Proceedings of 12th IOE Graduate Conference
Peer Reviewed
ISSN: 2350-8914 (Online), 2350-8906 (Print)
Year: 2022 Month: October Volume: 12

Analysis of wind flow in Lumbini Sanskritik Municipality through ANSYS Fluent

Biplav Pokhrel [a], Sanjaya Uprety [b], Niranjan Bastakoti [c]

[a, b] *Department of Architecture, Pulchowk Campus, IOE, Tribhuvan University, Nepal*
[c] *Department of Mechanical Engineering, Kathmandu University, Nepal*
✉ [a] 076mseeb006.biplav@pcampus.edu.np, [b] suprety@ioe.edu.np, [c] niranjan.bastakoti@ku.edu.np

Abstract

Urban Planning and Design plays a major role in affecting the micro as well as macro climate of an area especially in terms of wind environment. An area with proper wind circulation not only reduces the thermal discomfort for the pedestrians but also removes the suspended pollutants in the area. The orientation and planning of buildings can significantly change the wind flow behavior within a settlement by acting as wind breakers as well as funnels. The wind breakers reduce the wind velocity by acting as a barrier while by being a funnel, it increases the wind velocity exponentially as verified by principle of continuity. So, it is imperative that wind environment must be strongly considered during the urban planning and design phase of newer settlements. This study is focused on understanding and analyzing the wind environment of the settlements around the road section of 100m of the road connecting Majhediya to Mahilwaar. The research was carried out through field wind data measurement as well as CFD simulation and modelling. This research shows that the building arrangement in the site selected is negatively affecting the wind behavior within the settlement by continuously acting as a barrier to the wind flow which deteriorates the wind environment of that region. The wind flow affected by the buildings results in a negligible flow of wind through the settlements leading to suffocating feeling as personally experienced by the author during field visit. In order to improve the wind circulation, development of wind corridors is imperative in future development works in Lumbini Sanskritik Municipality.

Keywords

CFD, ANSYS Fluent, Stagnation Points, Pedestrian Comfort, Setbacks

1. Introduction

Wind is a key factor in determining urban thermal environments [1]. It helps in maintaining air circulation along with transport of suspended particles throughout the volume. A high-density development in urban areas, affects the ventilation of buildings as well as the comfort and safety of pedestrians. Tall and bulky high-rise building blocks with limited open spaces between them, uniform building heights, and large podium structures lead to lower permeability for urban air ventilation at the pedestrian level [2].

Wind corridors are basically channels present between buildings which can result from roads, open spaces, and passages through which air reaches the interiors of urbanized areas. Building configuration (e.g., height, width, arrangement, and density) are the significant factors affecting wind at ground level [3]. Building arrangement is an important consideration for planners and designers seeking to improve the wind environment. Therefore, it is necessary to understand wind circulation patterns around buildings and take them into account in urban design [4].

The case study of this research lies in the Terai belt of Nepal where the implementation of modern urban planning concepts is still in its verdant state or even nonexistent. The uneven development of building infrastructures has played a major role in obstructing the usual wind path in the region resulting in inefficient circulation of wind across the settlements all over the year. This research aims to understand the present scenario of wind circulation within a settlement and factors affecting the flow in it. So, the research has carried out a Computational Fluid Dynamics (CFD) simulation of the present-day arrangement of buildings in the Lumbini Sanskritik

Municipality on the road section connecting Majhediya to Mahilwaar village in ANSYS Fluent. Among many CFD software such as Abaqus, SimScale, ANSYS Fluent is more accurate and useful in professional simulation in industries. The drawbacks being that it consumes more space and computation memory than other software and the output are still presented in rudimentary graphs [5].

The simulation is used to understand the present condition of wind circulation in the section of buildings which lie in the major commercial hub of the area constantly frequented by the locals throughout the day during the summer season.

2. Urban Ventilation

Urban ventilation is produced by carefully arranging a location's building layouts and adding appropriate landscape to boost the cooling effect. Urban ventilation is influenced by a wide range of urban geometrical factors, including frontal area density, plan area density, aspect ratio of the urban morphology, and others. For instance, it has been demonstrated that different building heights can improve air quality, but bigger urban canyon aspect ratios might result in higher pollution concentrations inside the roadway. Proper wind flow distribution within a community is greatly influenced by the building spacing and proportions, including height and breadth in the direction of the wind as well as perpendicular to the wind [6]. Urban ventilation corridors are defined in clear, precise terms in the German national guideline ”Environmental meteorology climate and air pollution maps for cities and regions (VDI 3787-Part 1)” as ”The area for the mass transport of air near the ground which is preferred due to direction, nature of the surface, and width.” The urban ventilation corridor is also known as ”Kaze-no-Michi” in Japan. Researchers there acquired this name after studying Germany's use of ventilation corridors [7].

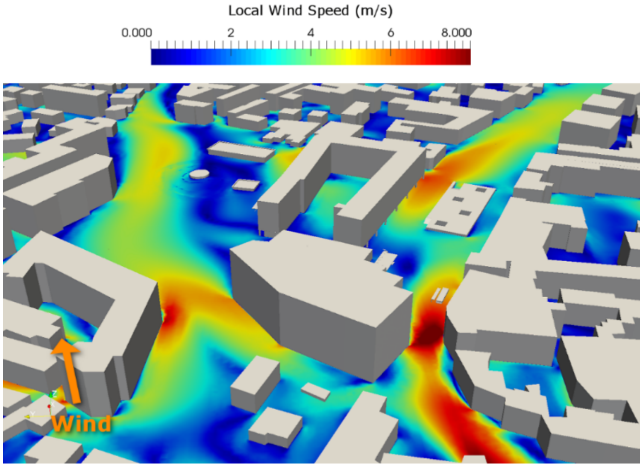

Figure 1: Wind Corridor design in CFD (fyfluiddynamics.com)

Bernoulli's theorem states that in locations where winds are often slow, wind corridors can exponentially speed up the wind flow within a settlement. In urban areas, high-density growth affects building ventilation as well as pedestrian comfort and safety. Big podium structures, constant building heights, and towering, bulky high-rise building blocks with little space between them result in lower permeability for urban air ventilation at the pedestrian level [2]. By carefully designing and creating wind corridors, which will enable breezes from the suburbs to be channeled into core areas and stimulate an exchange of fresh air between inner and suburban zones, it is feasible to ensure that cities' natural air routes remain unaltered.

It has been shown that the shape and direction of the street canyon affect both its capacity to cool the whole metropolitan system and its permeability to airflow for urban ventilation. The wind flow pattern is impeded and pathways diverge from the wind direction due to erroneous building placements and trees. This results in erratic wind patterns, which in turn cause the atmosphere to warm up and gather pollutants. As a result, airflow in urban areas is slower than in nearby rural regions, and it is also possible to draw the conclusion that deep street canyons have slower airflow than uniform or shallow ones [8].

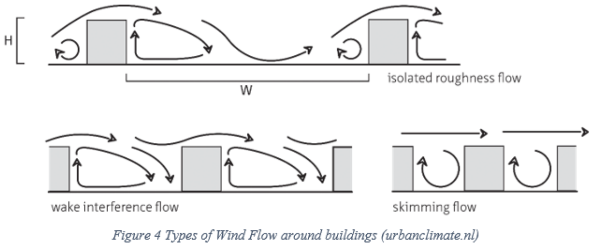

Figure 2: Type of wind flow around buildings (urbanclimate.nl)

Many studies have been carried out on determining the effect of street designs on air flow. For example, [9] examined for more than 1.5 years, using real site data, the impact of street layout on airflow in the cities of Morocco. Aspect ratios of 9.7 and 0.6 for deep and shallow street canyons, respectively, were carefully examined. The findings demonstrated a clear connection between the geometry of street canyons and the micro-climate within urban canyons (1.7 m). This study demonstrates that, throughout both the winter and summer seasons, wind speeds are slower and steadier in the deep canyon (0.4 m/s). The average wind speed in the narrow street canyon was 0.8 m/s in the winter and 0.7 m/s in the summer. Another research conducted in Dubai found that streets with canyons little wider than 4 meters might enhance wind speed across them, improving passive cooling performance while producing eddies at bending angles. Higher wind speeds (5 m/s) allowed for greater penetration into the traditionally small streets, improving the possibility of thermal comfort[10]. The majority of locations (49–57 percent of the study region) with street canyons aspect ratios of 2-0.67 experienced light to moderate breezes. In certain studies, the effects of building heights on street canyon airflow were also evaluated. When the airflow is parallel or perpendicular to the street canyon, it has been discovered that strategically adding a few blocks of high-rise buildings would increase the velocity within the canyon.

More vertical flow up from the street canyon to the urban boundary layer would result from towering structures upstream. Additional vertical flow from the urban border layer into the urban canopy layer would be caused by structures further downstream. Additionally, allowing for sufficient apertures between streets and courts enhances air circulation inside the urban canopy layer. The layout of the roadway may impact the air flow at the canopy layer in addition to the geometry and direction of the street. The circulation of air into and within metropolitan areas would be facilitated by streets that are straight and parallel to one another. Narrow and curving roadways lessen the impact of cold, hot, and severe winds [8].

3. Methodology

This study was carried out in three phases where the first phases consisted of field data measurement where velocity of wind was measured in different sections of the settlement with the help of a standard anemometer (ERICKHILL HT625C) for about 7 days in different times of day. The buildings within the selected site were measured for their 3D dimensions along with the spacing between each building with the help of laser measure (NOYAFA NF 271) and a standard 50 m measuring tape (Fibreglass Measuring Tape). For validation of building layouts, the 2D plans were downloaded from Cadmapper. The buildings had a minimum height of 6m to a maximum of 15m while their spacing varied from 1.5m to 5m.

The second phase of the research consisted of 2d modelling of the measured data in SOLIDWORKS which is highly efficient in development of models and plans for CFD simulations. The buildings were carefully modelling using the exact field data in order to obtain as accurate results as possible in the simulation for a precise simulation. For the ease of CFD analysis, the buildings were created inside a domain of length 300m and a width of 200m using a basic thumb rule. The general thumb rule of domain development also supports the idea that the length and width of the domain should be around 10-15 times the maximum height of the object that is to be placed inside it [11].

The third phase consists of CFD simulation in ANSYS Fluent of the model created in SOLIDWORKS. The simulation was carried out by providing the boundary conditions as measured from site and setup values as required for the condition. The North and the West face of the domain were considered as velocity inlet with wind speeds of 6m/s and 4m/s respectively while the south and the east face were simulated as pressure outlets for convection flow. Since this was a large-scale simulation the mesh size was kept at 0.5m and the simulation was carried out for about 500 iterations for increased accuracy. The fluid taken was air and it was considered inviscid for quicker calculations and also because inviscid flow analysis neglects the effect of viscosity on the flow and are appropriate for high-Reynolds-number applications where inertial forces tend to dominate viscous forces [12]. The buildings were supposed as aluminum to ignore the friction loss in the building surface as friction loss plays a big role in the flow of a fluid in contact with a solid.

The results were obtained in graphical formats with the post processing window of ANSYS Fluent in the form of contours, streamlines and vectors to facilitate the analysis of simulations results. The no of contours was kept at 1000 while the no of streamline points was kept at 10,000 for increased accuracy and better visualization of results.

4. Study Area

4.1 Lumbini Sanskritik Municipality

The location chosen for the study is in Nepal's Lumbini Province's Rupandehi District, which is located in the country's hot and humid climate. The site selected lies at an elevation of 150m (490 ft) from MSL with the coordinates of (27°28'53"N; 83°16'33"E). In Lumbini, the wind is typically quiet. May has the most wind, followed by June and July. Winds average approximately 6.4 knots (7.3 MPH or 11.8 KPH) in May. Late April is the time of year when maximum sustained winds are most common, with average top sustained speeds reaching 15 knots, which is regarded as a moderate breeze [13]. The average temperature in summer reaches as high as 40.1°C while the winters here is quite normal where the minimum temperature is around 26.8°C. The site selected has 48 houses within which the simulation and the field data measurements have been carried out primarily made out of concrete. The area consists mainly of commercial housings and government offices which make this section a highly frequented area by the locals each day of the week .

Figure 3: Site Selection Top View (Google Earth)

5. Simulation and Findings

The simulation of the site was carried out in two parts due to availability of limited computing power and memory. The sites were selected in Autodesk AutoCAD 2022 from the site map acquired from Cadmapper and then modelled in Solidworks 2022. The simulation was carried out in two parts as mentioned before with the building arrangements North of the road taken as top plot and buildings south of the road as Bottom plot. The plots are modelled and simulated separately as the road section was wide enough to negate any effects entailed by one plot on the other as observed in the field. The simulation was done as a 2D format to increase computation efficiency and reduce computation time. The 2D simulation gives us an ease of understanding the basic fluid flow (wind movement) over a large mass of land with comfortably more numbers of buildings included in the simulation. While the height to the building may result in draft movement of the wind which increases the wind speed in the pedestrian level, the buildings were not that high enough to be considered as the maximum height was measured to be around 15m from the ground level of a single building in that vicinity while majority of the building were of around 10m in height. Since, this research primarily prioritizes on building arrangement and orientation in the wind direction, the height of building was not selected as a primary factor that affects the research simulation. The randomness of the building arrangement and orientation was considered before selecting the required no of buildings in the plots North and South of the road. The plots selected cover a large variety of building orientation and arrangements along with their planar dimensions in the simulation. The combined effects of both plots were not studied due to limited availability of computation power.

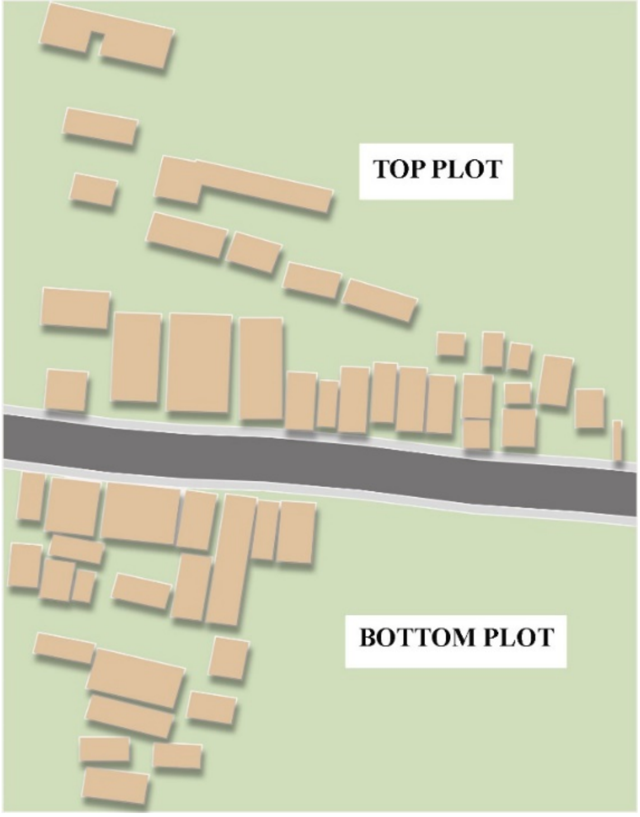

Figure 4: Site Selection

5.1 Wind Velocity Measurement

The data measured in the field from 16th June to 19th June in different interval of time in both West and North direction shows that the average wind velocity along the west is around 3.6 m/s while it was 6.4m/s. The observed wind flow was very much negligible inside the settlement when compared to open spaces around the settlements where the wind was flowing strongly.

The results obtained are summarized in Table 1.

Table 1: Wind Velocity Measurement

S.N.	Date	Instrument	Direction	Magnitude m/s
1	16th June,2022	Anemometer	W/N	3.5/6
2	17th June,2022	Anemometer	W/N	3.6/6.4
3	18th June,2022	Anemometer	W/N	3.8/6.7
4	19th June,2022	Anemometer	W/N	3.4/6.4

5.2 Modelling in SolidWorks

The plots thus divided and selected in Autodesk AutoCAD were then modelled in Solidworks interface as 2D models on Top Plane. The buildings were created as a 2D sketch inside a 2D domain of dimension (L X B) 300m x 150m so that the domain completely incorporates the buildings and proper wind flow can be provided over the required area. The domain was made a bit bigger than the plot area so as to monitor the wind flow at a distance away from the building surfaces and along the building surface as well to observe the difference in values such as pressure, velocity and wind direction. The building surfaces along with the edges create different flow conditions to the wind thus for uniformity of flow bigger domain was preferred over a tight fit domain. The general thumb rule of domain development also supports the idea that the length and width of the domain should be around 10-15 times the maximum height of the object that is to be placed inside it [5]. The final sketch was then exported as IGIS (.IGS) format that is readable by ANSYS Fluent Geometry interface.

5.3 Simulation in ANSYS Fluent

5.3.1 Results and Findings of Simulation

For the final results of site simulation in ANSYS Fluent, there are mainly 3 results and their charts that need to be developed and analyzed in ANSYS. The charts Velocity, Pressure Contour, Velocity Streamline.

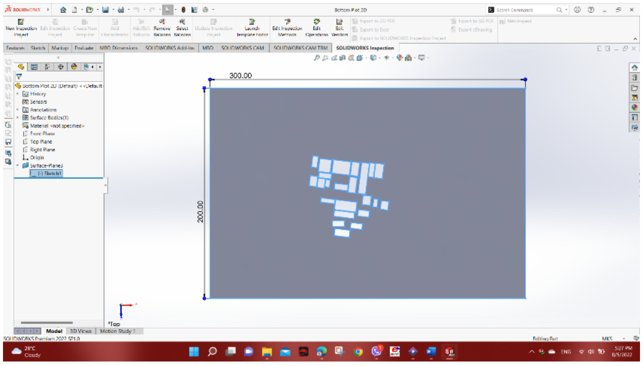

Figure 5: Solidworks Interface for modelling

Contour lines are lines of constant magnitude for a selected variable (here, Pressure and Velocity). A profile plot draws these contours projected off the surface along a reference vector by an amount proportional to the value of the plotted variable at each point on the surface. Streamline is use to mark the path taken by wind particles where the velocity is tangential to the path [14].

1. **Velocity Contour**

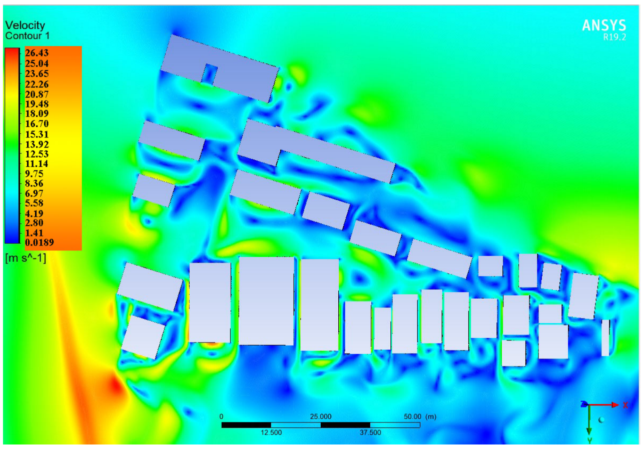

Figure 6: Velocity contour of Top Plot

As seen from figure 6 and 7, the velocity contour of the top plot shows that apart from the immediate region of wind flow, the areas in and round the buildings observe very low wind velocity 1.5m/s which is quite slow when compared to the average wind speed of the region. The situation is even worse here because of blocking of wind by the top plot buildings. This results in even lesser velocity of wind flow in and around the buildings of the region with as low as 1.2m/s with occasional

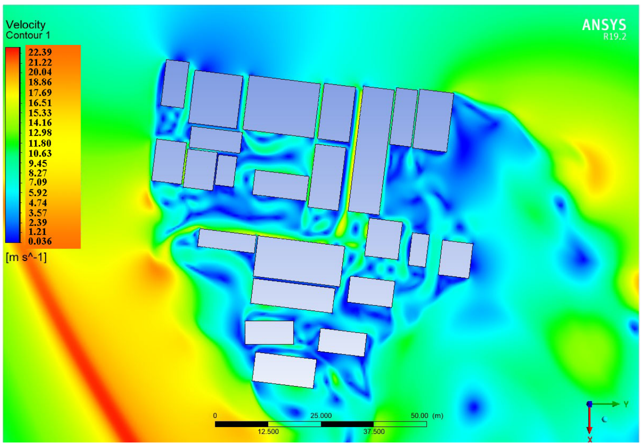

Figure 7: Velocity contour of Bottom Plot

speed of 16.5m/s between the buildings primarily resulting from funnel effect which increases the wind speed considerably following the Bernoulli's principle.

This haphazard building arrangement and orientation towards the wind direction results in lower wind flow pressure and velocity in the region.

2. **Velocity Streamline**

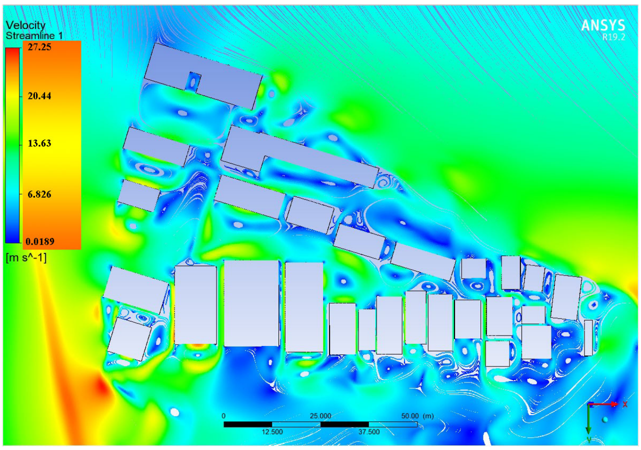

Figure 8: Velocity Streamline of Top Plot

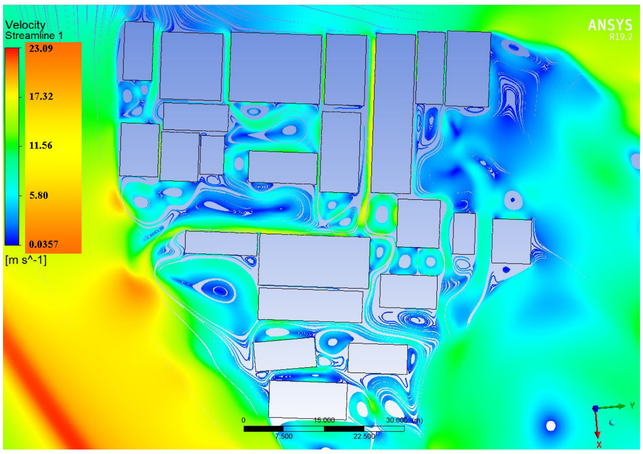

Figure 9: Velocity Streamline of Bottom Plot

From figure 8 and 9, we can observe the wind flow routes in the region and around the buildings. As we can see, the flow is not laminar and slightly turbulent around the building region. The irregular building arrangement and orientation creates blockages of wind thus resulting in stagnant pockets of air which recirculate in that very region continuously until another mass of fluid propels it. This phenomenon is called Stagnation Point development (Circular white regions in the chart). The situation is even worse in the bottom plot due to the previously mentioned blockage from the top plot which has created multiple and bigger stagnation points in the vicinity of individual buildings.

The pathway followed by wind should be as undisturbed as possible which creates high velocity flow and prevents formation of any sort of stagnation points in the region. This can only be achieved through proper urban planning and design of buildings.

3. **Pressure Contour**

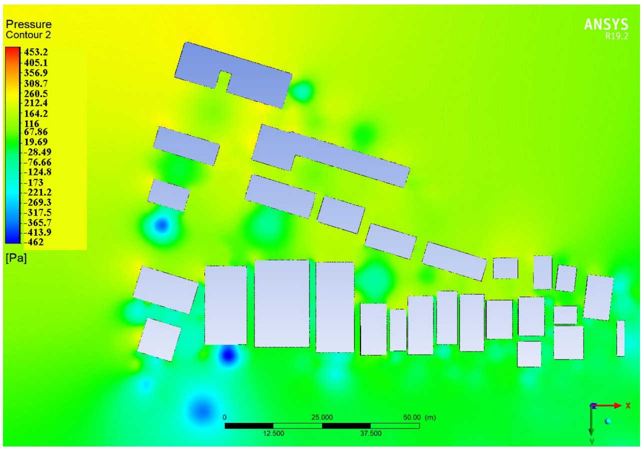

Figure 10: Pressure Contour of Top Plot

As we can see from figure 10 and 11, the wind pressure is quite high in the area in direct contact with the wind but the wind ward areas or shaded regions observe limited air pressure which denotes minimum wind flow in and around that specific area. The less wind flow generally denotes lower thermal comfort for the pedestrians and residents in the area. The

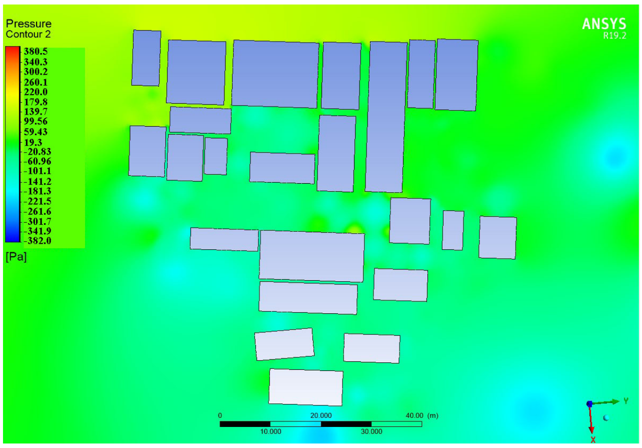

Figure 11: Pressure Contour of Bottom Plot

pressure ranges from 265 Pascal high to as low as -462 Pascal which is not what we desire. The situation in the bottom plot is a bit different to the top plot in terms of pressure variation over the region as it ranges from 265 Pascal high to as low as -18 Pascal but still is undesirable in terms of proper wind circulation in and around the region.

The haphazard land allocation and building construction is certainly negatively impacting the wind pressure variation in the region as shown in figure 10 and 11. Even though this much pressure difference is not detrimental to the thermal comfort of the overall macro-climate, micro-climate analyzing, this might create some stagnation points around the buildings which causes discomfort to the people.

6. Conclusion and Recommendation

The present building arrangements within the site settlements is detrimental to the wind environment of the region. The irregular building orientation and haphazard layout provides obstacle to the wind flow which results in variation of pressure distribution, wind velocity and circulation behavior of the wind throughout the buildings present. The simulation of the site suggests there is an improper distribution of wind throughout the area especially in the bottom plots due to higher pressure difference across different buildings as well as development of higher number of stagnation points. The presence of velocity of around 16m/s in the area denotes that through proper development of wind corridors we can increase the wind speed significantly.

Creation of proper building arrangements along with properly sized street canyons can helps us achieve higher wind circulation which ultimately increases the comfort of the pedestrians and residents living in the area especially during the hot and humid time of the year. Reduction of stagnation points also decreases the stuffy feeling observed by the pedestrians while commuting across the street section which significantly improves the pedestrian flow through the section. The increase in pedestrian interaction with the structures in the street section will improve the economy of the section significantly. The improved wind circulation throughout the settlements will also increase the ventilation inside the buildings thus improving the thermal comfort. The wind load data obtained from this research can also be utilized in further research in similar fields.

Thus, the CFD simulation of the street section joining Majhediya to Mahilwaar village in the Lumbini Sanskritik Municipality shows that in order to improve the situation in the future real estate development projects, it is mandatory that we understand the wind behavior in the region and incorporate development of proper wind corridors development in the design phase. Incorporating certain setback and planning conditions will help improve the wind circulation in the area which can be further researched in the future.

The research could not be completed without multiple limitations faced along the way. The data obtained from Department of Hydrology and Meteorology (DOHM) were found to be inaccurate which prevented any sort of data validation for field measurements. The lack of computation memories meant a limit to reduction of mesh size in CFD which affects the precision of the results. The research was carried out on a small section of the entire road on a single week which does not represent the whole area and its climate over the year. The research fails to incorporate the building surface roughness and design which resulted in modelling of the buildings as homogeneous smooth rectangular objects. The research only considers the buildings as wind breakers and does not consider existing vegetation in the area as other wind breaking factors.

Further research can be carried out by overcoming the limitations faced during this research project in the future. The building layouts must be improved and made as previous as possible for the uninterrupted wind flow throughout the region. The use of setbacks can be implemented to modify the size of wind corridors between the buildings in the future developments. Development of uniform buildings shapes and sizes according to their classification can be useful in maintaining uniform wind pressure in the area. Further research can be carried out incorporating natural wind breakers such as vegetation in CFD to analyze the wind environment within the region. Surface roughness of the buildings and pavements can be considered to study their effect in wind behaviors. Further research can be done under higher computing power to obtain more precise and accurate results which can help understand the condition even better.

References

[1] CJG Morris, Ian Simmonds, and Neil Plummer. Quantification of the influences of wind and cloud on the nocturnal urban heat island of a large city. *Journal of Applied Meteorology and Climatology*, 40(2):169–182, 2001.

[2] Edward Ng. Policies and technical guidelines for urban planning of high-density cities–air ventilation assessment (ava) of hong kong. *Building and environment*, 44(7):1478–1488, 2009.

[3] Eddy Willemsen and Jacob A Wisse. Design for wind comfort in the netherlands: Procedures, criteria and open research issues. *Journal of Wind Engineering and Industrial Aerodynamics*, 95(9-11):1541–1550, 2007.

[4] Miura Kubota and Mochida Tominaga. Wind tunnel tests on the relationship between building density and pedestrian-level wind velocity: development of guidelines for realizing acceptable wind environment in residential neighborhoods. pages 1699–1708, 2008.

[5] ANSYS. Ansys fluent reviews, 2022. [Online; accessed 2022-08-16].

[6] Kangkang Gu, Yunhao Fang, Zhao Qian, Zhen Sun, and Ai Wang. Spatial planning for urban ventilation corridors by urban climatology. *Ecosystem Health and Sustainability*, 6(1):1747946, 2020.

[7] Chao Ren, Ruozi Yang, Chen Cheng, Pei Xing, Xiaoyi Fang, Shuo Zhang, Huifang Wang, Yuan Shi, Xuyi Zhang, Yu Ting Kwok, et al. Creating breathing cities by adopting urban ventilation assessment and wind corridor plan–the implementation in chinese cities. *Journal of Wind Engineering and Industrial Aerodynamics*, 182:170–188, 2018.

[8] Nastaran Shishegar. Street design and urban microclimate: analyzing the effects of street geometryand orientation on airflow and solar access in urban canyons. *Journal of clean energy technologies*, 1(1), 2013.

[9] Erik Johansson. Influence of urban geometry on outdoor thermal comfort in a hot dry climate: A study in fez, morocco. *Building and environment*, 41(10):1326–1338, 2006.

[10] Khaled A Al-Sallal and Laila Al-Rais. Outdoor airflow analysis and potential for passive cooling in the traditional urban context of dubai. *Renewable energy*, 36(9):2494–2501, 2011.

[11] Sumei Liu, Wuxuan Pan, Hao Zhang, Xionglei Cheng, Zhengwei Long, and Qingyan Chen. Cfd simulations of wind distribution in an urban community with a full-scale geometrical model. *Building and Environment*, 117:11–23, 2017.

[12] ANSYS. Inviscid flow, 2009. [Online; accessed 2022-08-16].

[13] ChampionTraveler. The best time to visit lumbini, nepal for weather, safety, & tourism. [Online; accessed 2022-08-16].

[14] ANSYS. Nsys fluent 12.0 user's guide - 29.1.2 displaying contours and profiles. wwwafs. [Online; accessed 2022-08-16].