises the development of a sorting facility to receive, sort, disassemble, and separate e-waste. The separated material streams would then be transported off-site for further processing or sale, in both Australia and overseas. Plastic waste such as printed circuit boards (PCBs), mobile phones, and CPUs would be stored on site to be used as feedstock for the proponent's Batch Rotary Pyrolysis (BRP) plant.

Pyrolysis is the thermal degradation of plastic waste at high temperatures (300–900°C), in the absence of oxygen, to produce liquid oil.

In summary, the facility comprises the following components:
- Loading and unloading of e-waste and subsequent process waste products;
- Short-term materials storage before sorting;
- Materials sorting and dismantling via hand picking;
- Data cleansing;
- Product refresh and resale;
- Materials packaging for on-transport;
- Shredding of small products, such as modems;
- Removal of aluminum and glass from solar panels ready for resale; and
- Storage of high-value feedstock to be processed through the BRP.

4.1 Local Wind Climate

A combination of the TAPM and CALMET models has been used to develop a site-representative meteorological dataset for the calendar year based on analysis of five years of meteorological data recorded at the closest Bureau of Meteorology (BoM) station, located approximately 7 km southwest of the Site.

Wind conditions predicted to occur on the Site are presented as annual and seasonal wind roses in Figure 1. The wind roses illustrate the frequency and intensity of wind based on directional origin.

In each wind rose diagram, the topmost bar denotes winds originating from the north, commonly referred to as northerly winds. The length of each bar reflects how frequently winds blow from that direction, while the varying widths of its segments correspond to different wind speed categories. The narrowest sections indicate the lightest breezes, with progressively wider bands representing stronger wind intensities.

A review of the associated seasonal wind roses shows that typically:
- In summer, light winds are predicted to blow predominantly from the southeast quadrant. A very low frequency of winds is predicted to blow from other directions. Calm winds are predicted to occur 0.2% of the time during summer.
- In autumn, light winds are predicted to blow predominantly from the south-southwest direction with a lower frequency of winds blowing from the southeastern quadrant. Calm winds are predicted to occur approximately 2% of the time during autumn.
- In winter, winds from the southwestern quadrant are predicted to be predominant, with a low frequency of winds predicted to be blowing from other directions. Calm winds are predicted to occur approximately 2% of the time during winter.
- In spring, light winds blowing from the southeastern and southwestern quadrants are predicted.

Figure 1 indicates that winds experienced in the study area are predicted to be predominantly light (between 1.5 m/s and 5.5 m/s) blowing from the southeast and southwest quadrants, with a lower frequency of winds blowing from the northern directions.
- Wind speeds higher than 1.5 m/s occur approximately 70% of the time.

- Calm wind (<0.5 m/s) conditions were predicted to occur approximately 1.3% of the time.

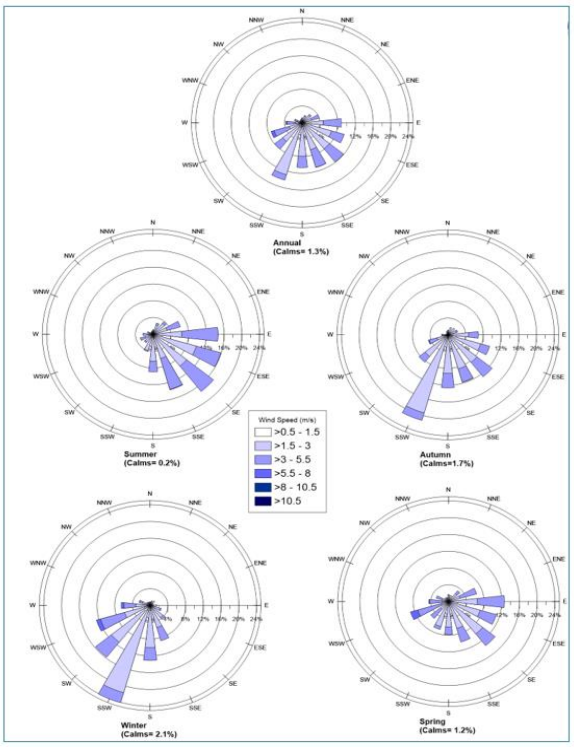

Fig. 1: Developed Seasonal Wind Rose for the Project Site
Source: created by the author

4.2 Modelling Configuration

To account for the influence of the immediate surroundings, the CFD model incorporates all neighboring buildings and the local topography of the site. An overview of the full model is presented in Figure 2, while the proposed e-waste recycling facility is illustrated in Figure 3.

The facility design includes open mesh structures between various components; however, these meshes were not modeled, as their impact on ventilation performance was deemed negligible.

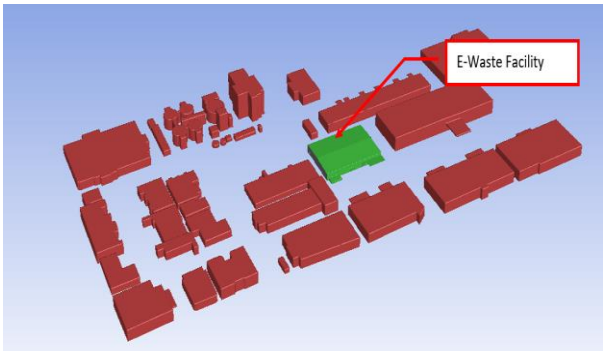

Fig. 2: 3D Model of the Proposed Development and Surrounds for CFD Model
Source: created by the author

The ventilation rates were predicted at the area of interest for a wide range of wind conditions, and sample results are presented in the current study for the following two critical wind directions:

- Prevailing south-southwest winds
  - The e-waste facility is partially shielded by the nearby buildings for this wind direction.
- Prevailing southeast summer winds
  - Southeast winds create a pressure difference between the roller door (Refer Figure 3B) and the proposed louvres (Refer Figure 3A), e.g., the wind pressure at the southeast-facing roller doors is greater than the pressure at the southwest-facing louvres; airflow will be pushed through the e-waste facility. The e-waste facility is also shielded by the nearby building for the southeast wind scenario.

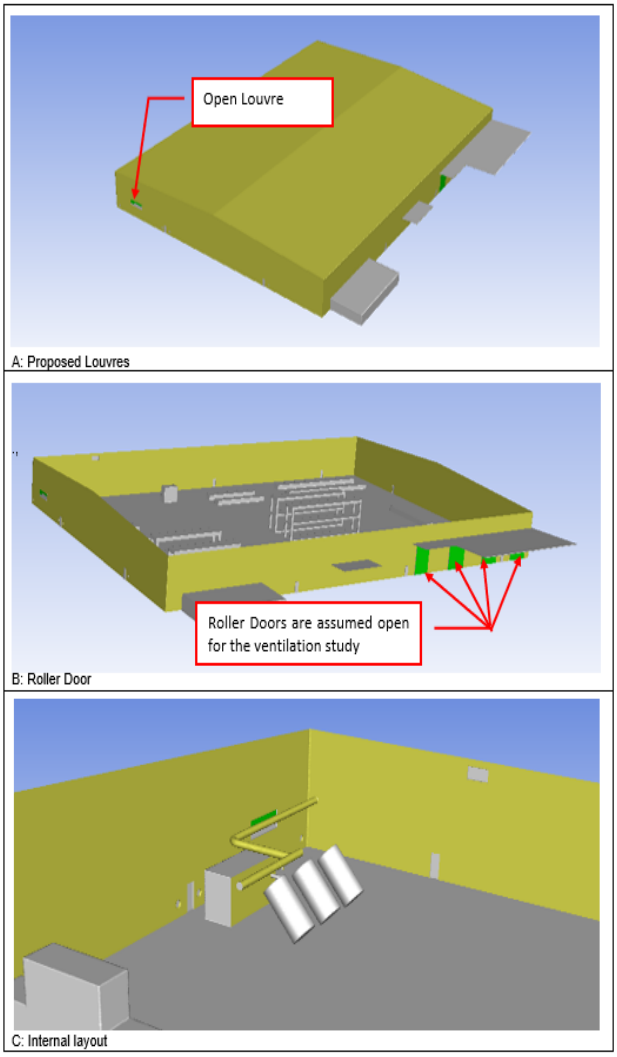

Fig. 3: 3D Model of the Project Site Including Proposed Openings for Natural Ventilation Study
Source: created by the author

4.2.1 Boundary Conditions

4.2.1.1 Wind Conditions

Based on local climate wind modelling, the average wind speed for the project Site (taking into account all wind directions) is 2.2 m/s.

A light breeze of 1.5 m/s or higher occurs approximately 70% of the time.

At the upwind free boundary inlet, velocity profiles were derived from BoM data and the Australian Wind Code AS1170.2. At the downwind and upper free boundaries, constant pressure boundary conditions were applied.

The following light wind conditions were modelled:

- Wind Speed = 1.5 m/s at 10 m above ground (1.5 m/s was chosen to represent the light breeze condition).
- Ventilation rates increase with increasing wind speeds approaching.

4.2.1.2 Other Boundary Conditions

The following additional boundary conditions were used:

- Turbulence quantities (kinetic energy and dissipation rate) were calculated from empirical relationships.
- A wall function data group was used to avoid using a very fine mesh near the wall and improve the turbulent flow simulation.

4.2.2 Discretisation

The software package utilised in the current CFD analysis is the commercially available code ANSYS-FLUENT. The CFD model solves continuity, momentum, and species concentration equations in the computational domain to predict the airflow inside and around the proposed development.

- Based on a mesh sensitivity assessment, 33,595,372 mixed cells were used to cover the computational domain.
- Realizable k-epsilon (rke) turbulence model was used for all analysed cases. The model is capable of capturing stagnation and high-pressure gradients in critical regions of interest.
- A Species Transport Equation is used to model the flue gas dispersion.
- A second-order numerical scheme was used for the discretization of pressure and momentum to obtain more accurate results.
- An iterative procedure was used to estimate the air velocity in terms of three directions, pressure profile, and turbulence parameters.
- For the pressure velocity coupling, a global solver based on the SIMPLE algorithm was employed.

The normalised residuals of continuity x-, y-and z-velocity, species concentration, k, and epsilon were reduced between four and seven orders of magnitude, demonstrating a valid numerical solution.

4.3 CFD Results and Discussions – Normal Operational Conditions

Sample results are provided for the following scenarios:

- Scenario 1 - South-southwest light wind
  - wind speed = 1.5 m/s at 10 m above ground
- Scenario 2 - Southeast light wind
  - wind speed = 1.5 m/s at 10 m above ground

4.3.1 Scenario 1

Figure 4 (Appendix) illustrates the airflow velocities within a two-dimensional (2D) section at a height of 1.5 meters above ground, approximately chest level. Velocity vectors are displayed on a color-coded scale ranging from 0 to 2.0 m/s, where deep blue signifies still air at 0 m/s, and red indicates the highest wind speed.

The CFD model effectively captures intricate fluid flow characteristics, revealing how wind approaches the site from the south-southwest, as dictated by the given boundary conditions. Upon encountering the built environment, the wind is funneled between structures, accelerating near their edges while stagnating and recirculating in the wake of the buildings.

Figure 5 (Appendix) depicts airflow at the openings. The following key conclusions can be drawn from its analysis:

- The wind pressure at the southwest louvres is greater than the pressure at the roller doors; airflow is therefore pushed through the e-waste facility.
- The inlet mass flow rate into the e-waste facility is 3.98 kg/s. Facilitating an air change rate of 0.17.

Overall, ventilation within the facility is inadequate for this wind direction, primarily due to shielding effects from surrounding buildings and the restricted size of the openings. As illustrated by the velocity magnitude contours in Figure 6 (Appendix), several low-velocity zones are present throughout the facility.

4.3.2 Scenario 2

The results of this scenario are illustrated in Appendix in Figure 7 and Figure 8, from which the following conclusions can be drawn:

- As dictated by the boundary conditions, wind approaches the site from the southeast. It is then channeled between buildings, accelerating near the edges while stagnating and recirculating behind the structures (Figure 7, Appendix).
- The wind pressure at the southeast roller doors exceeds that at the louvres, resulting in airflow being directed through the e-waste facility (Figure 8, Appendix).

The inlet mass flow rate into the e-waste facility is 17.8 kg/s, enabling an air change rate of 0.77 ACPH. Additionally, wind partially short-circuits through the roller doors.

4.4 CFD Results and Discussions – Abnormal Operational Conditions

The results in Section 4.3 indicate that the natural ventilation system of the e-waste facility is inadequate under light wind conditions, as observed at the project site during the normal operating conditions.

To address this, further assessments have been conducted to evaluate facility performance under abnormal conditions. The primary objective is to modify the ventilation system based on findings from both normal and abnormal scenarios.

The Batch Rotary Pyrolysis (BRP) plant conducts thermal treatment of waste electrical and electronic equipment, generating a flammable syngas that is directed to a thermal oxidizer unit.

Under abnormal operating conditions, syngas may be released into the plant building, creating a significant fire hazard. This risk is elevated if the released syngas is exposed to ignition sources such as unprotected electrical equipment near the BRP machines or the gas burners attached to them.

A possible flue gas leak from the BRA has been modelled with the following parameters.

- Gas density = 1.125 kg/m³
- Leak Rate = 0.035 m³/s
- Gas Temperature = 200 °C

The following assumptions have been made to predict the emission parameters for a possible leak emergency scenario:

- A pipe burst is assumed to occur at a selected location in the BRP plant flue gas pipe (Figure 9).
- The leak is assumed to remain uncontrolled and uncontained for a minimum of 30 minutes. As such, a transient analysis is employed to simulate this abnormal operating condition.
- No mechanical ventilation system is used to contain pollutant dispersion (CO, SO_2, and NOx emissions were assessed) for light wind conditions.
- The concentrations of carbon monoxide (CO), sulfur dioxide (SO_2), and nitrogen oxides (NOX) in the leaked gases were assumed to be at the level given by the equipment supplier for the exhaust gases after the emission control system.
- All NOX was assumed to be in the form of NO_2.

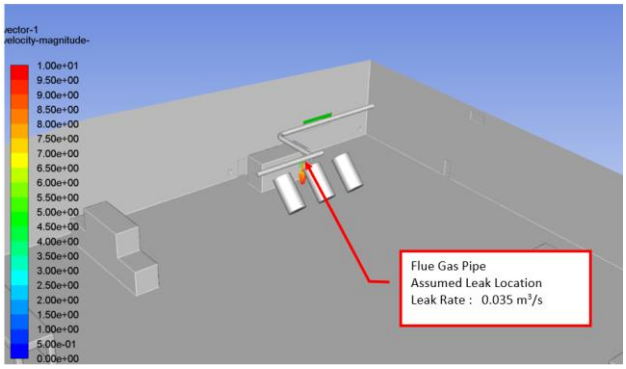

Fig. 9: 3D Model of the Site Including Proposed Openings for Natural Ventilation Study – Pipe Leak
Source: created by the author

Sample results for the worst-case scenario 1 wind conditions are shown in Figure 10, Figure 11, Figure 12, Figure 13, Figure 14, and Figure 15.

The peak concentration at a typical chest level at Time = 1 minute is 5.12x10-5 times the source concentration, equivalent to a 19,531 dilution factor (Figure 10).

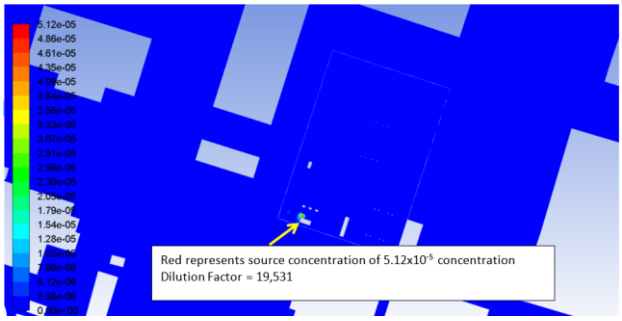

Fig. 10: Concentration Profile (Dilution Factor) at a Typical Chest Level – Time = 1 Minute
Source: created by the author

Figure 11 illustrates the flue gas dispersion in three dimensions. At Time = 1 minute, the flue gases are dispersed away from the emission source

and the gases rise due to their higher temperature (200°C) compared to room temperature (27°C).

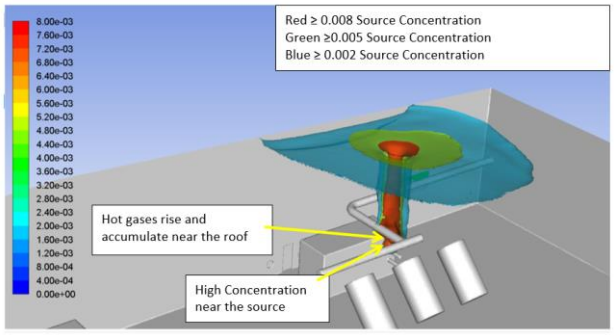

Fig. 11: Iso Surfaces of Dispersed Gases Concentration Profile (Dilution Factor – Time = 1 Minute
Source: created by the author

The peak concentration at a typical chest level at Time = 5 minutes is 9.77x10-5 times the source concentration, equivalent to a 10,325-dilution factor (Figure 12).

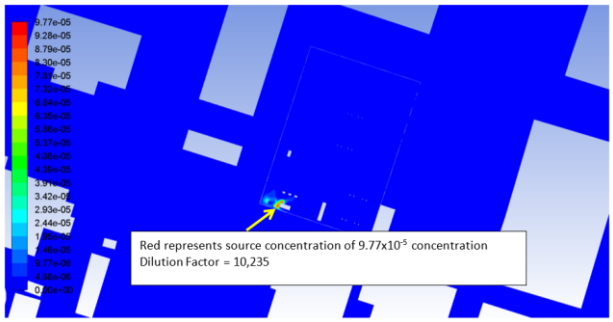

Fig. 12: Concentration Profile (Dilution Factor) at a Typical Chest Level – Time = 5 Minutes
Source: created by the author

The peak concentration at a typical chest level at Time = 30 minutes is 1.0x10-4 times the source concentration, equivalent to a 10,000-dilution factor (Figure 13).

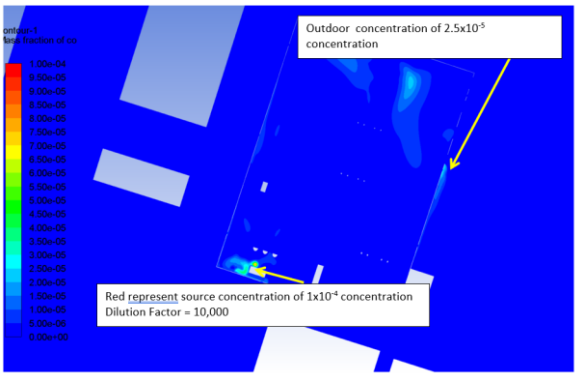

Fig. 13: Concentration Profile (Dilution Factor) at a Typical Chest Level – Time = 30 Minutes
Source: created by the author

Figure 14 illustrates the flue gas dispersion in three dimensions at Time = 30 minute. The flue gas concentrations increase near the roof and within the building with time.

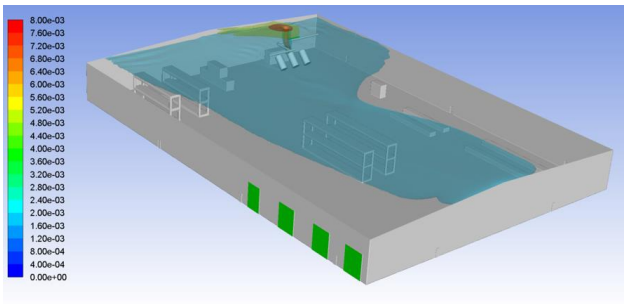

Fig. 14: Concentration Profile (Dilution Factor) at a Typical Chest Level – Time = 30 Minutes
Source: created by the author

A peak concentration of 0.2 times the source concentration (or dilution factor = 5) has been predicted at a small, isolated area near the leak location (Figure 15). The peak concentration will increase if the pipe rupture occurs at a lower elevation or the leakage rate is increased.

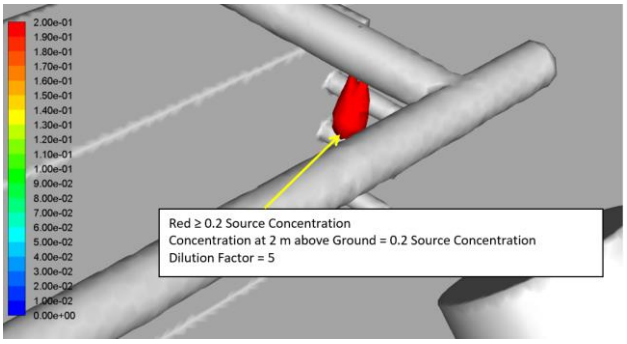

Fig. 15: Peak Concentration at Time = 30 Minutes
Source: created by the author

Table 1 presents the predicted peak concentration levels in the region's most significantly impacted.

The acceptable indoor concentration limits for CO, SO_2, and NO_2 are 6.25 mg/m³ (5 ppm), [21], 13 mg/m³ (5 ppm), [22], and 9.4 mg/m³ (5 ppm), [22], respectively.

The following conclusions can be reached from Table 1:

- Peak SO_2 level is predicted to be well within acceptable levels.
- Unacceptable CO and NO_2 levels are predicted at the most impacted location at 2 m above ground.

Table 1. Predicted Peak Concentration

Pollutant	**Assumed Concentration in Emitted Gases**	**Predicted Peak Concentration – CFD Results**
Not Guaranteed pollutant emissions (6% v/v O_2 dry gas)		
Carbon monoxide (CO)	<75 mg/Nm³ (60 ppm)	11.36 mg/Nm³ (12 ppm)
Sulphur dioxide (SO_2)	8 mg/Nm³ (3.05 ppm)	1.6 mg/Nm³ (0.16ppm)
Not Guaranteed Stack pollutant emissions (3% v/v O_2 dry gas)		
Nitrogen dioxide (NO_2)	400 mg/Nm³ (213 ppm)[3]	80 mg/m³ (42.6 ppm)

Source: created by the author

4.5 Recommendations

Natural Ventilation Recommendations *CIBSE Guide A – Environmental Criteria for Design* requires that a natural ventilation design must be flexible so that it can respond to the variability in climate.

Several design solution options have been developed based on simulation results, with one recommendation presented in Figure 16. The proposed solutions include the following:

- BRP Processing Plant
  - Enclose the BRP processing plant, except for air inlet louvres, to effectively control vapour and gas emissions under both normal and abnormal conditions (e.g., leaks or spills). The enclosure can be constructed using any fire-rated material, such as Deco-clad, PVC cladding, or precast concrete. The initial design featured open mesh fencing around the BRP area.
  - Install roof-mounted hybrid turbine ventilators or suction fans to achieve the required ventilation rate. Air delivery should be no less than 96,144 m³ per hour. The proposed ventilators will primarily operate in natural ventilation mode but can switch to mechanical ventilation during calm or low wind conditions. Turbine ventilators harness wind power to initiate spinning, generating an extraction vortex that removes stale air from the building. These ventilators can also facilitate smoke release in case of fire and may be equipped with dampers.
- All Other Areas
  - Provide roof-mounted hybrid turbine ventilators to ensure air delivery rates do not fall below 190,000 m³ per hour to satisfy the design criteria and regulatory requirements.

Other design modifications under consideration include adjustments to the building layout and additional openings via low-level louvres.

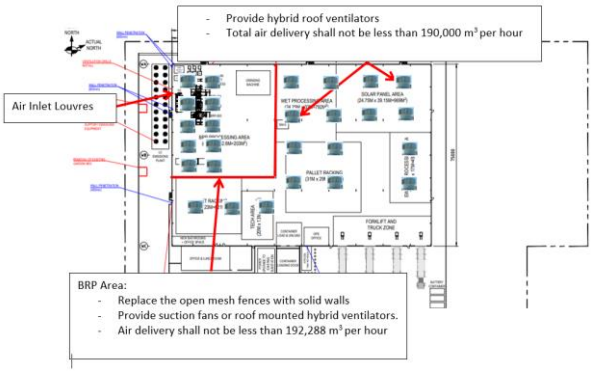

Fig. 16: Proposed Design Modifications
Source: created by the author

5 Conclusion

Designing an effective ventilation system for e-waste recycling facilities presents unique challenges due to the complexity of waste materials and potential emissions of hazardous air pollutants. Under abnormal operating conditions, syngas may be released into the facility, increasing the risk of fire and explosion.

The current study provides valuable insights into optimising airflow distribution, contamination control, and emergency ventilation strategies, and explores key considerations in designing an effective ventilation solution for e-waste facilities featured in an industrial showcase project in NSW, Australia

The paper introduces a comprehensive strategy for optimising natural ventilation, aimed at enhancing both sustainability and operational efficiency. By incorporating site-specific wind rose data into a sophisticated Computational Fluid Dynamics (CFD) model, it addresses complex design challenges arising from site-specific constraints and proposes solutions under both routine and abnormal operating conditions, factors that are frequently overlooked yet critically important. This integrated approach carries significant commercial relevance, particularly considering the anticipated rapid growth of e-waste processing facilities in the near future.

Additionally, the combined external-internal CFD study could be expanded to address Air quality and odour control strategies, Environmental wind effects influencing facility performance, Thermal comfort assessment for worker health and safety, Fire risk evaluation to enhance emergency preparedness, and Smoke dispersion analysis for improved air management.

Declaration of Generative AI and AI-assisted Technologies in the Writing Process

The authors wrote, reviewed and edited the content as needed and verifies that none utilised artificial intelligence (AI) tools were used. The authors take full responsibility for the content of the publication.

References:

[1] NSW Government, *NSW Government's Code of Practice: Managing the Work Environment and Facilities,* 2019.

[2] Chartered Society of Physiotherapy Workplace, *Health, Safety and Welfare Regulations*, UK 1992.

[3] Health and Safety Executive, *EH40/2005 Workplace exposure limits*, ISBN 9780717667031, 2020.

[4] Australian Building Codes Board, *The Australian National Construction Code*, Part 3.8.5.3, Vol. 2, 2022.

[5] Al-Khalidy N, Better Natural Ventilation Design for Single Sided Apartments Utilising Computational Fluid Dynamic, *International Journal of Architectural Engineering Technology*, Vol. 7, 2020, pp. 13-22.

[6] Australian Standard, *Electrical Equipment for Explosive Atmospheres Protection by Ventilation – Type of Protection V*, AS 1482-1985 (Reconfirmed 2013).

[7] Al-Khalidy N, The Role of Computational Fluid Dynamics in Solving Wind Engineering Problems, *IEEE Computer Society Conference - 3rd International Conf. on Mathematics and Computers in Sciences and Industry*, Greece 2016.

[8] Nishioka T., Moteki M., Miyagi H., Tatumi T., Kadoya. Contamination Control by Unidirectional Flow Ventilation in a Refuse Disposal Facility, *Air Infiltration and Ventilation Centre,* [Online]. https://www.aivc.org/sites/default/files/airbase_12115.pdf (Accessed Date: September 17, 2024).

[9] Heiselberg P., Svidt K., Kragh H., Application of CFD in Investigation of Ventilation Strategies for Waste Incineration. *Proceedings of the 5th International Symposium on Ventilation for Contaminant Control*, Ottawa, Canada, September 14–17, 1997, pp. 155–163.

[10] Li Y., Nielsen P.V. CFD and Ventilation Research. *Indoor Air*, 21(6), 2011, pp. 442–453.

[11] Heng-Jin Tham, Mohd Suffian, and Chi-Ming Chu, CFD Assessment of Natural Ventilation Designs for Composting Systems, In *Waste Management, Processing and Valorisation*, Edited by A. Yaser, H. Tajarudin, and A. Embrandiri, Springer, 2022. DOI: 10.1007/978-981-16-7653-6_5.

[12] Egedy, A., Fogarasi, S., Varga, T. et al., CFD models in the development of electrical waste recycling technologies, *Clean Technologies and Environmental Policy*, Vol. 16, 2014, pp. 1255–1263. https://doi.org/10.1007/s10098-014-0816-6.

[13] Patziger, M., Improving Wastewater Treatment Plant Performance by Applying CFD Models for Design and Operation: Selected Case Studies, *Water Science and Technology*, Vol. 84, No. 2, 2021, pp. 323–332.

[14] Hafeez, S., Aristodemou, E., Manos, G., Al-Salem, S. M., and Constantinou, A. Computational Fluid Dynamics (CFD) and Reaction Modelling Study of Bio-oil Catalytic Hydrodeoxygenation in Microreactors, *Reaction Chemistry & Engineering*, Vol. 5, 2020, pp. 1083–1092.

[15] Hartulistiyoso, E., Sigiro, F., and Yulianto, M., Temperature distribution of the plastics pyrolysis process to produce fuel at 450°C, *Procedia Environmental Sciences*, Vol. 28, 2015, pp. 234–241.

[16] Luis Alberto De la Flor-Barriga and Ursula Fabiola Rodríguez-Zúñiga, Numerical analysis on a Catalytic Pyrolysis Reactor Design for Plastic Waste Upcycling using CFD Modelling, *RSC Advances*, Vol. 12, 2022, pp. 12436–12445. doi: 10.1039/D2RA01407F.

[17] Zhang, J., Johnson, W., and Plikas, T., Application of Computational Fluid Dynamics for Solving Ventilation Problems in Metallurgical Industrial Processes, *International Journal of Ventilation*, Vol. 16, No. 3, 2017, pp. 200–212.

https://doi.org/10.1080/14733315.2017.1299516.

[18] Heiselberg, P. K., Svidt, K., and Kragh, H., Application of CFD Investigation of Ventilation Strategies for Improvement of Working Environment in a Waste Incineration Plant, *Ventilation '97: Global Development in Industrial Ventilation*, Proceedings of the 5th International Symposium on Ventilation for Contaminant Control, Ottawa, Canada, Sept., 1997, PP.14–17.

[19] ANSYS, *Ansys Fluent Theory Manual*, USA 2024.

[20] Shih, T. et al., "A New Eddy-Viscosity Model for High Reynolds Number Turbulent Flows: Model Development and Validation," *Computers & Fluids*, vol. 24, no. 3, 1995, pp. 227–238.

[21] ASHRAE, *ANSI/ASHRAE Standard 55-2017: Thermal environmental conditions for human occupancy*. American Society of Heating, Refrigerating and Air-Conditioning Engineers, 2017.

[22] Safe Work Australia, *Workplace exposure standards for airborne contaminants*. Safe Work Australia, 2019, [Online]. https://www.safeworkaustralia.gov.au/ (Accessed Date: September 17, 2024).

Contribution of individual authors to the creation of a scientific article (ghostwriting policy)

The author contributed to the present research, at all stages from the formulation of the problem to the final findings and solution

Sources of funding for research presented in a scientific article or scientific article itself

No funding was received for conducting this study. The original study was carried out to support a Development Application for the project site.

Creative Commons Attribution License 4.0 (Attribution 4.0 International, CC BY 4.0)

This article is published under the terms of the Creative Commons Attribution License 4.0.
https://creativecommons.org/licenses/by/4.0/deed.en_US

APPENDIX

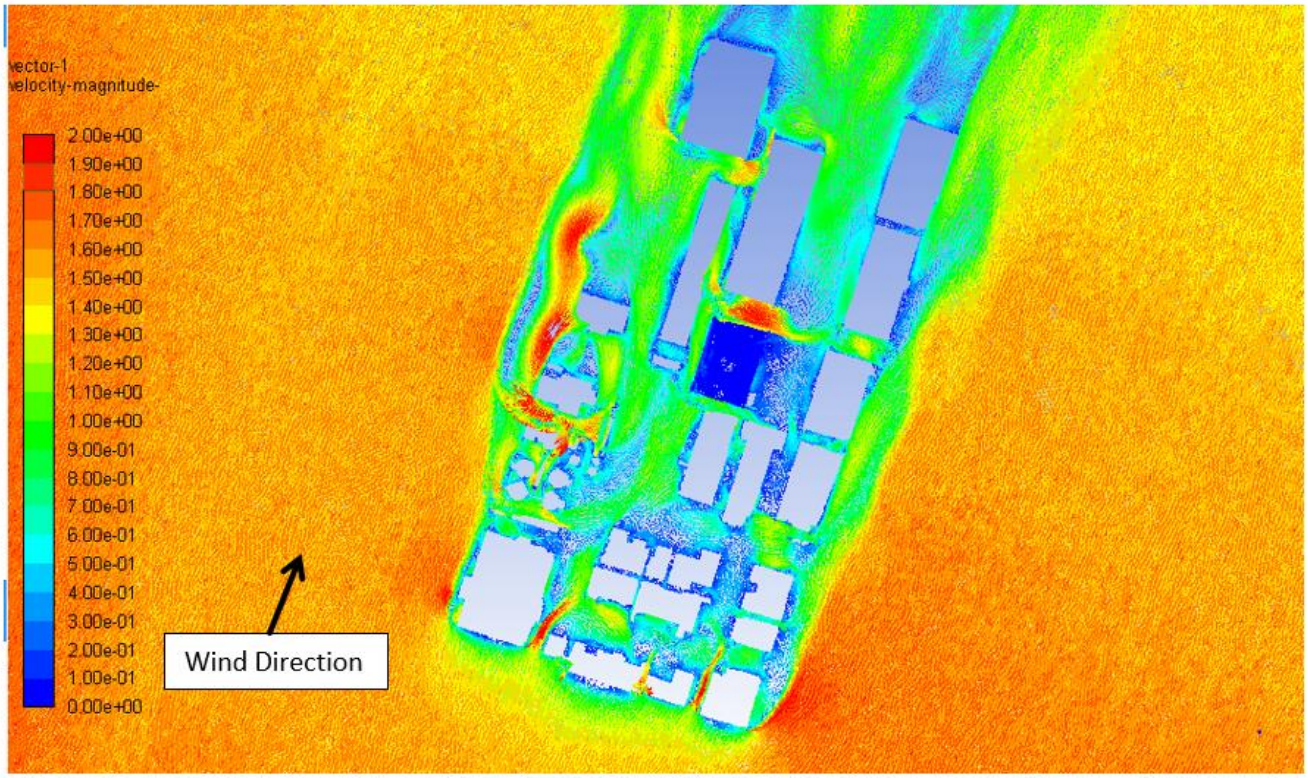

Fig. 4: Velocity Vector at 1.5 m above Ground (Typical Chest level), South-Southwest Wind, Approaching Wind Speed = 1.5 m/s at 10 m High

Source: created by the author

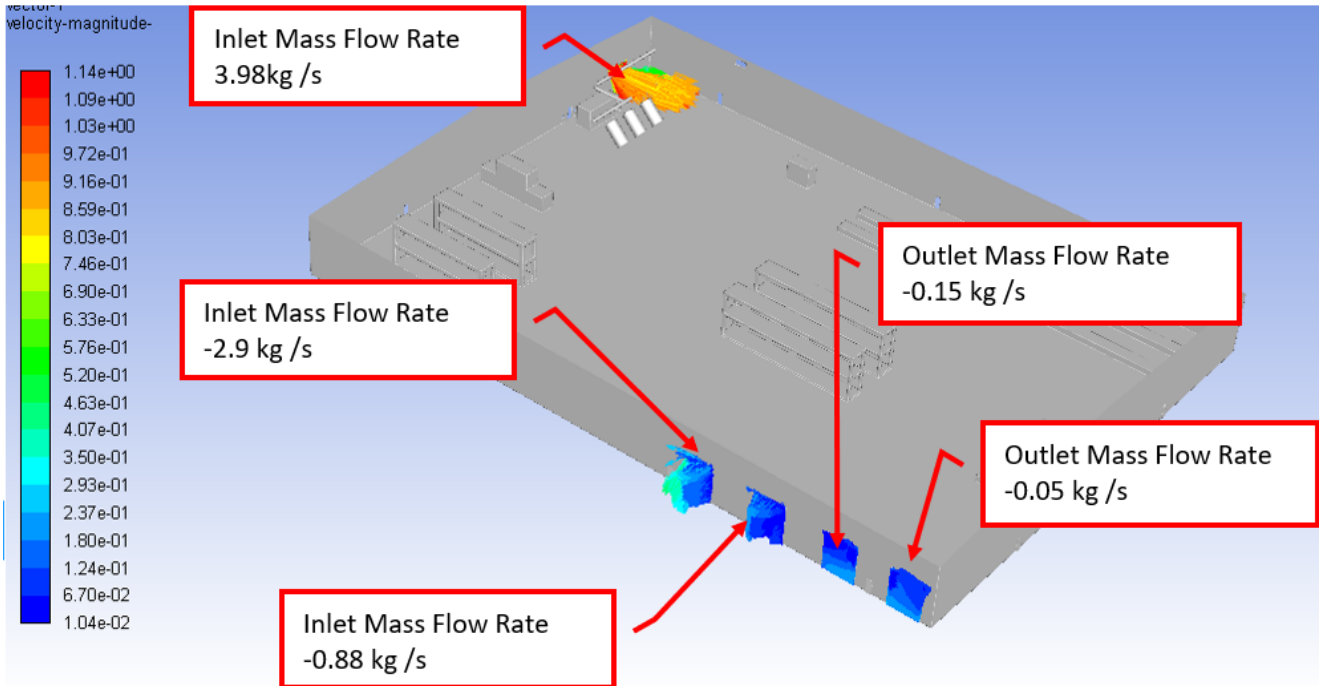

Fig. 5: Velocity Vector (m/s) at the Openings

Source: created by the author

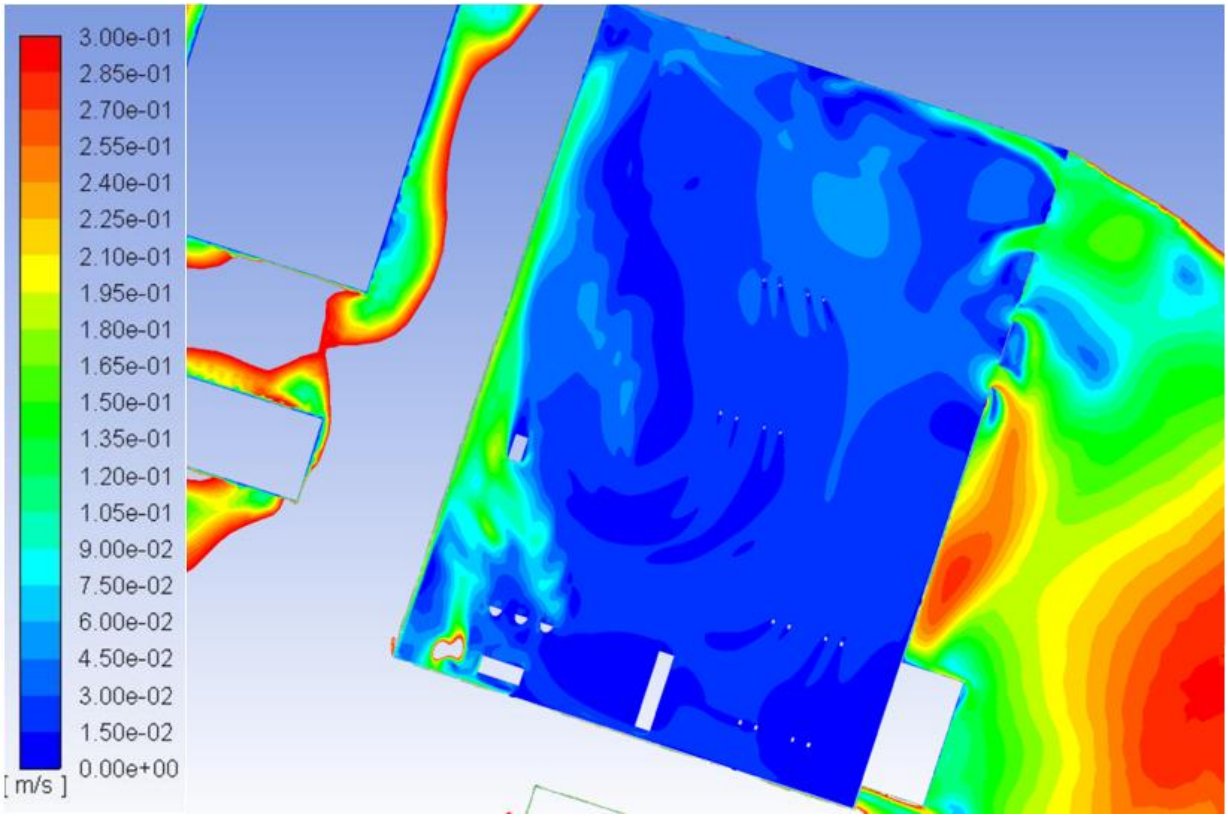

Fig. 6: Contours of Velocity Magnitudes (m/s) at 1.5 m above Ground (Typical Chest level), South-Southwest Wind, Approaching Wind Speed = 1.5 m/s at 10 m High

Source: created by the author

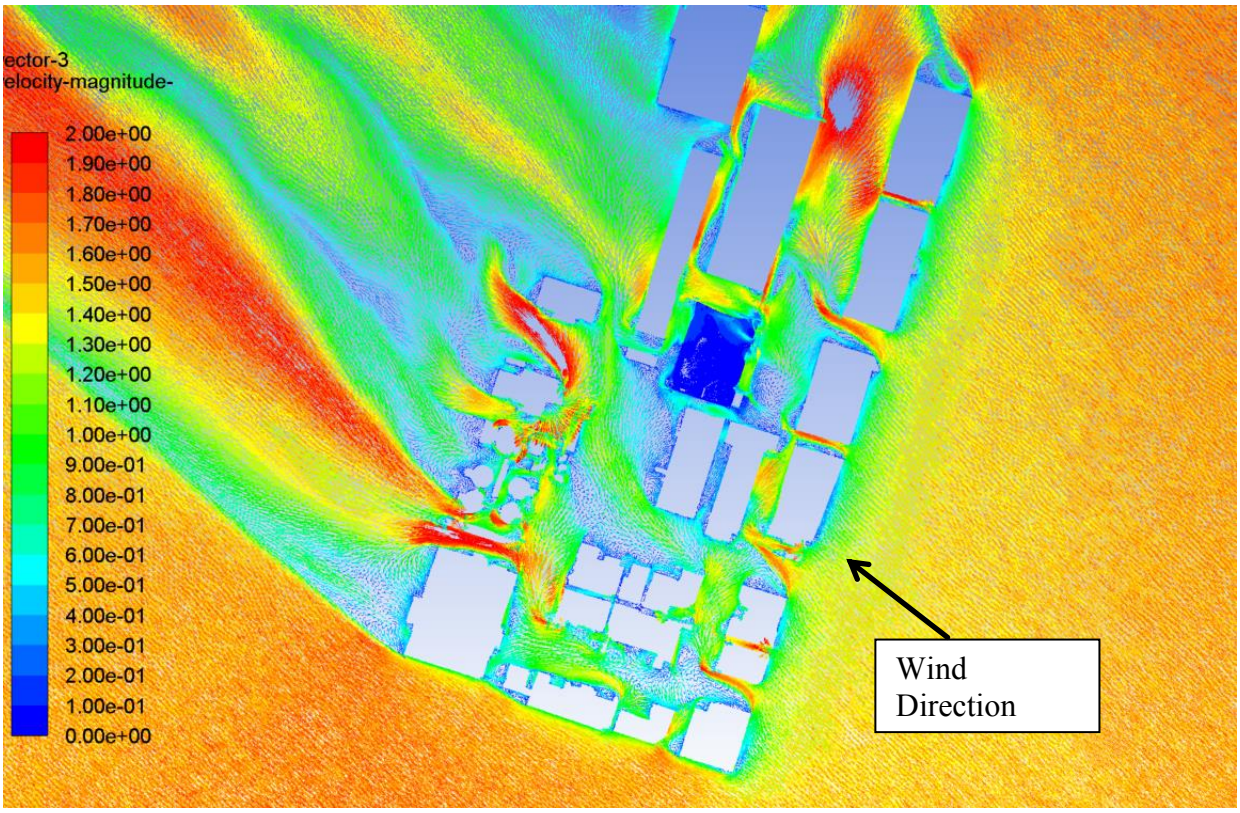

Fig: 7: Velocity Vector at 1.5 m above Ground (Typical Chest level), Southeast Wind, Approaching Wind Speed = 1.5 m/s at 10 m High

Source: created by the author

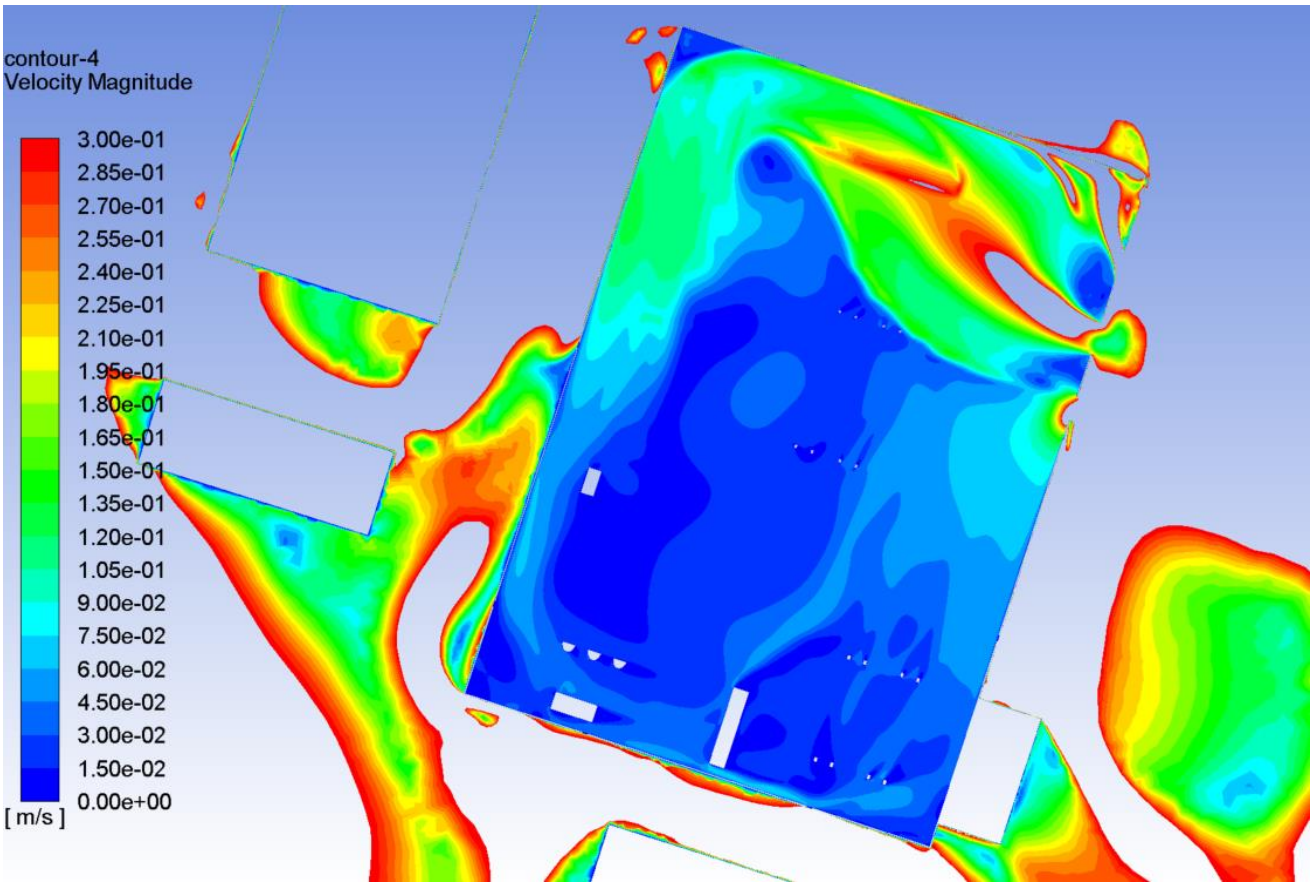

Fig. 8: Contours of Velocity Magnitudes (m/s) at 1.5 m above Ground (Typical Chest level), South-Southwest Wind, Approaching Wind Speed = 1.5 m/s at 10 m High

Source: created by the author