Flatulence Diffusion in a Confined Space:

A Numerical Simulation Report

Ling Xiao

Student No.: 1M221091

Virtual Earth: Simulating the Environment 01

July 21, 2025

Contents

1 Introduction

1.1 Motivation and Scenario

Consider a common yet scientifically intriguing scenario: nine people are sharing a small, 2m x 2m elevator. Suddenly, one person releases a silent, localized puff of gas. The immediate, unspoken question in everyone's mind is a matter of both social anxiety and physics: who will smell it first? The spread of this odor is not instantaneous or random; it is governed by the fundamental physical principles of fluid dynamics and mass transfer.

This project was motivated by the desire to answer this question using a scientific, computational approach. By modeling the flatulence puff as a burst of gas with an initial concentration and velocity, we can simulate its journey through the confined space. This allows us to predict the dispersion pattern and determine the precise moment the concentration at each bystander's location crosses a detectable threshold. Beyond the humorous context, this problem serves as an excellent, accessible case study for the complex processes of advection and diffusion that are fundamental to environmental science, chemical engineering, and atmospheric physics.

1.2 Research Objectives

The core objectives of this study were defined as follows:

1. **Develop a Numerical Model:** To construct a robust two-dimensional simulation based on the advection-diffusion equation to model the transport of a gas puff in a confined, square domain.

2. **Implement a Stable Solver:** To employ a hybrid numerical scheme that accurately and efficiently solves the governing equation, ensuring stability over the entire simulation time.

3. **Simulate and Visualize:** To run simulations for various initial conditions (source location and wind direction) and generate dynamic visualizations (animations) of the concentration field.

4. **Analyze Detection Events:** To record and analyze when and where the gas concentration exceeds a predefined sensory threshold at the locations of stationary observers (detectors).

2 Physical and Mathematical Framework

2.1 Governing Equation: The Advection-Diffusion Equation

The evolution of the gas concentration, denoted by $C(x, y, t)$, is described by the 2D unsteady advection-diffusion equation. This partial differential equation (PDE) is a conservation law that accounts for the change in concentration at a point due to two distinct transport phenomena:

$$\underbrace{\frac{\partial C}{\partial t}}_{\text{Rate of Change}} + \underbrace{\mathbf{u} \cdot \nabla C}_{\text{Advection}} = \underbrace{D \nabla^2 C}_{\text{Diffusion}}$$

where:

- $C(x, y, t)$ is the concentration of the gas at spatial coordinates (x, y) and time t.

- $\mathbf{u} = (u_x, u_y)$ is the velocity vector field of the medium (i.e., the "wind" generated by the initial burst).
- D is the molecular diffusion coefficient, which quantifies the rate of diffusion.
- $\nabla C = \left(\frac{\partial C}{\partial x}, \frac{\partial C}{\partial y}\right)$ is the gradient of the concentration. The term $\mathbf{u} \cdot \nabla C$ represents the transport of concentration due to the bulk fluid motion (advection).
- $\nabla^2 C = \frac{\partial^2 C}{\partial x^2} + \frac{\partial^2 C}{\partial y^2}$ is the Laplacian of the concentration, representing the net "flow" of substance due to random molecular motion from regions of high concentration to low concentration (diffusion).

2.2 Initial Conditions: The Gaussian Puff

At the beginning of the simulation ($t = 0$), the flatulence event is modeled as an instantaneous, localized release of gas. This is mathematically represented by a two-dimensional Gaussian distribution centered at the source location (x_0, y_0):

$$C(x, y, 0) = C_0 \exp\left(-\frac{(x - x_0)^2 + (y - y_0)^2}{2\sigma^2}\right)$$

Here, C_0 represents the maximum initial concentration at the center of the puff, and σ (the standard deviation) defines its initial radius or spread.

2.3 Boundary Conditions: The Confined Space

The simulation takes place within a closed 2m x 2m domain, representing the elevator. To ensure that no gas escapes this domain, zero-flux Neumann boundary conditions are enforced. This means that the concentration gradient normal to any boundary must be zero:

$$\left.\frac{\partial C}{\partial \mathbf{n}}\right|_{\partial\Omega} = 0$$

where $\partial\Omega$ represents the boundary of the domain and $\mathbf{n}$ is the vector normal to that boundary. In the finite difference implementation, this condition is conveniently approximated by setting the concentration value in a boundary grid cell equal to the value in the adjacent interior cell.

3 Numerical Implementation

The advection-diffusion PDE is solved numerically on a discrete grid using a finite difference method. To handle the different mathematical properties of the advection and diffusion terms, an operator splitting approach was adopted. This powerful technique allows us to solve for the effects of advection and diffusion sequentially in each time step, Δt.

3.1 Advection Solver: Explicit Upwind Scheme

The advection part of the equation, $\frac{\partial C}{\partial t} = -\mathbf{u} \cdot \nabla C$, is solved first. We use an explicit first-order upwind scheme. This scheme is "upwind" because it approximates the spatial derivative using a one-sided difference taken from the direction the flow is coming from. For a positive velocity $u_x > 0$, the derivative $\frac{\partial C}{\partial x}$ is approximated using values from the left; for $u_x < 0$, it uses values from the right. This ensures that information is propagated in the correct direction and, crucially, prevents the non-physical oscillations that can arise with central difference schemes in advection-dominated problems. Being explicit, its stability is limited by the Courant-Friedrichs-Lewy (CFL) condition.

3.2 Diffusion Solver: Crank-Nicolson with ADI

The diffusion part, $\frac{\partial C}{\partial t} = D\nabla^2 C$, is solved second, using the result of the advection step as its initial condition. For this, we use the Crank-Nicolson method, an implicit scheme that averages the spatial derivatives at the current and next time levels. Its key advantage is that it is **unconditionally stable** for the diffusion equation, meaning the choice of time step Δt is dictated by accuracy requirements rather than stability constraints. This allows for significantly more efficient simulations compared to explicit methods.

To apply this to a 2D problem efficiently, the Crank-Nicolson scheme is implemented using an **Alternating-Direction Implicit (ADI)** method. ADI splits the 2D diffusion solve into two separate, one-dimensional solves:

1. **Step 1 (Implicit in x):** A 1D implicit Crank-Nicolson solve is performed for each row of the grid. This involves solving a simple tridiagonal system of linear equations for each row.
2. **Step 2 (Implicit in y):** Using the intermediate results from Step 1, a 1D implicit Crank-Nicolson solve is performed for each column of the grid, which again involves solving tridiagonal systems.

This approach avoids the need to solve a large, complex 2D system of equations, making the implicit method computationally tractable and fast.

4 Simulation Setup

4.1 Model Parameters

The simulation was configured using a set of carefully chosen parameters, detailed in the Python implementation and summarized in Table 1.

Table 1: Key Simulation Parameters

Parameter Category	Parameter Description	Symbol	Value
Spatial Domain	Domain Width	W	2.0 m
	Domain Height	H	2.0 m
	Grid Points in X-direction	N_X	201
	Grid Points in Y-direction	N_Y	201
Temporal Domain	Total Simulation Time	T_{total}	10.0 s
	Time Step	Δt	0.02 s
	Total Number of Time Steps	N_T	500
Physical Properties	Diffusion Coefficient	D	0.01 m^2/s
	Initial Peak Concentration	C_0	1.0 (normalized units)
	Initial Puff Standard Deviation	σ	0.075 m
	Initial Burst Speed	v_{init}	1.0 m/s
	Wind Duration (linear decay)	T_{wind}	2.0 s
Detection	Spectator/Detector Locations	-	8 fixed points (see Fig 1)
	Concentration Threshold	C_{thresh}	0.03 (normalized units)

Note: The diffusion coefficient D was set to 0.01 m^2/s, a value significantly higher than that of typical gases in air (e.g., $D_{H_2S} \approx 1.6 \times 10^{-5}$ m^2/s). This was an intentional choice to make the diffusion process more visually apparent within the short 10-second simulation timeframe.

4.2 Source and Detector Configuration

The 2m x 2m domain contains nine potential "person" locations arranged in a 3x3 grid. In any given simulation run, one of these is chosen as the gas source, and the other eight act as stationary detectors. Figure 1 illustrates this initial configuration. Each detector's status is tracked, and its color in the visualization changes permanently from grey ('safe') to red ('detected') if the gas concentration at its coordinates surpasses the threshold value C_{thresh}.

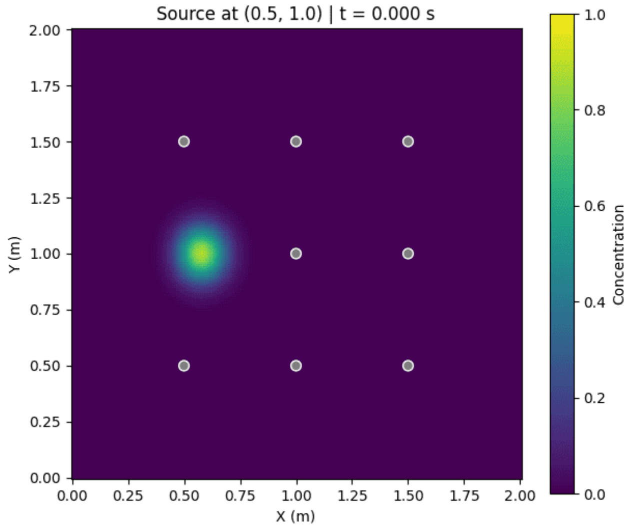

Figure 1: Initial state of a simulation. The gas originates from Source 4 (center-left), represented by the yellow-green Gaussian puff. The eight other locations are detectors, shown as grey circles.

5 Results and Analysis

Simulations were executed for multiple scenarios, varying the source location (1 through 9) and the initial wind angle (0° to 360°). The output of each simulation is a GIF animation that visualizes the concentration field's evolution.

5.1 Case Study 1: Source 1, Wind Angle 300°

In this scenario (Figure 2), the gas originates from the top-left corner (Position 1). The initial wind angle of 300° imparts a velocity directed towards the bottom-right of the domain.

- **Advection Phase (0-2s):** The puff is rapidly transported across the diagonal of the elevator. The central detector (Position 5) and the detectors in the path of the main plume (Positions 6, 8, 9) are impacted quickly.
- **Diffusion Phase (2-10s):** After the initial wind decays, the concentrated plume begins to spread out more slowly due to diffusion. The gas eventually reaches the corners that were initially shielded from the direct advective path (e.g., Position 3 and 7), but at much lower concentrations and at later times.

This case clearly illustrates how individuals directly "downwind" of the source are affected almost immediately, while others might not detect the odor for several seconds, if at all.

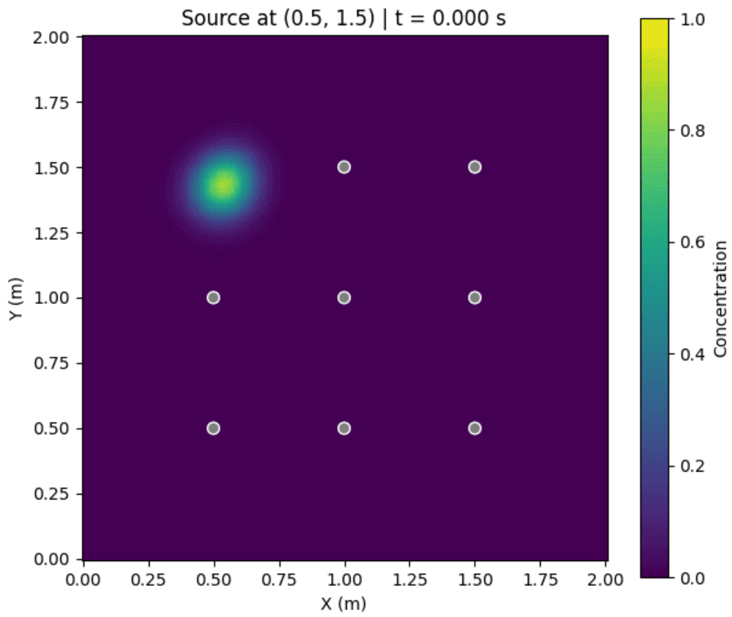

Figure 2: A snapshot from the simulation with the source at Position 1 and a wind angle of 300°. The plume travels from the top-left towards the bottom-right.

5.2 Case Study 2: Source 7, Wind Angle 42°

This simulation (Figure 3) starts with the source at the bottom-left (Position 7) and a wind angle of 42°, propelling the gas towards the upper-right.

- **Advection Phase (0-2s):** The bulk of the gas moves along a path towards Position 3. Detectors 4, 5, and 8 are on the immediate edge of this path and are affected early on.
- **Diffusion Phase (2-10s):** As diffusion takes over, the gas cloud expands from the main advection path, gradually enveloping more of the space. Detectors further away, like Position 1 and 9, are reached much later by the slow-moving diffusion front.

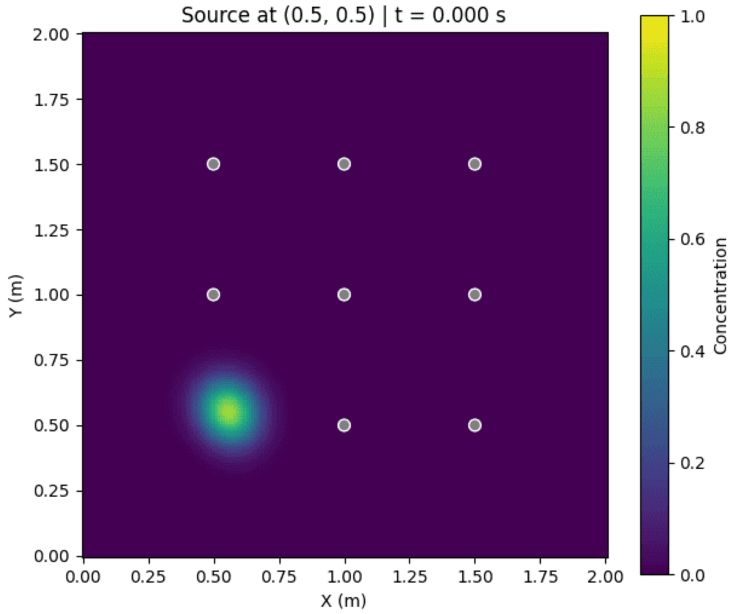

Figure 3: A snapshot from the simulation with the source at Position 7 and a wind angle of 42°. The plume travels from the bottom-left towards the upper-right.

6 Discussion and Limitations

6.1 Interpretation of Results

The simulation results consistently underscore a key insight: in the initial moments following the gas release, **advection is the overwhelmingly dominant transport mechanism**. The initial velocity and direction of the puff almost single-handedly determine the short-term outcome and the sequence of detection. Diffusion acts as a secondary, slower process that spreads the gas more uniformly over time, but its effects are less dramatic in the first few seconds. The choice of a stable numerical scheme like Crank-Nicolson was validated, as it allowed the simulation to proceed efficiently without numerical instabilities that could corrupt the physical results.

6.2 Model Limitations

While the simulation provides valuable insights, it is built on a number of simplifying assumptions. Acknowledging these limitations is crucial for correctly interpreting the results.

1. **Two-Dimensional Model:** The simulation is constrained to a 2D plane, ignoring all vertical gas movement. In reality, flatulence gas is often warmer than the ambient air and would experience buoyancy, causing it to rise. This 3D effect is completely absent.

2. **Absence of Turbulence:** The model assumes laminar (smooth) flow. The air in an elevator is never perfectly still; it is subject to complex turbulent eddies caused by ventilation systems and the movement and breathing of its occupants. Turbulence would dramatically increase the rate of mixing and lead to a much more chaotic and rapid dispersion.

3. **Simplified Physics:** The model uses a constant diffusion coefficient (D) and a simple, spatially uniform velocity field that decays linearly. Real-world diffusion is temperature-dependent, and airflow patterns are highly complex and non-uniform.

7 Conclusion and Future Work

7.1 Summary of Findings

This project successfully developed and implemented a 2D numerical simulation to model gas dispersion in a confined space. By employing an operator splitting method combining an upwind scheme for advection and a Crank-Nicolson scheme for diffusion, the model proved to be both stable and computationally efficient. The simulations effectively demonstrate how the initial burst of flatulence—its speed and direction—is the critical factor in determining who is first to detect the odor.

7.2 Recommendations for Future Research

The current model serves as a strong foundation that can be expanded upon in several key areas to enhance its realism and predictive power:

- **Extension to 3D:** The most impactful next step would be to extend the simulation to a full three-dimensional domain. This would allow for the modeling of buoyancy effects (vertical gas movement), which are critical for a realistic simulation.
- **Incorporation of Turbulence Models:** To capture the chaotic mixing of real-world airflow, a turbulence model, such as a simple Reynolds-averaged Navier–Stokes (RANS) model or a more complex Large Eddy Simulation (LES), could be incorporated.
- **Coupling with a CFD Solver:** For maximum realism, the advection-diffusion solver could be coupled with a full computational fluid dynamics (CFD) solver. This would allow for the simulation of complex airflow patterns generated by ventilation systems or the movement of people within the elevator, providing a dynamic and realistic velocity field $\mathbf{u}(x, y, z, t)$.