

DOI: 10.37943/XTXY2261

Sabina Duisaliyeva

Master's Student, School of Information Technology and Engineering,
duisaliyeva.sabina@gmail.com, orcid.org/0009-0006-5462-7943
Kazakh-British Technical University, Kazakhstan

Aliya Tursynzhanova

Master's Degree, Senior Lecturer, School of Applied Mathematics,
aliya.tursynzhanova@gmail.com, orcid.org/0000-0002-3918-6688
Kazakh-British Technical University, Kazakhstan

Perizat Omarova

PhD, Associate Professor, Department of Software Engineering
omarova.peryzat2@gmail.com, orcid.org/0000-0001-9502-7959
Satbayev University, Kazakhstan
Y.Amirgaliyev Laboratory of Artificial Intelligence and Robotics, Kazakhstan
Institute of Information and Computational Technologies, Kazakhstan

INFLUENCE OF BUILDING CONFIGURATION AND ORIENTATION ON AIRFLOW PATTERNS AND VENTILATION EFFICIENCY IN RESIDENTIAL URBAN ENVIRONMENTS: A NUMERICAL STUDY

Abstract: Air pollution remains a major health concern in urban areas, where building density and layout strongly influence airflow and natural ventilation. While many studies focus on pollutant concentrations and emission sources, fewer examine how building geometry affects the formation of low-velocity zones. This study investigates the influence of building shape and arrangement on ventilation efficiency using two-dimensional numerical modelling. The incompressible Navier–Stokes equations are solved in Python using the projection method on uniform finite-difference grids.

Before analysing the main residential layouts, a test case was performed for two identical domains. In the second domain, the square buildings were rotated by 45°. The results showed that building orientation changes both peak velocity and the extent of low-velocity regions.

Three residential configurations were then examined. Case 1 included L-shaped and rectangular buildings on a 501×501 grid, Case 2 contained only rectangular buildings on the same grid, and Case 3 included diamond-shaped and square buildings on a 301×301 grid. Case 2 showed the smallest low-velocity area (11.1%) and the highest mean air velocity (1.101 m/s), corresponding to a 45% reduction in low-velocity area and a 48% increase in mean velocity compared with Case 1 (20.2%, 0.745 m/s). In Case 3, the diagonal orientation caused local flow acceleration, but the low-velocity fraction remained high at 20.6%.

Overall, the results show that building arrangement has a strong effect on horizontal airflow distribution. They also indicate that peak velocity alone is not sufficient to assess ventilation quality. The proposed two-dimensional approach can therefore be used as an initial screening tool for comparing alternative residential layouts before more detailed three-dimensional simulations are carried out.

Keywords: air pollution; ecology; modeling; Navier-Stokes; finite difference method; numerical modeling; projection method.

Introduction

Air pollution is one of the major causes of illness in all sections of society across the globe. Furthermore, air pollution remains a significant factor in global mortality, according to the 2025 State of the Global Air study [1-2].

Exposure to air pollution is linked to problems such as cardiovascular and respiratory diseases, asthma, diabetes and strokes. A significant proportion of the population suffers from

chronic conditions influenced by long-term exposure to pollutants. This results in both human suffering and an additional burden on the healthcare system. Recent studies show that air pollution even affects the nervous system and can cause conditions including dementia [3].

Many countries have introduced their own air-quality policies to address pollution in cities. In India, the National Clean Air Programme focuses mainly on reducing particulate matter in urban areas [4]. The European Union follows a broader approach through the Zero Pollution Action Plan, which aims to reduce the long-term effects of pollution on both human health and the environment [5]. China has also introduced a series of Blue Sky initiatives aimed at limiting major sources of urban emissions and improving ambient air quality [6]. Comparable measures have been adopted in the United States [7] and the United Kingdom [8]. Across East Asia, air-quality strategies increasingly combine emission reduction with continuous monitoring and urban planning measures [9-10].

These projects demonstrate the growing importance of modelling techniques in assessing and planning strategies to reduce pollutant concentrations in urban air. From an information technology perspective, numerical modeling of air pollution is based on computational algorithms designed to solve complex physical equations and process large volumes of environmental data. The combination of numerical methods and modern computing technologies makes it possible to model the transport of pollutants and assess air quality in complex urban environments.

The European Environment Agency reports that, despite progress in tackling air pollution, around 95 per cent of urban residents in Europe still breathe air that exceeds the standards set by the World Health Organisation (WHO). This means that air pollution remains a serious threat to public health.

Rapid urban expansion and increased building density can degrade urban air quality by altering local airflow patterns and pollutant dispersion processes. For instance, research by Bui and Nghiem [11] demonstrated that urbanization leads to a significant rise in fine particulate matter (PM_{2.5}) pollution: a one-percentage-point increase in the level of urbanization results in an average rise in PM_{2.5} concentration of 0.54 µg/m³. Additionally, in a large-scale study covering 2,273 Chinese cities, Niu et al. [12] analyzed the spatiotemporal patterns of the "air pollution island" effect in urban environments and the factors contributing to its formation.

In today's world, residential complexes function as cities within cities. This means that the complex also includes nurseries, schools, playgrounds, shops, beauty salons, restaurants and other commercial facilities. This enables people to live, work and spend their leisure time all in one place [13-14]. In such cases, the development of architectural solutions that improve natural ventilation plays a key role [15]. Thus, the relationship between the complex's geometry and air quality can be of decisive importance when assessing people's living conditions. Understanding how different building shapes and configurations affect airflow patterns can help in assessing air quality and developing urban planning strategies [15-16].

In a study on urban morphology, Sadaf Vahidi, Mansour Yeganeh and Masoud Gasaban analysed various geometric building forms and concluded that different building forms respond differently to changes in height, the porosity of the surrounding environment, density and orientation [17-18]. Airflow dynamics and ventilation are the main mechanisms influencing the transport and dispersion of pollutants in the urban environment. Even a slight change in the pitch of a residential building's roof can significantly affect air quality [19-20].

Physically based numerical simulations are essential for examining flow dynamics in intricate systems. Studies in computational fluid dynamics utilizing the incompressible Navier–Stokes equations, along with effective pressure–velocity coupling techniques like projection or PISO methods, have been effectively employed in various environmental and engineering flow challenges, showcasing their precision and numerical stability under suitable boundary conditions [21]. Recent advances and new computational methods have expanded the capabilities of air pollution modeling. High-performance computing and data-driven approaches make it possible to process large environmental datasets.

At the same time, machine learning methods are being studied; these enable the identification of spatial patterns in complex physical models, with a key focus on the influence of geometry on the behaviour of the system. Nevertheless, such models operate on the basis of abstract characteristics. Consequently, they do not capture the basic and fundamental principles of physics that govern the formation of wind flows [22-23].

Numerical airflow modelling is a practical way to examine how air velocity is redistributed within urban layouts, including the formation of recirculation zones and areas with weak ventilation. These results can be used to compare different building configurations before conducting more detailed three-dimensional or pollutant-dispersion simulations. For example, Rakhimberdin et al. [24] developed a predictive model for assessing air quality in Astana (Kazakhstan) and demonstrated the potential of computational methods to support the analysis of the urban environment.

Many previous studies have utilized realistic and notably complex urban configurations. The results of these studies provide valuable information, but are specific to sites, making it difficult to identify and systematically analyse the fundamental influence of building orientations and layouts on wind flow behaviour. Furthermore, a significant number of studies on environmental science and air pollution focus on identifying emission sources and concentration levels, whilst overlooking the importance of natural ventilation and the nature of airflows.

Previous studies have shown that urban morphology has a strong influence on natural ventilation. However, many investigations rely on detailed, site-specific simulations or focus mainly on pollutant concentration fields. Although such approaches provide valuable information, they do not always make it easy to determine whether a local increase in velocity also improves ventilation across the whole layout. This creates a need for a simpler comparative approach that can identify weakly ventilated regions before more computationally demanding three-dimensional simulations are performed.

The novelty of this study is not related to the projection method itself, which is already well established, but to the way it is used for the comparative assessment of residential building layouts. The proposed framework combines local and domain-scale airflow indicators, including maximum velocity, domain-averaged velocity, and the fraction of low-velocity regions. This makes it possible to distinguish between local flow acceleration and overall ventilation performance. The results show that an increase in velocity in some parts of a layout does not necessarily lead to a smaller weakly ventilated area. The approach can therefore be used as an initial screening tool to identify configurations that may require more detailed three-dimensional analysis.

Methods and Materials

A. Problem formulation and test geometry

One of the housing complexes being evaluated in our research has a notable feature: many units in this development are aligned at a 45-degree angle to the street. To understand the effect of this design on the complex's ventilation, the following test problem was formulated.

This test task involved modeling a square two-dimensional area to depict a possible housing development, with two cases examined for comparative analysis. The initial instance comprised a geometric configuration of two square edifices, each measuring 25 x 25, and two rectangular structures, each measuring 25 x 50. The second scenario had an alternative configuration comprising two rhombus-shaped buildings measuring 25 x 25, alongside two rectangular buildings measuring 25 x 50.

The residual conditions are identical for both scenarios. The input was situated along the entire left perimeter of the area, enabling an airflow at a velocity of 1 m/s. The outflow is located along the entire right boundary of this area. The upper and lower limits are characterized by no-slip boundary conditions or barriers.

Each model was computed until a steady-state condition was attained. The ensuing results are presented in Figure 1.

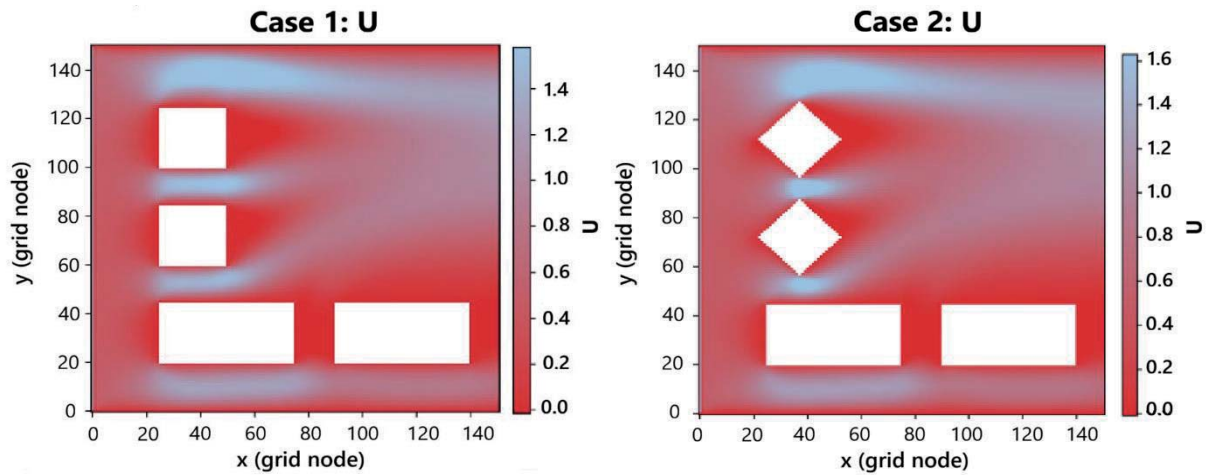


Figure 1. Velocity contours for test building configurations.

Figure 1 shows the velocity directions and velocity contours of U , which corresponds to the velocity along the x -axis. It is important to note that the maximum flow velocity varies across the scenarios under consideration. In the first case, the peak velocity is 1.4 m/s, whilst in the second case it increases to 1.6 m/s, which is 14.3 per cent higher. The results show that changing the building orientation affects the local flow pattern and the maximum velocity predicted by the model. This preliminary test provides a useful numerical check before applying the same framework to compare the three residential configurations.

B. Description of the Mathematical Model

The airflow around the analyzed building configurations was modeled as a two-dimensional, incompressible, viscous flow. The mathematical model was based on the conservation of momentum in two coordinate directions, together with the conservation of mass. The governing equations were expressed in a non-dimensional form to allow for a controlled comparison of the airflow distribution among different building designs.

$$\frac{dU}{dt} + U \frac{dU}{dx} + V \frac{dU}{dy} = -\frac{1}{\rho} \frac{dP}{dx} + \frac{1}{Re} \left(\frac{d^2U}{dx^2} + \frac{d^2U}{dy^2} \right) \quad (1)$$

$$\frac{dV}{dt} + U \frac{dV}{dx} + V \frac{dV}{dy} = -\frac{1}{\rho} \frac{dP}{dy} + \frac{1}{Re} \left(\frac{d^2V}{dx^2} + \frac{d^2V}{dy^2} \right) \quad (2)$$

$$\frac{\partial U}{\partial x} + \frac{\partial V}{\partial y} = 0. \quad (3)$$

Equations (1) and (2) describe momentum conservation in the x - and y -directions, respectively. The first term in each equation represents the local change in velocity with time, while the following two terms describe the convective transport of momentum. The pressure-gradient terms account for the acceleration induced by spatial pressure variations, whereas the terms multiplied by $1/Re$ represent the viscous diffusion of momentum.

Equation (3) is the continuity equation for an incompressible fluid. It requires the divergence of the velocity field to be zero and therefore ensures conservation of mass within the computational domain.

In this context, $U(x, y, t)$ and $V(x, y, t)$ represent the velocity components in the x - and y -directions, respectively. $P(x, y, t)$ denotes the pressure. The spatial coordinates are denoted by x and y , while t represents time. The Reynolds number is denoted by Re .

In the context of the adopted dimensionless formulation, the factor $1/Re$ represents the dimensionless contribution of viscosity to the equations of motion.

The Reynolds number was used to describe the balance between inertial and viscous effects in the flow and was calculated as:

$$Re = \frac{U_{ref} L_{ref}}{\nu} \quad (4)$$

where ν is the kinematic viscosity, U_{ref} is the reference velocity, and L_{ref} is a characteristic length scale. In this study, we fixed the Reynolds number at $Re = 100$ in all simulations. This corresponds to a nondimensional viscous coefficient of $\frac{1}{Re} = 0.01$. The nondimensional density used in the pressure–velocity coupling was taken as $\rho = 1$. These values were kept the same in all cases so that the different configurations could be compared under identical numerical conditions. The prescribed inlet velocity was used as the reference velocity, $U_{ref} = 1$ m/s. Accordingly, the nondimensional velocity components were defined as $U^* = U_{phys} / U_{ref}$ and $V^* = V_{phys} / V_{ref}$. Since $U_{ref} = 1$ m/s, the numerical values of the nondimensional velocities are numerically identical to the corresponding dimensional values expressed in m/s. Local velocities may exceed the inlet value because the building configuration redistributes and locally accelerates the flow around building edges and through inter-building passages.

The boundary conditions were specified in the same way for all considered configurations so that the differences in the calculated flow field were associated mainly with changes in building geometry and layout. A uniform horizontal flow was prescribed at the inlet and directed along the x-axis. The inlet values were set as $U=1$, $V=0$, and $P=1$.

At the outlet, the pressure was fixed at $P=0$. The velocity components were assigned zero-gradient conditions,

$$\frac{\partial U}{\partial x} = 0, \frac{\partial V}{\partial y} = 0, \quad (5)$$

which allowed the flow to leave the computational domain without prescribing an additional velocity value.

The upper and lower boundaries, as well as the surfaces of all buildings, were treated as solid walls with no-slip conditions,

$$U = 0, V = 0 \quad (6)$$

For pressure, a zero-normal-gradient condition was approximated at the solid boundaries using the value of the nearest fluid node. In all simulations, a uniform Cartesian grid was used with spatial steps $\Delta x = \Delta y = 0.01$.

C. Numerical approach

The projection method, originally proposed by Chorin [25], was used to solve the incompressible Navier–Stokes equations. This approach treats the velocity prediction and pressure correction in separate steps. In the present simulations, a uniform Cartesian finite-difference grid with $\Delta x = \Delta y = 0.01$ was used. Cases 1 and 2 were solved in 5×5 computational domains using 501×501 grid points, while Case 3 was considered in a smaller 3×3 domain with 301×301 grid points. Thus, the difference in the number of grid points was related only to the size of the computational domain, since the spatial resolution was kept the same in all cases.

It should be noted that the total number of grid points is not the same as the local spatial resolution. Case 3 contains fewer grid points only because its computational domain is smaller, while the grid spacing remains the same in all three cases, with $(\Delta x = \Delta y = 0.01)$. Therefore, Case 3 was not solved on a coarser grid. However, a separate grid-refinement study

using different spatial step sizes was not carried out, so the possible influence of further mesh refinement on the quantitative results remains a limitation of the present study.

The Reynolds number, boundary conditions, discretization scheme, and pressure–velocity coupling procedure were also kept unchanged.

The time step was set to $\Delta t=0.001$ for Cases 1 and 3, while a smaller value of $\Delta t=0.0001$ was used for Case 2. This case produced the highest local velocities, so a smaller time step was selected to provide a more stable temporal resolution. The maximum Courant numbers obtained from the simulations were about 0.296, 0.040, and 0.239 for Cases 1, 2, and 3, respectively. The corresponding viscous diffusion numbers were 0.20, 0.02, and 0.20. These values indicate that the explicit scheme remained within a reasonable stability range for the selected time steps. However, a separate time-step independence study was not carried out, and this should be considered when interpreting the results.

The three configurations also differ in more than one geometric feature. In addition to the building shape and orientation, the cases include different numbers of buildings as well as different spacing and spatial arrangements. For this reason, the differences observed in the flow fields should be regarded as the combined effect of the overall configuration rather than the influence of a single geometric parameter.

Building structures, encoded as NaN values in the output fields, were automatically excluded from the analysis, ensuring that only points within the fluid domain were considered. The proportion of low-velocity zones was defined as the ratio of the number of low-velocity fraction to the total number of points in the fluid domain, expressed as a percentage. The results are summarised in Table 1.

Table 1. Quantitative comparison of airflow characteristics for three building configurations.

Metric	Case 1	Case 2	Case 3
Grid resolution	501×501	501×501	301×301
Fluid grid points	236,706	242,671	81,704
Max velocity (m/s)	2.120	2.855	2.026
Mean velocity (m/s)	0.745	1.101	0.775
Low-velocity fraction (%)	20.2	11.1	20.6

The momentum equations were solved explicitly in time. Central finite differences were used to approximate the convective terms, while the viscous terms were evaluated using a second-order central difference scheme. The pressure and velocity fields were coupled using the projection method. At each time step, the velocity field was first predicted without including the pressure-gradient term, resulting in an intermediate velocity field.

At the first stage of the projection procedure, the intermediate horizontal velocity U^* was determined from:

$$U_{ij}^* = U_{ij}^n + \Delta t \cdot \left(\frac{1}{Re} \left(\frac{d^2 U}{dx^2} + \frac{d^2 U}{dy^2} \right) - U \frac{dU}{dx} - V \frac{dU}{dy} \right) \quad (7)$$

whereas the intermediate vertical velocity V^* was obtained from:

$$V_{ij}^* = V_{ij}^n + \Delta t \cdot \left(\frac{1}{Re} \left(\frac{d^2 V}{dx^2} + \frac{d^2 V}{dy^2} \right) - U \frac{dV}{dx} - V \frac{dV}{dy} \right) \quad (8)$$

Here, the superscript n denotes the current iteration level, while the superscript “*” denotes the intermediate velocity field obtained before pressure correction. Equations (4) and (5) account for convective momentum transport and viscous diffusion but do not yet include the pressure-gradient contribution.

To enforce the incompressibility condition, the intermediate velocity field was used to formulate the pressure Poisson equation:

$$\frac{d^2 P}{dx^2} + \frac{d^2 P}{dy^2} = \frac{\rho}{\Delta t} \cdot \left(\frac{dU_{ij}^*}{dx} + \frac{dV_{ij}^*}{dy} \right) \quad (9)$$

Equation (10) follows from substituting the velocity-correction equations into the continuity equation. Its right-hand side represents the divergence of the intermediate velocity field. The calculated pressure is therefore used to remove this divergence during the correction stage.

The pressure Poisson equation was solved iteratively using the Jacobi method. For a uniform Cartesian grid, the discretized pressure equation was written as:

$$P_{i,j}^{k+1} = \frac{\Delta y^2 (P_{i,j+1}^k + P_{i,j-1}^k) + \Delta x^2 (P_{i+1,j}^k + P_{i-1,j}^k)}{2(\Delta x^2 + \Delta y^2)} - \frac{\rho \Delta x^2 \Delta y^2}{2\Delta t(\Delta x^2 + \Delta y^2)} \left[\frac{U_{i,j+1}^* - U_{i,j-1}^*}{2\Delta x} + \frac{V_{i+1,j}^* - V_{i-1,j}^*}{2\Delta y} \right] \quad (10)$$

After calculating the pressure field, the intermediate horizontal velocity was corrected according to:

$$U_{ij}^{n+1} = U_{ij}^* - \frac{\Delta t}{\rho} \frac{dP}{dx} \quad (11)$$

and the intermediate vertical velocity was corrected according to:

$$V_{ij}^{n+1} = V_{ij}^* - \frac{\Delta t}{\rho} \frac{dP}{dy} \quad (12)$$

Equations (13) and (14) represent the projection of the intermediate velocity field onto an approximately divergence-free field that satisfies the incompressibility constraint.

The outer iteration was allowed to proceed for up to 30,000 iterations. Convergence was monitored using the maximum absolute change in the horizontal velocity component between two consecutive iterations,

$$E_u = \max_{i,j} |u_{i,j}^{n+1} - u_{i,j}^n| \quad (13)$$

At each pressure-correction step, the pressure Poisson equation was solved using no more than 100 Jacobi iterations. The corresponding pressure convergence measure was defined as

$$E_p = \max_{i,j} |p_{i,j}^{k+1} - p_{i,j}^k| \quad (14)$$

In the original implementation, both convergence criteria were assigned a nominal tolerance of 10^{-100} . Since this value is extremely small, the maximum iteration counts effectively acted as the practical stopping limits in the calculations. A separate convergence assessment based on residual histories was not carried out, and this is considered a limitation of the present numerical approach.

Results

A. Qualitative Analysis

The graph below shows the flow velocity contours for the three cases under consideration. Figure 2 shows the velocity-magnitude fields for the three configurations. In each case, low-velocity zones appear behind several buildings, while higher velocities are observed near exposed edges and in some of the passages between buildings. The size and spatial distribution of these regions differ noticeably from one layout to another.

Furthermore, the noticeable differences in flow structure between the various cases can be explained by the different geometries and layouts of the buildings. In Case 3, the building arrangement leads to a noticeably different airflow pattern. The flow accelerates near the exposed building edges, while broader regions of reduced velocity develop downstream of the buildings.

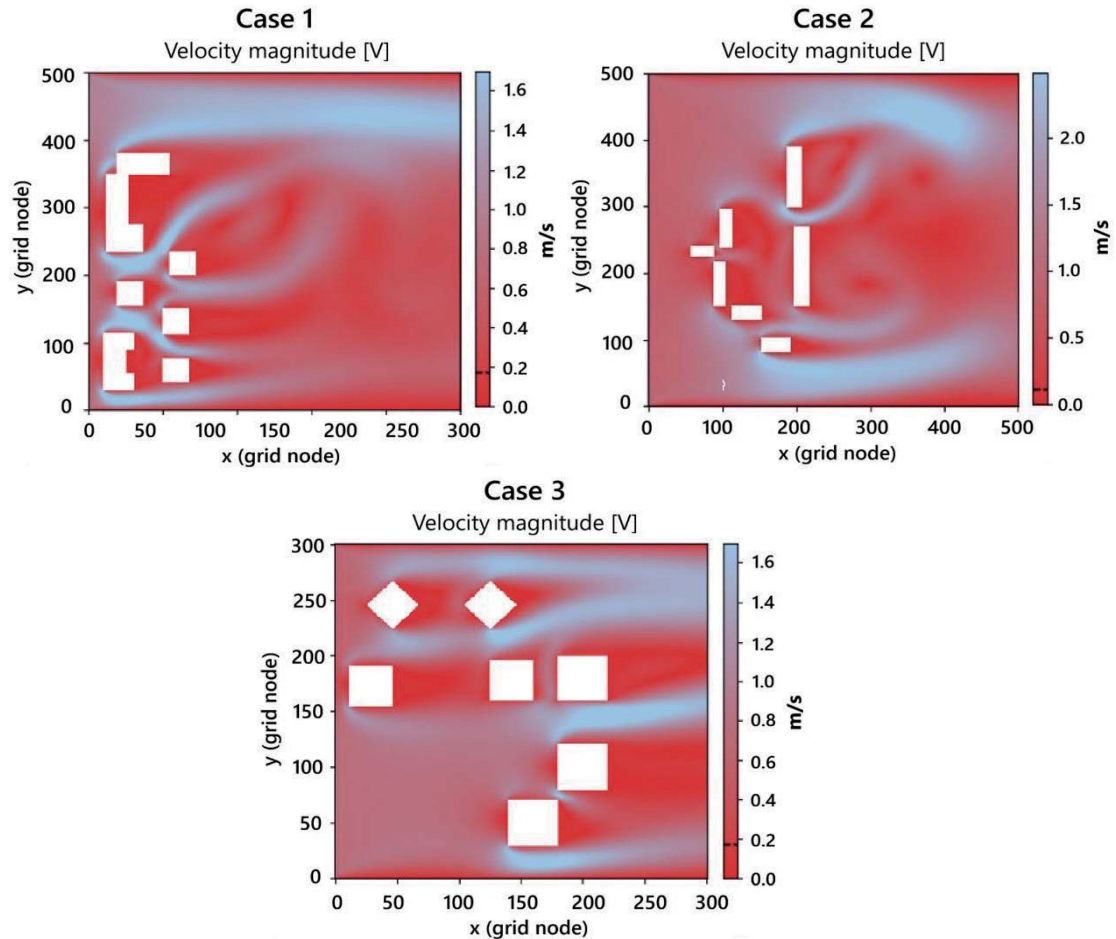


Figure 2. Two-dimensional velocity magnitude contours and streamlines for three building layout configurations.

Several recirculation zones develop behind the buildings, with the strongest vortical motion observed in Case 2. These flow structures alter the local velocity field and redistribute momentum within the spaces between the buildings. Such changes may affect the pathways along which pollutants are transported. However, pollutant concentrations were not included in the present simulations, so the results do not allow direct conclusions to be made about pollutant dispersion or concentration reduction.

The results indicate a change in the direction of the airflow. Whereas in the two previous cases the space between the houses was filled with areas of low air velocity, in this scenario a uniform, straight-line movement of air masses is observed. The velocity is approximately 1.2 m/s. This flow pattern improves ventilation and air extraction. It is important to note the areas adjacent to the right-hand side of each residential building, where local vortices occur, leading to low-velocity of the air.

A comparative analysis of the three designs shows that the orientation and shape of the buildings significantly influence the flow pattern. The second configuration demonstrates the most favourable ventilation conditions, characterised by a more uniform velocity distribution and a reduction in low-velocity zones compared with the first two cases.

B. Quantitative Analysis

In addition to the qualitative analysis of the velocity vector fields presented in the previous section, a quantitative analysis of air low-velocity zones was carried out for all three residential area configurations. In this study, regions with a velocity below 0.3 m/s were treated as weakly ventilated areas. This value was selected based on criteria reported in earlier urban ventilation studies [26]. It should be noted that the threshold is used only for comparison between the investigated cases and does not represent a universal definition of complete flow low-velocity.

$$U_{\text{mag}} \leq 0.3 \text{ m/s} \quad (15)$$

were classified as belonging to low-velocity zones. This threshold was introduced to provide a consistent quantitative criterion for comparing the three building configurations. It should not, however, be interpreted as a universal physical definition of complete airflow low-velocity. Airflow at velocities below 0.3 m/s remains non-zero, and the classification therefore represents weakly ventilated areas rather than regions of strictly stationary air.

The velocity magnitude U_{mag} at each fluid grid point was calculated from the two velocity components as

$$U_{\text{mag}} = \sqrt{U^2 + V^2} \quad (16)$$

Table 2 summarizes the main geometric and computational parameters used for the three configurations. The occupied and open-area fractions are also reported to show the differences in plan-area density between the layouts.

Table 2. Quantitative airflow characteristics of the investigated building configurations.

Indicator	Case 1	Case 2	Case 3
Maximum velocity (m/s)	2.120	2.855	2.026
Mean velocity (m/s)	0.745	1.101	0.775
Median velocity (m/s)	0.617	1.000	0.790
Velocity standard deviation (m/s)	0.499	0.657	0.465
Coefficient of variation	0.670	0.596	0.601
Low-velocity area fraction (%)	20.18	11.12	20.61
Area (%)	29.97	50.03	28.93
Reverse-flow area (%)	5.72	11.52	7.92
Mean speed within reverse-flow regions (m/s)	0.184	0.408	0.122
Nondimensional pressure range	6.026	10.383	5.112

The three configurations differ noticeably in the amount of area occupied by buildings. Case 2 has the lowest occupied-area fraction, 3.32%, and therefore the largest open-area fraction, 96.68%. In contrast, Case 3 has the highest occupied-area fraction, reaching 9.82%. These differences indicate that the observed ventilation patterns are influenced not only by building shape and orientation, but also by spacing, overall layout, and the amount of open space available for airflow.

The additional quantitative results further highlight the differences between the three configurations. Case 2 has the highest median velocity, 1.000 m/s, and the largest share of the fluid domain where the velocity exceeds the reference inlet value. This fraction reaches 50.03%, compared with 29.97% in Case 1 and 28.93% in Case 3. At the same time, only 11.12% of the domain in Case 2 falls within the low-velocity range, which is about 45% lower than in Case 1. Overall, these values show that a much larger part of the flow field in Case 2 is characterized by relatively high velocities.

The reverse-flow regions also differ between the cases. In Case 2, 11.52% of the grid points have a negative streamwise velocity component, whereas the corresponding values are 5.72% for Case 1 and 7.92% for Case 3. Despite this larger reverse-flow area, the mean velocity magnitude within these regions is also higher in Case 2, reaching 0.408 m/s compared with 0.184 m/s in Case 1 and 0.122 m/s in Case 3. This suggests that the reverse-flow zones in Case 2 are associated with relatively strong recirculating motion rather than broad stagnant regions. It also explains how pronounced vortical structures can occur together with a relatively small fraction of low-velocity flow.

Case 2 also shows the largest nondimensional pressure difference, with ($\Delta P=10.383$), while the corresponding values for Cases 1 and 3 are 6.026 and 5.112, respectively. The wider pressure range is associated with stronger changes in flow velocity and direction near building edges and within the passages between buildings. When considered together with the velocity results, this indicates that momentum redistribution is more pronounced in Case 2.

The quantitative comparison in this study is based on velocity-related indicators, which are sufficient for the intended screening of horizontal ventilation patterns within the simplified two-dimensional model. Maximum velocity was used to identify local flow acceleration, while the domain-averaged velocity provides an overall measure of airflow intensity. The fraction of low-velocity regions was included to show how much of the domain remains weakly ventilated. Other quantities, such as pressure-based indices, vorticity, turbulence characteristics, air-exchange effectiveness, and pollutant-removal efficiency, were not considered in the present analysis and may be included in future extensions of the model.

The results of the quantitative analysis reveal significant differences between all configurations. Cases 1 and 3 showed nearly the same proportion of low-velocity regions, accounting for 20.2% and 20.6% of the fluid domain, respectively. Case 1 had the lowest mean velocity, with a domain-averaged value of 0.745 m/s. In Case 3, the mean velocity was slightly higher at 0.775 m/s, although this configuration had the largest fraction of low-velocity flow. For Case 1, the arrangement of the L-shaped and rectangular buildings created relatively large wake regions downstream, where the airflow remained weak.

In Case 2, the rectangular layout produced the smallest low-velocity area, 11.1%, together with the highest domain-averaged velocity of 1.101 m/s. The more open arrangement between successive buildings allows the flow to pass through the layout with fewer interruptions. Flow separation and wake formation still occur near and behind the buildings, but these low-velocity regions remain more limited and are better connected to faster-moving parts of the flow. As a result, weakly ventilated zones occupy a smaller portion of the domain than in Cases 1 and 3.

In Case 1, the L-shaped buildings create several sheltered regions on their leeward sides. Flow separation at the exposed corners and the interaction between neighboring wakes reduce the penetration of higher-momentum air into some of the spaces between the buildings. This is consistent with the lowest domain-averaged velocity obtained in the comparison, 0.745 m/s.

In Case 3, rotating several buildings changes the orientation of the surfaces exposed to the incoming flow. Higher velocities appear locally around some exposed corners, while the resulting wakes are deflected and interact farther downstream. This produces relatively broad low-velocity regions, and the low-velocity area fraction reaches 20.6%. Thus, the local increase in velocity observed near some building edges does not correspond to better ventilation across the whole domain.

The spatial distribution of low-velocity zones for all three configurations is shown in Fig. 3, together with velocity maps and histograms of the velocity distribution. The low-velocity zone maps clearly corroborate the quantitative results: in Case 2, the low-velocity zones are concentrated mainly near the building walls, whereas in Cases 1 and 3 they occupy significantly larger areas of the space between the buildings.

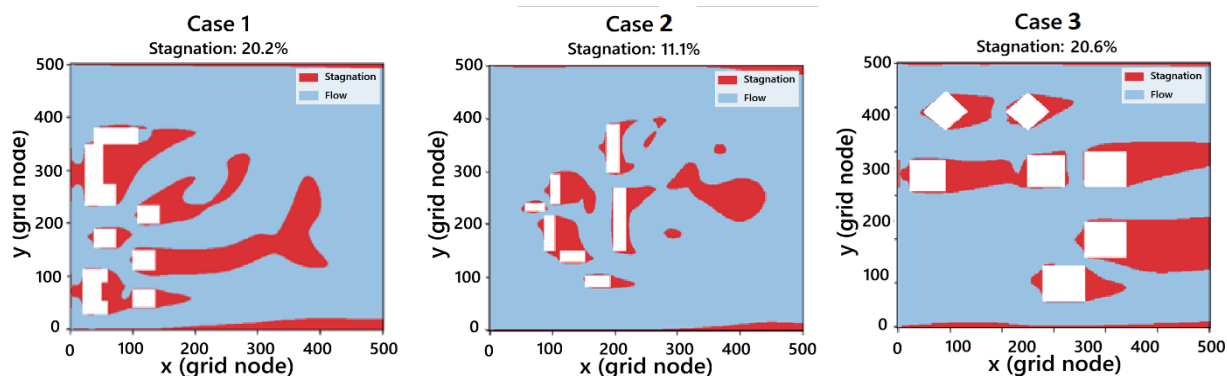


Figure 3. Spatial distribution of low-velocity regions in the three investigated configurations.

From an engineering perspective, the results suggest that early-stage layout assessment should consider not only local peak velocities but also the spatial extent of weakly ventilated regions. Configurations that provide continuous airflow passages and avoid large sheltered leeward cavities may offer more favourable horizontal ventilation characteristics. The proposed numerical framework can therefore be used to screen alternative layouts and identify designs that should be examined further using three-dimensional CFD before final architectural decisions are made.

Conclusion

The aim of the study was to carry out numerical modelling of wind flows to compare three configurations of residential neighbourhoods with different building shapes. An assessment was made of the impact of various geometric shapes and building orientations on the effectiveness of natural ventilation, with particular attention paid to the analysis of areas of stagnant air. Before the residential layouts were analysed, a simple two-dimensional test case was considered to check the consistency of the numerical implementation. The test examined how the model responded to a controlled change in building orientation. It was used only as an internal numerical check and not as an independent experimental or benchmark validation. This confirmed the need for a systematic comparison of configurations before moving on to full-scale models of residential neighbourhoods. Based on the results of the main calculations, a quantitative analysis of low-velocity zones was carried out, which revealed the following. Low-velocity zones in variant 1 (L-shaped and rectangular buildings) and variant 3 (diamond-shaped and square buildings) were similar in terms of percentage share (20.2% and 20.6%, respectively) at average air velocities of around 0.745–0.775 m/s. The highest ventilation efficiency was found in variant 2, consisting exclusively of rectangular buildings. The proportion of the stagnant zones in this case was 11.1%, 45% lower than that of in Case 1. The average air velocity was 1.101 m/s, which was almost one and a half times higher than the corresponding value for other configurations. One of the key findings is that a high peak wind velocity does not always correlate with a high degree of natural ventilation. In Case 3, rotating the buildings changed the local flow pattern and increased the velocity near exposed building edges. However, this local acceleration did not lead to a reduction in the overall low-velocity area. This shows that local increases in airflow cannot, by themselves, be used to assess the ventilation performance of the entire residential layout. In other words, this shape of the residential complexes creates geometrically complex leeward zones where air exchange remains insufficient. This indicates that, when designing residential areas, the main focus should be not so much on maximising peak velocities as on ensuring an even distribution of airflow across the entire development site. The results show that numerical modelling based on the Navier–Stokes equations, together with simple quantitative airflow indicators, can be used as an initial tool for comparing the ventilation performance of different residential layouts during the early design stage. These results also demonstrate the potential of computing and information technologies in the field of environmental monitoring and decision-making. Numerical modeling

can serve as the computational foundation for future intelligent air quality assessment systems by integrating modeling results with environmental data and real-time monitoring technologies.

The present study was carried out as a comparative numerical analysis rather than as a full-scale atmospheric CFD prediction. Its main aim was to compare airflow behaviour among different residential building layouts under the same numerical conditions, including the governing equations, Reynolds number, boundary conditions, spatial resolution, and solution procedure.

No turbulence model was included in the simulations, so the model does not reproduce the full range of turbulent fluctuations and momentum exchange found in real atmospheric flow around buildings. The numerical results were also not validated against wind-tunnel measurements or a standard benchmark case, and separate grid- and time-step independence studies were not performed. For these reasons, the results should be interpreted mainly as relative differences and general airflow trends between the investigated configurations, rather than as exact predictions of full-scale urban flow.

Future work should extend the analysis to three-dimensional simulations, include an appropriate turbulence model, and examine grid and time-step sensitivity in a systematic way. Experimental or benchmark validation would also be needed before the model could be used for full-scale predictive applications.

Acknowledgment

This research has been funded by the Committee of Science of the Ministry of Science and Higher Education of the Republic of Kazakhstan (Grant No. AP26101793 “Modeling of the spread of greenhouse gases and forecasting the impact of the level of pollution on the environment using numerical methods and artificial neural networks”).

References

- [1] Health Effects Institute. (2025). State of Global Air 2025: A report on air pollution and its role in the world's leading causes of death. <https://www.stateofglobalair.org/resources/report/state-global-air-report-2025>
- [2] European Environment Agency. (2025). Air pollution. <https://www.eea.europa.eu/en/analysis/publications/air-quality-status-report-2025>
- [3] European Environment Agency. (2025). Harm to human health from air pollution in Europe: Burden of disease status 2025 (EEA Report). <https://www.eea.europa.eu/en/analysis/publications/harm-to-human-health-from-air-pollution-burden-of-disease-status-2025>
- [4] Ganguly, T., Selvaraj, K. L., & Guttikunda, S. K. (2020). National Clean Air Programme (NCAP) for Indian cities: Review and outlook of clean air action plans. *Atmospheric Environment: X*, 8, Article 100096. <https://doi.org/10.1016/j.aeaoa.2020.100096>
- [5] European Commission. (2025, March 6). EU advancing on 2030 zero pollution targets, but stronger action is needed. EU Reporter. <https://www.eea.europa.eu/en/newsroom/news/eu-advancing-on-2030-zero-pollution-targets-but-stronger-action-is-needed>
- [6] State Council Information Office of the People's Republic of China. (2018, March 23). China's actions on air pollution prevention and control. http://english.scio.gov.cn/m/featured/chinakeywords/2023-03/20/content_85178625.htm
- [7] U.S. Environmental Protection Agency. (n.d.). Summary of the Clean Air Act. <https://www.epa.gov/laws-regulations/summary-clean-air-act>
- [8] Government of Japan. (2024). The Sixth Basic Environment Plan. Climate Policy Database. <https://climatepolicydatabase.org/policies/6th-basic-environment-plan>
- [9] Jeong, J. I., Park, R. J., Song, C.-K., Yeh, S.-W., & Jung-Hun, W. (2024). Quantitative analysis of winter PM_{2.5} reduction in South Korea, 2019/20 to 2021/22: Contributions of meteorology and emissions. *Science of The Total Environment*, 907, Article 168179. <https://doi.org/10.1016/j.scitotenv.2023.168179>

- [10] UK Department for Environment, Food & Rural Affairs. (2019). Clean Air Strategy 2019. <https://www.gov.uk/government/publications/clean-air-strategy-2019>
- [11] Bui, D. H., & Nghiem, X.-H. (2024). Urbanization and fine particulate matter (PM_{2.5}): Empirical evidence from developing regions. *Asian Journal of Water, Environment and Pollution*. <https://accscience.com>
- [12] Niu, L., Zhang, Z., Liang, Y., & van Vliet, J. (2024). Spatiotemporal patterns and drivers of the urban air pollution island effect for 2273 cities in China. *Environment International*, 184, Article 108455. <https://doi.org/10.1016/j.envint.2024.108455>
- [13] Yedilkhan, D., Salesnova, S., Omarova, G., Akhmetzhanov, B., Sarsenova, Z., & Amirgaliyev, B. (2025). AI-driven urban analytics using IoT-enabled mobile sensor networks for environmental monitoring. *Procedia Computer Science*, 272, 619-625. <https://doi.org/10.1016/j.procs.2025.10.257>
- [14] Yedilkhan, D., Sarsenova, Z., Yermekov, A., & Salesnova, S. (2025, May). Integrating meteorological and air quality data for advanced correlation analysis in urban environments: A case study of Astana. In *2025 IEEE European Technology and Engineering Management Summit (E-TEMS)* (pp. 270-275). IEEE. <https://doi.org/10.1109/E-TEMS64751.2025.11239125>
- [15] Jiang, Z., Cheng, H., Zhang, P., & Kang, T. (2021). Influence of urban morphological parameters on the distribution and diffusion of air pollutants: A case study in China. *Journal of Environmental Sciences*, 105, 163-172. <https://doi.org/10.1016/j.jes.2020.12.035>
- [16] Augusto, B., Lopes, D., Rafael, S., Coelho, M. C., & Ferreira, J. (2024). Assessing the impact of different urban morphology scenarios on air pollutant emissions distribution. *Science of The Total Environment*, 950, Article 175341. <https://doi.org/10.1016/j.scitotenv.2024.175341>
- [17] Omarova, P., Yang, T., Ataniyazova, A., Kozbakova, A., & Yedilkhan, D. (2026). Assessment of air quality in the urban environment taking into account vegetation and building geometry. *Journal of Ecological Engineering*, 27(2), 388-400. <https://doi.org/10.12911/22998993/211688>
- [18] Vahidi, S., Yeganeh, M., & Ghasaban, M. (2025). The effect of the morphology of highly polluted urban areas neighborhood on suspended particles of PM 2.5 and PM 10 air pollutants (Case Study: Ahvaz, Iran). *Energy Nexus*, 20, Article 100527. <https://doi.org/10.1016/j.nexus.2025.100527>
- [19] Sun, H., Bensalem, R., Dik, A., Tao, Z., Wang, Z., Jimenez-Bescos, C., & Calautit, J. K. (2025). The impact of courtyard roof shape on adjacent building natural ventilation and passive cooling. *Journal of Building Engineering*, 111, Article 113331. <https://doi.org/10.1016/j.job.2025.113331>
- [20] Biloshchytskyi, A., Kuchanskyi, O., Andrashko, Y., Yedilkhan, D., Neftissov, A., Biloshchytska, S., ... & Vatskel, V. (2023). Reducing outdoor air pollutants through a moss-based biotechnological purification filter in Kazakhstan. *Urban Science*, 7(4), Article 104. <https://doi.org/10.3390/urbansci7040104>
- [21] Omarova, P., Merembayev, T., & Amirgaliyev, Y. (2023). Mathematical modeling of water movement during a dam break using the vof method. *Scientific Journal of Astana IT University*, 14, Article 14. <https://doi.org/10.37943/14NEBW7927>
- [22] Kurmanbek, B., Merembayev, T., & Amanbek, Y. (2024). Prediction of natural fracture network patterns using feature engineering and machine learning approaches. *Computational Energy Science*, 1(4), 167-174. <https://doi.org/10.46690/compes.2024.04.02>
- [23] Yedilkhan, M., Berdyshev, A., Galiyev, M., & Merembayev, T. (2025). Air quality prediction based on the lstm with attention using meteorological data in urban area in kazakhstan. *Journal of Problems in Computer Science and Information Technologies*, 3(1), 3-12. <https://doi.org/10.26577/jpcsit20253101>
- [24] Rakhimberdina, A., Ormanova, G., & Yedilkhan, D. (2025, May). Development of a prediction model for the assessment of air quality in the city of Astana, Kazakhstan. In *2025*

- IEEE European Technology and Engineering Management Summit (E-TEMS) (pp. 303-308). IEEE. <https://doi.org/10.1109/E-TEMS64751.2025.11239309>
- [25] Chorin, A. J. (1968). Numerical solution of the Navier-Stokes equations. *Journal of Computational Physics*, 2(4), 745-762. <https://doi.org/10.2307/2004575>
- [26] Yuan, C., & Ng, E. (2012). Building porosity for better urban ventilation in high-density cities – A computational parametric study. *Building and Environment*, 50, 176–189. <https://doi.org/10.1016/j.buildenv.2011.10.023>