TRIBHUVAN UNIVERSITY
INSTITUTE OF ENGINEERING
PULCHOWK CAMPUS

Thesis No: 076/MSEEB/006

Enhancement of Pedestrian Comfort with the help of Wind Corridors through CFD Simulation

by

Biplav Pokhrel

A THESIS
SUBMITTED TO THE DEPARTMENT OF ARCHITECTURE
IN PARTIAL FULFILLMENT OF THE REQUIREMENTS FOR THE DEGREE OF MASTERS IN ENERGY EFFICIENT BUILDING (MSEEB)

DEPARTMENT OF ARCHITECTURE
LALITPUR, NEPAL

October, 2022

COPYRIGHT

DECLARATION

I hereby declare that the thesis entitled "**Enhancement of Pedestrian Comfort with the help of Wind Corridors through CFD Simulation**" which is submitted to the Department of Architecture, Pulchowk Campus, Institute of Engineering, Tribhuvan University. in partial fulfillment of the requirements for the degree of Masters in Energy Efficient Building (MSEEB) is a research work carried out by me, under the supervision of Dr. Sanjaya Uprety and Er. Niranjan Bastakoti. I declare that the work is my own and has not been submitted for a degree of another University.

………………………………………………..

Biplav Pokhrel

076MSEEB006

TRIBHUVAN UNIVERSITY
INSTITUTE OF ENGINEERING

PULCHOWK CAMPUS

Department of Architecture

The undersigned certify that they have read, and recommended to the Institute of Engineering for acceptance, a thesis entitled **" Enhancement of Pedestrian Comfort with the help of Wind Corridors through CFD Simulation"** submitted by **BIPLAV POKHREL** (076MSEEB006) in partial fulfillment of the requirements for the degree of Masters in Energy Efficient Building.

………………………………..

Supervisor, Dr Sanjaya Uprety
Associate Professor
Department of Architecture

………………………………………

Co-Supervisor, Niranjan Bastakoti
Assistant Professor
Department of Mechanical Engineering
Kathmandu University

.………………………………………

External Examiner, Saurav Shrestha
Senior Architect

…….…………………………………………

Committee Chairperson, Dr Sanjaya Uprety
Associate Professor,
Department of Architecture
Date: 10/13/2022

Abstract

The exponential growth of earth's temperature alongside the use of materials having high heat capacity, to cater the housing demands, brings about the need for significant cooling mechanisms to improve thermal comfort. Researchers have verified that the use of wind as a cooling agent is the most cost effective and efficient way to reduce the surface air temperature in the long run.

This research is focused upon the development of wind corridor to increase the pedestrian thermal comfort for the hot and humid region of Lumbini Sanksritik Municipality by using CFD simulation (ANSYS Fluent). The research includes the simulation of base case scenario with the proposal of multiple solutions for the region to significantly improve the thermal comfort. ANSYS Fluent is used to simulate the plots modelled in SOLIDWORKS where the outputs primarily are obtained in the form of velocity vectors, pressure & velocity contours and velocity streamlines.

The research shows that it is possible to have better air flow velocity and circulation than the base case by just following the simple building bylaws during the design process. This increase in flow velocity and improvement of wind circulation has been proven to increase the pedestrian thermal comfort of the region.

Keywords: Wind Corridors, CFD, Thermal Comfort, Urban Heat Island

Acknowledgement

I would like to begin by expressing my heartfelt gratitude to my thesis supervisor Dr. Sanjaya Uprety, without whom this project would not be completed with the required standards as expected from a graduate student of Department of Architecture, Pulchowk Campus. Without the continuous encouragement and guidance of Dr Uprety, I would not be able to visualize the concept and the framework for this report.

I am also greatly indebted to Er. Niranjan Bastakoti, who as my co-supervisor, has provided me with all the knowledge and foundation, even during his hectic schedule, regarding the use of CFD and understanding the behavior of fluid motion which is paramount in the completion of my thesis. His unwavering criticisms and comments are equally important in the complete realization and development of this report.

I would like to pay my special regards to Er. Hari Bahadur Dura for facilitating me during my troubled times of using CFD analysis in 2D simulation. His suggestion and advise in analyzing the research gap along with understanding the results of ANSYS has helped me significantly in understanding the concept of CFD and completing the report on time.

I would also like to recognize the invaluable assistance of Ms. Apekshya Ghimire in the architectural aspect of my thesis report without whom I would not be able to understand the building bylaws and concept of basic architecture within my topic. Her consistent encouragement and feedbacks have greatly helped me refine the contents of my report.

I would like to express my thanks to Er Ananta Acharya, RA at Kathmandu University, for helping me during the difficult times of using ANSYS Fluent while balancing his own thesis works at that time.

Finally, I would like to pay my greatest respect and thanks to my parents and family, without whom I would not be able to keep on working hard regularly and tirelessly till the completion of my report. Their constant love and encouragement were second to none in revitalizing my tired neurons in order to keep on thinking and producing results.

Table of Contents

Table Of Figures

List of Tables

List of Charts

List of acronyms and abbreviations

CFD – Computational Fluid Dynamics

UHI – Urban Heat Island

ISO - International Organization for Standardization

PMV - Predicted Mean Vote

PPD - Predicted Percentage of Dissatisfied

FAR – Floor Area Ratio

WCI – Wind Chill Index

2D/3D – Two dimension/ Three dimension

ENIAC - Electronic Numerical Integrator and Computer

HPC – High Performance Computing

GIS - Geographic information system

LCP – Least Cost Path

LCZ – Local Climate Zone

LIDAR - Light Detection and Ranging

VCP – Ventilation Corridor Path

GPR – Ground Penetrating Radar

TOD – Transit Oriented Development

DOHM - Department of Hydrology and Meteorology

GUI – Graphical User Interface

MPH/KPH – Miles per hour/ Kilometres per hour

Chapter 1: Introduction

1.1 Background

In the world, more people live in urban regions than in rural ones; in 2014, about 54% of people lived in metropolitan areas. In 1950, 30% of the world's population lived in urban regions, and according to the projected growth rates, by 2050, about 70% of people will live in urban areas (Nations, 2014). Continuous urbanization as a result of population growth and socioeconomic advancement is a major contributor to climatic and environmental change worldwide. Meanwhile, the urban heat island (UHI) effect, which occurs when urban areas are frequently warmer than their rural surroundings, especially during warm seasons, is a result of the urbanization process. British meteorologist Luke Howard originally identified the UHI phenomenon in the early 19th century (TR, 1982). The choice of construction materials, the intricate geometry of the urban environment, the loss of natural cover, and human heat emissions are some of the main causes of this problem (PE, et al., 2015). Urban residents are more at risk for heat-related diseases and mortality because of the heightened temperature caused by UHI, particularly during intense heat waves (Yang et.al., 2019).

The causes of the variables contributing to heat islands may be roughly divided into two groups. The meteorological elements, which include wind direction and speed, humidity, and cloud cover, fall under the first group. The second category essentially reflects the outcomes of city planning, including built-up area density, aspect ratio, sky view factor (SVF), and building materials. Shashua-Bar et al. (2004) looked at how urban layout affected the local climate. According to their findings, locations with shallow open spaces and broader spacing had temperatures that were 4.7 °C higher than the baseline readings obtained from a meteorological reference.

People tolerate greater relative humidity levels better when there is much more ventilation, according to comfort assessments. The acceptability of greater temperatures does not, however, always rise with an increase in airflow (Ahmed, 2003). In their 2010 investigation of street design and its effects on urban microclimate in semi-arid environment, Bourbia and Boucheriba discovered that temperature decreased with increasing aspect ratio. They recommended using the SVF in the design of urban geometry since it is essential for reducing the impact of urban heat islands. According to Golany (1996), who conducted research on the connection between thermal performance and urban morphology and their effects on climatic responses, a city's layout can help with wind circulation and have an impact on wind velocity, which in turn impacts temperature changes. In accordance with its design, form, and the direction of its highways, a city's morphology directly influences the wind's passage inside it. In defining urban thermal situations, wind is important (Morris et al. 2001). It increases heat and air circulation while lowering perceived temperatures to a level that is comfortable for people. High-density development has an impact on pedestrian comfort and safety as well as the ventilation of buildings in metropolitan settings. Lower permeability for urban air ventilation at the pedestrian level is caused by big podium constructions, consistent building heights, and tall, bulky high-rise building blocks with little room between them (Ng, 2009).

As air enters the interiors of urbanized areas through roads, open spaces, and tunnels, these pathways between buildings are known as wind corridors. The main elements influencing wind at ground level include building configuration (such as height, width, arrangement, and density) (Willemsen & Wisse, 2007). In Asian nations like Japan (Chen et al. 2009) and South Korea, projects to build urban ventilation corridors to increase wind flow across cities are now being developed (Cha et al. 2007). Planners and designers who want to enhance the wind environment must take building layout into account. As a result, it's critical to comprehend and account for the pedestrian wind environment surrounding buildings while designing cities (Kubota et al., 2008).

It was discovered that in the summer, a 1 m/s increase in wind speed results in a 0.14°C decrease in heat island intensity. In metropolitan locations, wind speed has a significant role in air quality, health, comfort indoors and out, and building energy use (Memon & Leung, 2010). In tropical areas like Singapore, a wind speed of 1.1 to 1.5 m/s produces a cooling impact that results in a 2 °C reduction in temperature (Erell et al., 2011).

In order to fight the UHI effect, this study focuses on applying Computational Fluid Dynamics (CFD) studies to build wind corridors. The movement of fluids under various situations that are regulated by physical principles is a topic of computational fluid dynamics. Computational fluid dynamics, or CFD, is the most widely used technique for deriving and quantitatively assessing the rules of control of fluid dynamics. CFD splits the geometric domain into tiny volumes and simplifies the process of solving difficult partial differential equations on a grid. CFD aids analysts in simulating and comprehending fluid flow theory. Instruments are not necessary to measure a variety of flow characteristics. It is possible to measure several flow factors (Kalkan & Dagtekin, 2015).

1.2 Need of Research

70% of the world's population is projected to reside in cities by 2050, with over half of them in tropical regions (United Nations 2017). Thermal discomfort, which is exacerbated by elevated temperatures and humidity as well as the urban heat island (UHI) phenomenon, is one of the biggest hazards to a large number of tropical urban residents (Wong & Chen, 2008). Due to climate change, more increases in average and severe temperatures are anticipated.

Even with more dire predictions, urban planners and architects should still take into account passive cooling methods to minimize the need for air conditioners and break the current cycle of increased UHI effects, higher temperatures, increased greenhouse gas emissions, and more severe climate change impacts. Fortunately, a number of techniques have been shown to minimize the impacts of UHI, cool physical surfaces, and enhance thermal comfort. Among these, natural ventilation has historically been the most popular cooling technique in the tropics and is still regarded as having a high level of efficiency in some situations while using zero energy. Large-scale building projects and the expansion of urban areas, however, have a significant influence on wind patterns, preventing cooler air from entering the city and stalling warmer air in urban canyons. In high-density metropolitan regions, buildings and breezeways should be built with care to maximize natural ventilation (Tabada & He, 2018).

Studies have demonstrated that the urban wind environment and the urban shape are strongly connected, and that increasing urban ventilation can lessen the impact of the urban heat island and lower the demand for energy in cities along with increasing pedestrian comfort in the region (Yang, 2021). As a result, the creation of urban ventilation corridors is becoming an increasingly crucial component of urban design.

1.3 Importance of Research

Urban settlement growth has been the primary factor behind the current rise in Urban Heat Island (UHI) impact. Insufficient ventilation and inadequate wind movement inside the settlement have contributed to an increase in thermal discomfort as a result of the rise in concrete structures with constrained space. In hot and humid settings, this issue can be seen more clearly. The growth in thermal discomfort among building occupants and pedestrians is a result of rising temperatures and humidity without adequate sources of evaporation. So, this research aims to determine the possibility of improving the pedestrian comfort effect by the means of properly designed wind corridors through CFD simulation through the reduction of street temperature. This research will help the planning committee and designers to design the placement of urban streets and passages in such a way that it maximizes wind flow to reduce the street temperature within the vicinity. This research can also play a major part in developing proper residential sub-urbs by government through land pooling process which can develop sustainable and energy efficient settlements in the near future.

1.4 Problem Statement

The urban heat island (UHI) effect, a phenomenon where metropolitan areas are frequently warmer than their surrounding rural environment, especially during warm seasons, is a result of the urbanization process (Zhang, 2017). Thermal discomfort, brought on by the area's relatively high temperatures and humidity levels as well as the aggravation of warmer circumstances due to the urban heat island (UHI) phenomenon, is one of the biggest risks. The urban heat island effect, air pollution, and increased urban energy consumption have become more and more significant limiting factors impacting the healthy growth of cities as a result of the expansion of city scale and uneven distribution of urban structures (Yang, 2021). Studies have demonstrated that the urban wind environment and the urban shape are strongly connected, and that increasing urban ventilation can lessen the impact of the urban heat island and lower the demand for energy in cities (Tabada & He, 2018). As a result, the creation of urban ventilation corridors is becoming an increasingly crucial component of urban design. In some climatic locations, the combination of a steady rise in surface temperature and excessive humidity tends to make people's thermal comfort levels poorer. Therefore, all governing authorities and designers should prioritize using the cooling capability of the wind by creating suitable wind corridors.

1.5 Objectives

The major objective of this research is to determine if developing wind corridors can help enhance pedestrian thermal comfort by reduction of street temperature.

Specific Objectives:

a. To understand factors affecting the development of wind corridors in an area.
b. To use CFD method to simulate wind flow in and around the wind corridor.
c. To observe the effect of introducing wind corridors in the air movement of the region.
d. To propose building layouts as solutions to increase air velocity and circulation for future developments.

1.6 Thesis Outline

The general outline of this thesis is as follows:

The first chapter is based upon the general introduction and the background to the concept of the thesis envisioned. It includes the need and importance of the research in question after determination of research gaps in the current world of academia in the field of outdoor thermal comfort and wind environment.

The second chapter introduces all the literatures referred along with areas of research focused to further gain knowledge on the topic. Different research articles and publications have been referred to glean and compile all the necessary knowledge in order to begin the research in an efficient and accurate way. Areas of research pursued by leading scientists and methods they used to obtain the results are also analyzed and discussed for the ease of understanding the general concept of the thesis project to be undertaken.

Moving on, the third chapter is totally focused on explaining the methodological philosophy undertaken to pursue the correct methods for the research. Methodological reviews of past research publications help a lot in providing information and knowledge on the best way to approach the research in order to obtain precise and accurate results.

The fourth chapter of this thesis introduces the site selected to carry out the experimental research for validation of hypothesis. It includes the exact location of study along with any additional information to narrow down the exact point of study with any necessary exceptions mentioned beforehand.

The fifth chapter of this research explains the details of experimental research carried out with its advantages and limitations. It also explains the reasons for opting this form of quantitative research over qualitative research for the formal completion of this project.

The second last, sixth chapter, discussed the results of the simulation carried out along with their inference from the researcher. This chapter also tends to explain the novelty of the results and its practicability. This chapter particularly explains the need of this thesis and the reasons why it was perceived as necessary by the researcher to be carried out.

Finally, chapter seven includes the references of the articles and literatures referred for the knowledge and information in order to successfully complete this thesis report. Also supplemented is the annex section with additional documents necessary for the completion of this degree.

Chapter 2: Literature Review

2.1 Urban Heat Island Effect

Modern medicine and the health system have continuously allowed the human population to grow and sustain itself across the world, unlike in the distant past when it was still in its rudimentary stage and caused repeated global pandemic outbreaks that led to population curtailment. This has helped to fuel rising calls for settlement and housing quotas. The increase in residential demand has continuously resulted in the loss of green space and the creation of concrete jungles with all the contemporary conveniences people need and want. Numerous issues have been unintentionally caused by increased urbanization and the speed at which it is occurring. More specifically, it has led to a rise in air pollution and urban heat island (UHI), which endangers the health of those living in cities (Wang, et al., 2020). The earliest known UHI was identified in Luke Howard's research of the climate in London in 1818. In contrast to the country, he discovered that the city possessed artificial heat. Emilien Renou made comparable findings about Paris in the latter part of the 19th century. This phenomenon is considered to be detrimental to thermal comfort (Gartland, 2012). Figure 2.1 shows that the temperature of the air around metropolitan areas is at least 2°C greater than that of areas with fewer buildings or more green spaces. The trees and open spaces tend to create an environment where the temperature is less than the city.

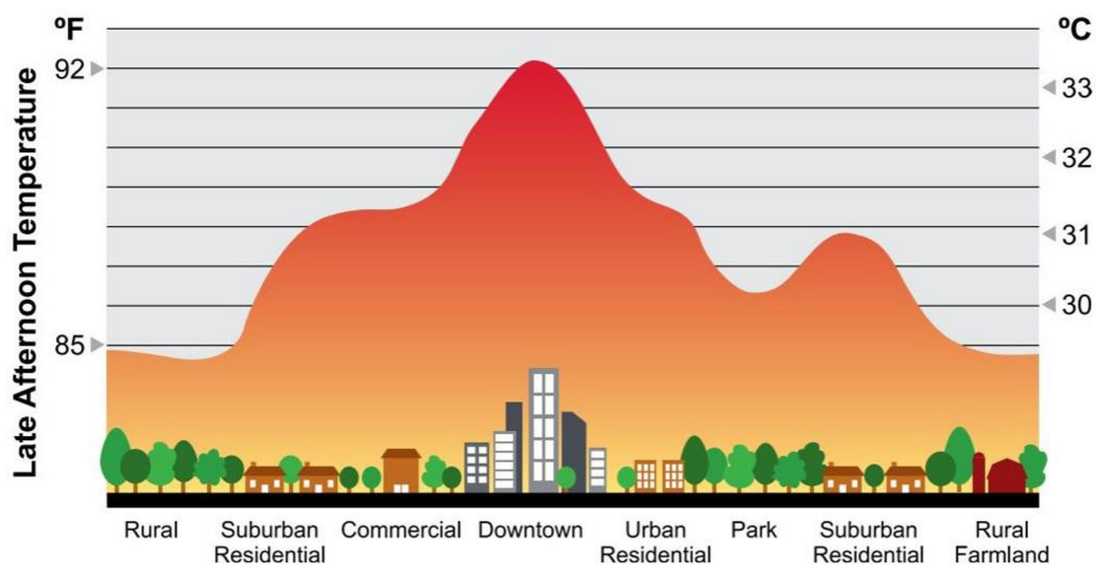


Figure 2- 1 Urban Heat Island Effect (greeninfrastructureontario.com)

By 2030, it is anticipated that metropolitan regions will house more than 61% of the world's population (Economic and Division, 2012). Cities have become densely inhabited urban environments with less vegetation and more impermeable surfaces as a result of rapid urbanization. When compared to their rural surroundings, urban areas have greater air and surface temperatures because of the loss of vegetation, which increases the amount of heat stored in the ground layer and building materials (Oke, 1982). Urban Heat Island, often referred to as Reverse Oasis, is a phenomenon that typically occurs where the air and surface are hotter than the surrounding rural areas. When urban development reaches a certain level, air flows upward in the city, sinks downward in the suburbs, and fills the urban area with low pressure as a result of changes in the urban nature of the pad surface, atmospheric pollution, and artificial waste heat emissions. This phenomenon is known as the urban heat island circulation (Su, et.al.,2016). The law of energy conservation is easily followed by the Urban Heat Island Effect process. Low-albedo materials serve as thermal storage units (batteries) that capture the sun's heat energy during the day and release it gradually at night, raising the temperature in the area around these batteries. The temperature of the environment in an urban region might rise by 0.5 to 1.5°C due to man-made land created by construction operations (Ibrahim, et al., 2018). While it may be seen as a benefit to high latitude cities in cold climates of the world due to its role in reducing the energy needed for heating purposes, in tropical regions it is a major cause of people's discomfort from the heat and health problems associated with it, which ultimately leads to increased use of energy through active cooling strategies (Rajagopalan, et. al., 2014).

Che-Ani et al. (2009) divided the factors that contribute to and define the intensity of heat islands into two categories: meteorological and urban design parameters. Meteorological factors include air temperature, wind speed and direction, humidity level, and cloud cover. Urban design parameters include density of urban areas, percentage of built-up areas, aspect ratio of urban canyons, sky view factor, building construction materials, and urban form. Additionally, the urban environment, human population increase, urban activities, and local geography can all be considered contributing factors to the UHI impact (Su, et.al., 2016). Urban activities, which directly contribute to the energy absorption by the urban built up regions, are the regular operations of the urban sectors, such as transportation, industry, and other service-oriented activities. The energy that manufacturers release in the form of smoke and heat from their machinery eventually combines with the air that travels through metropolitan areas and interacts with the environment.

The smoke from the cars we often use to carry people and products is responsible for the same process. Waste energy is a vague term that describes this type of energy. The concrete buildings and other low-albedo materials absorb this energy in addition to the solar radiation, which worsens the situation for the local population. Compared to less-paved surfaces in rural settings, paved surfaces in urban and suburban areas are frequently warmer. Typically, materials for pavement have relatively low albedo or reflectivity. The percentage of incoming radiation that is reflected off a surface is known as albedo. It plays a significant role in the energy balance on the surface of the earth since it defines how much solar radiation is absorbed.

The majority of radiation that comes in contact with surfaces is typically absorbed by those surfaces. These raise the temperature of the surrounding ambient air by heating the pavement material, which then radiates the heat. Only around 18°C of heat is absorbed when vegetation-covered surfaces with wet soil beneath are exposed to direct sunshine.

However, dark, dry surfaces exposed to the same circumstances may absorb heat of up to 52°C (Ibrahim et al., 2018). Figure 2 shows that in addition to polluting the air in the vicinity, the smoke from industries and daily-use automobiles also heats the air around them. This radiation of heat into the atmosphere with the help of smoke is also a major contributor in UHI effect.

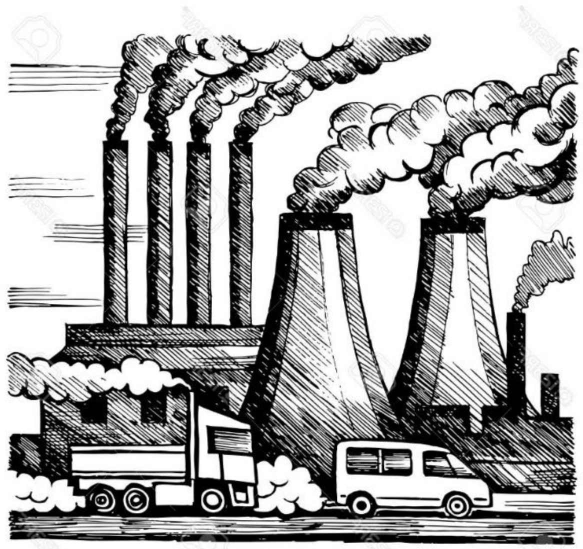


Figure 2- 2 Waste heat and pollution (getdrawings.com)

The UHI effect causes a steady rise in daytime and nighttime temperatures, which over time causes thermal discomfort for those who live nearby. As a result, individuals spend more money on active cooling systems, which consume more energy and pollute the environment by releasing harmful Green House Gases into the atmosphere. As a result, the quality of the local air has declined. Air pollution can also have an impact on human health, resulting in respiratory problems, heat cramps, non-fatal heat stroke, and overall discomfort. The most vulnerable groups, including elderly persons and children, are those who most susceptible to these adverse health outcomes (Ibrahim, et.al. 2018). The cooling load of urban buildings may double in Athens, where the mean heat island intensity exceeds 10 °C, and the peak electricity load for cooling purposes may triple, especially for higher set point temperatures, while the minimum COP value of air conditioners may decrease by up to 25% as a result of the higher ambient temperatures (Rajagopalan, et.al., 2014). UHIs alter several urban ecological processes, including material metabolism and the energy cycle, in addition to the distribution and behavior of urban life forms, the urban thermal environment, regional temperature, urban hydrology, air quality, and physical and chemical qualities of urban soil. Several ecological and environmental issues have resulted from this (Xie, et.al., 2020).

Researchers have conducted a number of studies to counteract and mitigate the effect owing to the growth in several large-scale and small-scale issues caused by the persistent UHI effect. Numerous research investigating the usage of green surfaces, water bodies, and high albedo materials have been conducted, and it has been discovered that they may significantly reduce it. However, they are frequently either impractical or too expensive. In the case of trees, vegetation offers shade, improves air quality, controls the water cycle, and increases a city's overall resilience. Among the most often utilized urban greening strategies are green roofs and facades, green/cool pavements, tree-lined streets, urban farming, and green ventilation corridors. Therefore, it has been determined that using natural wind is the most efficient and practical way to mitigate the UHI impact. In the tropics, natural ventilation has traditionally been the most popular cooling technique. Under some circumstances, it is still believed to be very effective and energy-free. Large-scale building projects and the expansion of urban areas, however, have a significant influence on wind patterns, preventing cooler air from entering the city and stalling warmer air in urban canyons. In high-density metropolitan regions, buildings and breezeways should be built with care to maximize natural ventilation (Tabada & He, 2018).

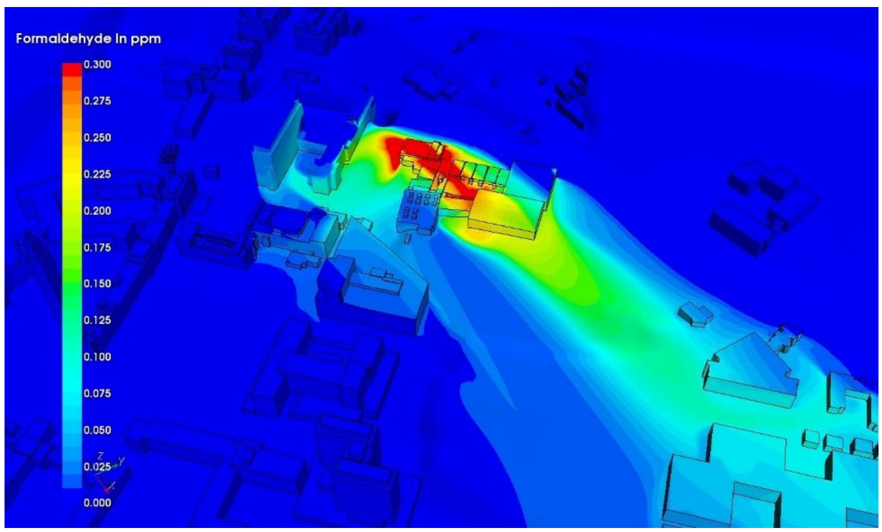


Figure 2- 3 Pollutants dispersion in CFD (IES)

Increased wind speed can undoubtedly lower the local temperature, lessen the impact of UHI, and clear contaminants from the region. As a consequence, the thermal comfort of the local residents and pedestrians is improved. In order to prevent local heat accumulation, only wind that is powerful enough may successfully advect heat away from its origination point. Three-dimensional turbulence frequently accompanies simple advection and acts as an advection catalyst (Hsieh & Huang, 2016). For the purpose of understanding and implementing air exchange, which would mitigate UHI and lower air pollution, studies on wind corridor during weak or quiet wind conditions are required. Additionally, establishing ventilation corridors has the benefit of preserving land usage for city improvements to the climate and ecology (Wang, et al., 2020).

Figure 2.3 shows the dispersion of pollutants from a group of buildings because of movement of air. The high velocity of wind tends to carry the pollutants along with any excess humidity present in the area away from the buildings in the direction of wind. This simulation output also shows us that creating a passage for wind helps to reduce pollution and increase thermal comfort in the area which ultimately reduces the UHI effect in the area.

2.2 Pedestrian Comfort

Since it is more of an abstract notion with a subjective character, human comfort cannot be measured in plain words or with alphanumeric numbers. A value that is continually influenced by an individual's perspective and personality cannot be regarded as a universal value by science. We cannot define comfort in terms of particular values because it is a psychological idea of human life, but we can, unexpectedly, understand its nature. Since it is complex, including several elements that have an impact on pedestrians, and cannot be well characterized by a single factor like ambient air temperature, the climate of the outside environment is even more challenging to define. According to Jeffrey Cook, it is impossible to compare outdoor with interior comfort standards. The outdoor criteria should be benchmarked by different parameters, and the benchmark of the individual’s comfort level outdoors is much higher than the one for indoor environment (Cook, 2001).

The ISO 7730 defines comfort as "The state of consciousness expressing contentment with the thermal environment." Thermal comfort may be described more qualitatively as the range of climatic circumstances in which most individuals feel comfortable, even if it cannot be fully measured in terms of values. The projected mean vote (PMV) index, created by Fanger and Toftum, is the most used measure of thermal comfort (2002). Fanger (1970) stated that in order to preserve thermal comfort, "two requirements must be met. The first gives the impression of thermal neutrality and is the actual sum of skin temperature and body core temperature. The second is realizing how the body maintains its energy balance, which is based on energy conservation.

Energy Balance equation:

$$M + W + R + C + E_D + E_{RD} + E_{SW} + S = 0$$

Where M is the metabolic rate, W is the physical work output, R is the net radiation of the body, C is the convective heat flow, E_D is the latent flow to evaporate water diffusing through the skin, E_{RD} is the sum of heat flows for heating and humidifying the inhales air, E_{SW} is the heat flow due to evaporation of sweat, S is the storage heat flow for heating or cooling the body mass.

The energy balance equation, which is a function of six factors that affect human thermal comfort, is how Fanger arrived at the PMV function. In actuality, PMV is a function of two human factors, human activity, air temperature, air relative humidity, air velocity, and mean radiant temperature in addition to two environmental variables. The PMV null value denotes a neutral sense, whereas the PMV index values vary from -3 to +3, which correlate to the occupant's feelings of cold to hot. Based on the constant state of heat transmission between the body and the environment, the PMV determines the thermal stress. It is an empirical equation used to assess the average response on a scale of how comfortable it is to be in the heat of a big group of people. People were exposed to various surroundings over varying time intervals in order to create a curve and obtain average findings. PPD stands for the anticipated proportion of unsatisfied customers at each PMV (Grifoni, Passerini, & M., 2013).

PMV	PPD	People predicted to vote (%)		
		0	-1, 0 or +1	-2,-1,0,+1 or +2
+2	75	5	25	70
+1	25	30	75	95
+0.5	10	55	90	98
0	5	60	95	100
-0.5	10	55	90	98
-1	25	30	75	95
-2	75	5	25	70

Figure 2- 4 PMV and PPD Values (Fanger, n.d)

2.2.1 Outdoor Thermal Comfort

Thermal comfort has long been a topic of scientific study, but the bulk of attention has been paid to interior thermal comfort, which is concerned with the circumstances inside a building that have an impact on how comfortable its occupants feel. The temperature, air quality, humidity level, etc. are the main factors here. Thermal comfort outside of a building refers to maintaining the same level of comfort but in a larger, more significant area. The scientific community should place the outside environment on an equal footing with the indoor environment since it is the owner of all the factors that influence indoor thermal comfort.

Relative humidity, wind speed, and air and mean radiation temperatures are climatic factors that influence thermal comfort. The mean radiation temperature, which is mostly produced from solar radiation, is known as the most important climatic parameter impacting thermal comfort in outdoor contexts. Various studies have confirmed and backed the same idea (Taleghani, et.al, 2018; Mahmoud, 2011; Li Lixiu, 2017; Tsitoura, et.al.,2014; Yoshida, et.al.,2015; Jin et.al. ,2017; Chen, et.al., 2015).

2.2.2 Street temperature

Rapid urbanization and home construction have greatly encouraged the use of building materials with low albedo and high heat storage capacity. In compared to regions covered by vegetation or natural soil, the pavement's temperature rises significantly as a result of its enhanced capacity to store heat. The air temperature near the surface is also significantly increased by the warmer pavement surfaces (streets), and this warmer air is subsequently distributed throughout the area through convection. In the long term, this raises the average temperature on the microclimatic scale. The Urban Heat Island Impact is a result of this combined effect (UHI). Increased urban temperatures have a substantial negative influence on people's quality of life by escalating energy use, degrading comfort, endangering health, and harming the city's economy. The thermal comfort and health of people are substantially impacted by high air temperatures, especially those who live in metropolitan settings (Grifoni, Passerini, & M., 2013).

The heat output from the pavement to the near-surface air could be affected by the change of pavement surface albedo and thermal conductivity. The heat output intensity from the pavement H_{out} to the near-surface air includes the reflected short-wave solar radiation Hr, the emitted pavement surface long-wave radiation He, and the convective heat H_c.

$$H_{out} = H_r + H_e + H_c$$
$$H_r = \tilde{\alpha} Q_s$$
$$H_e = \varepsilon\sigma(T_s^4 - T_{sky}^4)$$
$$H_c = h_c(T_s - T_a)$$
$$T_{sky} = T_a(0.754 + 0.0044T_{dp})^{0.25}$$

Equation above demonstrates that in addition to pavement surface temperature, additional parameters such as pavement surface albedo and emissivity, Stefan-Boltzmann constant, convection coefficient, sky temperature, and air temperature also affect how much heat is produced. As a result, heat output is a more thorough metric to assess the impact of UHI. As can be observed, pavement with higher surface albedo has a higher heat output intensity throughout the day. After sunset, however, the pavement with the higher surface albedo produces less heat production.

As a result, the heat output intensity Hout is equal to the total of He and Hc, which drop a s surface albedo increases at night. The amount of solar energy that is reflected increases significantly as surface albedo increases. Additionally, Hr is growing faster than He and Hc are losing momentum. As a result, throughout the day, the surface albedo increases al ong with the heat output intensity Hout. Based on the aforementioned study, it was discov ered that raising pavement surface albedo might lower pavement surface temperature but also raise daily total heat output intensity Hout and reflected solar radiation Hr.

As a result, the location of Hr greatly influences how albedo affects UHI. Hr may generally reflect off of nearby structures or back into space. Given that it departed from the planet, the amount of HR reflected back into space has relatively little impact on the UHI effect (Chen, Wang, & Zhou, 2016).

Bottema (n.d), defines pedestrian discomfort as: ‘‘Pedestrian discomfort occurs when wind effects become so strong and occur so frequently (say on time scales up to 1 h), that people experiencing those wind effects will start to feel annoyed, and eventually will act in order to avoid these effects.’’ This definition states that an acceptable wind comfort criteria may include a threshold for discomfort and a likelihood that the threshold will be exceeded. The lowest wind speed and turbulence level for unpleasant circumstances is known as a discomfort threshold. Discomfort thresholds are generally of the form:

Ue ¼ U þ k u > UTHR ð1Þ

where Ue is the equivalent wind speed, U is the mean wind speed, k is the peak factor, u is the standard deviation of the wind speed (turbulence) and UTHR is the threshold value (all at pedestrian height) (Blocken & Carmeliet, 2003).

The mechanical force of the wind, the body's thermal comfort, and the impact of wind chill on exposed skin make up the three main aspects of outdoor pedestrian comfort. To preserve comfort, all three requirements must be met at once; otherwise, the pedestrian would experience discomfort. The cooling impact on exposed skin is measured by the wind chill, which combines wind speed and ambient temperature. In contrast to body heat balance, wind chill is typically unaffected by degree of clothing in urban settings since the face, ears, and frequently the hands are exposed even when pedestrians are wearing thick winter clothes. When evaluating pedestrian comfort, areas prone to seasonal temperatures less than 10°C should take wind chill into account. The wind chill index (WCI) provides an empirical estimate of wind chill [15]:

$$\mathrm{WCI} = (10.45 + 10V^{1/2} - V)(33 - T_a)\ \mathrm{kcal/(m^2\ h)},$$

where V » is the wind speed in m/s (1.78 m/s) and the ambient temperature in °C (10°C).

Another wind chill scale is the equivalent wind chill temperature, the ambient temperature in still air required to give the same

$$T_{eq} = -0.04544(\mathrm{WCI}) + 33^{\circ}\mathrm{C},$$

T_{eq} is the equivalent temperature in °C.

(Soligo, Irwin, Williams, & Schuyler, 1998)

Table 2- 1 Beufort Scale of wind speed and effect

Beaufort Number	Description	Wind Speed at 1.75 m height (m/s)	Effect
0	Calm	0.0–0.1	
1	Light air	0.2–1.0	No noticeable wind
2	Light breeze	1.1–2.3	Wind felt on face
3	Gentle breeze	2.4–3.8	Hair disturbed, clothing flaps, newspaper difficult to read
4	Moderate breeze	3.9–5.5	Raises dust and loose paper, hair disarranged
5	Fresh breeze	5.6–7.5	Force of wind felt on body, danger of stumbling when entering a windy zone
6	Strong breeze	7.6–9.7	Umbrellas used with difficulty, hair blown straight, difficult to walk steadily, sideways wind force about equal to forwards walking force, wind noise on ears unpleasant
7	Near gale	9.8–12.0	Inconvenience felt when walking
8	Gale	12.1–14.5	Generally impedes progress, great difficulty with balance in gusts
9	Strong gale	14.6–17.1	People blown over

Table 2-1 shows the effect that winds blowing at different speeds have on the environment and the numeric ranking provided by Beufort. The wind speed of around 6m/s at Lumbini

Sanskritik Municipality (Site) can be classified as a fresh breeze. This velocity of wind isn't dangerous enough to deteriorate the pedestrian comfort but is high enough to maintain proper air circulation around the outdoor region with proper movement of pollutants and humidity of the air thus increasing the thermal comfort.

2.3 Design of Wind Corridors

Human development along the years has created a rise in built up environment as a replacement of green spaces in the world. The constant decrease of green spaces and increase in concrete monoliths has led to the rise in the global temperature: cumulative of individual rise in urban temperatures. This rise in temperature thus results in massive discomfort to the living organisms residing in and around the area. Constant rise in temperature and pollution as a result of rampant urbanization also leads to health-related problems in the long run. In order to combat these problems, researchers have unequivocally concluded that the use of natural elements such as wind to be paramount. The use of wind in cooling and ventilating a building has been traditionally opted by our forefathers from the Roman age where underground air canals were built into the residences of Nobles. Another efficient way of utilizing wind in cooling the area is through the development of Wind Corridors. During the Romans Era, they constructed their buildings in such a way that they are well ventilated using only passive strategies as there no options of using energy for ventilation during those times. They created passageways and conduits where the air flowed all around the buildings circulating cool winds to every room of the Royals as shown in figure 2.4 in The Pantheon. This was the beginning of the concept of ventilation in modern buildings.

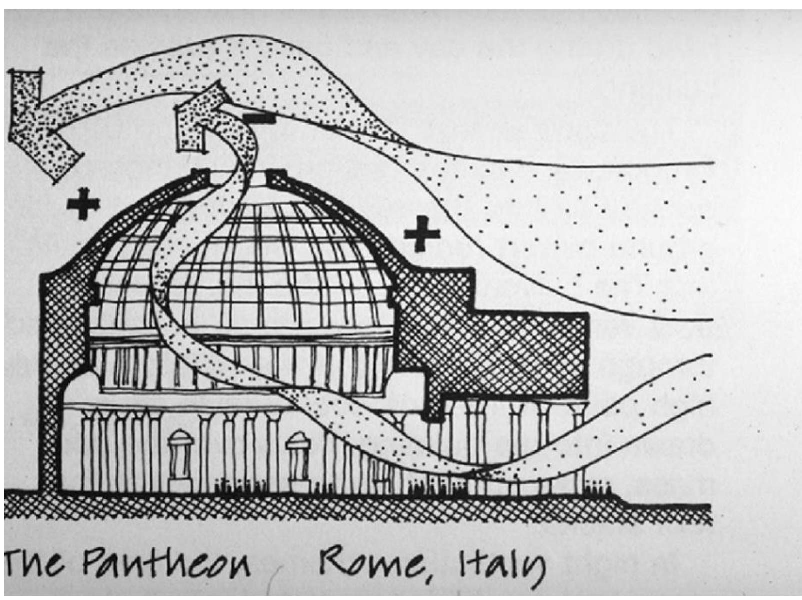


Figure 2- 5 Use of air in cooling in ancient Rome (Columbia.edu)

As it increases air flow capacity and lowers summer building energy use, optimal urban ventilation is crucial for reducing the consequences of UHI. Pollutant dilution is assisted by ventilation as well. Because wind flows carry air contaminants and turbulence mixes them, the higher the wind speed, the lower the concentration of pollutants downwind of the emission site. Therefore, research on the wind corridor under weak or quiet wind conditions is necessary to comprehend and execute air exchange, hence minimizing air pollution and UHI. Wind corridors can be used to remove humidity and cool an area without using any energy, even if they do not always increase the natural wind speed in the area. It was discovered that Beijing, China, was extremely susceptible to the UHI impact. In order to prevent the UHI impact, the government therefore advocated the creation of land sites linking parks, water bodies, and buildings in order to create adequate wind corridors (Wang, et al., 2020).

German physicist Kress proposed the idea of "Urban ventilation corridors" under the name "Ventilationbahn." Before building any urban ventilation corridors to connect these two places and ensure efficient cool air circulation in the city center, he advised people to take into account two crucial factors, namely "Functioning Area" and "Compensating Area." Later, Mayer et al. (1994) divided urban ventilation corridors into four categories based on the quality of the air and the diverse air sources: normal, polluted, cool fresh air, and biometeorological-related (Ren, et al., 2018). Urban canyons and urban ventilation are other names for wind corridors.

In layman terms, Urban Ventilation is created simply by planning the building layouts of a place along with proper landscaping to enhance 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. For urban ventilation, the influence of climatic factors including wind speed, wind direction, temperature, and humidity is also crucial.

The appropriate distribution of wind flow inside a settlement is greatly influenced by the building spacing and proportions, including height and width in the wind's direction as well as perpendicular to the wind. The buildings' shapes should also be taken into consideration since they can aid recirculate wind away from the wind's direction by creating eddy vortices in their faces (Gu, et.al., 2020). German national guideline ‘Environmental meteorology climate and air pollution maps for cities and regions (VDI 3787-Part 1)’ names it as ‘Ventilation Lane’ and also gives a clear and detailed definition, which is the “Area for the mass transport of air near the ground which is preferred owing to direction, nature of the surface and width. In Japan, the urban ventilation corridor is also called ‘Kaze-no-Michi’.

Japanese researchers employed ventilation corridors, which they learned about from Germany's implementation experiences, primarily to cool down urban cores and enhance summertime thermal comfort for people. Since 2007, the Tokyo metropolitan region and several Japanese cities, including Yokohama, Nagoya, Osaka, and Fukuoka, have carried out their wind environment studies and developed plans for urban ventilation corridors.

More than 48 Chinese cities from 20 different provinces have either completed or are getting ready to complete their ventilation corridor plans as well as produce design actions and control mechanisms for local planning practices (Ren, et al., 2018).

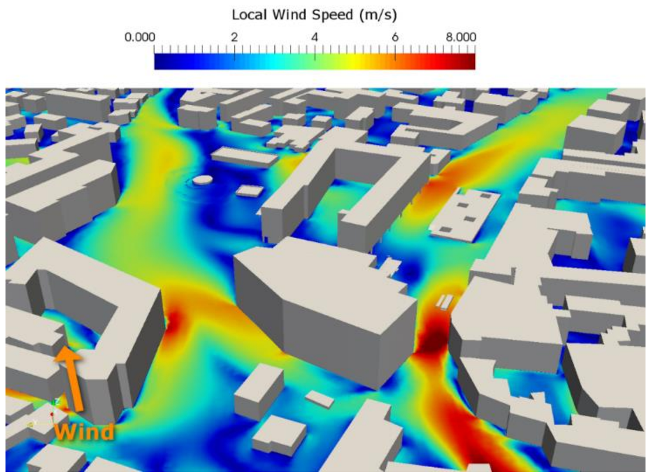


Figure 2- 6 Wind Corridor design in CFD (fyfluiddynamics.com)

Urban wind corridors are highly efficient in decreasing temperatures in hot weather and can be used to alleviate urban heat island effect. Furthermore, in zones where winds are relatively slow, wind corridors can accelerate the wind flow exponentially within the settlement through funnelling effect as explained by Bernoulli's theorem. High-density development has an impact on pedestrian comfort and safety as well as the ventilation of buildings in metropolitan settings. Lower permeability for urban air ventilation at the pedestrian level is caused by big podium constructions, consistent building heights, and tall, bulky high-rise building blocks with little room between them (Ng, 2009). It is possible to guarantee that cities' natural air channels stay undisturbed by properly planning and developing wind corridors, which will allow breezes from the suburbs to be directed into core regions and foster an interchange of fresh air between inner and suburban zones.

These corridors will play an objective role in reducing urban heat islands, reducing urban air pollution and energy consumption, and improving the quality of life for urban residents as the consequences of climate change become more pronounced. In the study and development of urban wind corridors, the experimental verification in the example of Stuttgart, Germany, might be viewed as a model. The city employed wind corridors to direct the local cold winds toward its center build-up region in order to best take use of the local meteorological and morphological circumstances. The city also offered advice to other towns on how these wind tunnels may be designed in the future. Following their successful application in other German cities, Stuttgart's "Urban Climatic Maps (UCMaps)" were made an official technical guideline for national urban planning. Since then, they have also been applied in the urban development of Hong Kong and Kaohsiung (in Taiwan) (Liu, et al., 2022).

Researchers have adopted an empirical approach to study and design in order to create Urban Ventilation Corridors, where they take into account various local climatic conditions and classify them as "Climatopes". At first, these divisions were mostly based on the different ways that land was used or the main function of structures constructed on the same piece of ground. For instance, "commercial land," "industrial land," "urban green spaces," and "open urban spaces" were among the 10 kinds of urban space employed in Stuttgart's urban climate planning system. Later, other elements including human heat loads, air pollution, noise, and socioeconomic situations started to be considered by the Climatopes system. The German city of Cassel, for instance, included other elements including building volume, urban heat islands, terrain, greenery, and surface roughness. Hong Kong, meanwhile, assessed the city's "ventilation potential" by taking into account other elements such building floor size, terrain, and green spaces in addition to employing building coverage, natural plant coverage, and open spaces to reflect the urban heat load.

Climatopes are now based mostly on assessments of heat load and ventilation capability. While there are several ways to gauge heating load, including objective metrics like degree of comfort, PET, and heat island intensity, there is no set of widely used metrics to gauge what is referred to as ventilation potential. Most frequently, cities choose a set of environmental variables connected to their well-ventilated regions based on local features as a method of evaluating ventilation potential. Building an efficient wind corridor is a laborious and challenging procedure since the empirical variables and elements that determine the best urban ventilation corridor design are always changing. Therefore, it is essential to create a set of indices that are more objective. In accordance with the size and geographical characteristics of the city in question, urban wind corridors must also be taken into account on macro- and microscales, that is, from the dual perspectives of regions and ties, cities and neighborhoods, or neighborhoods and buildings (Liu, et al., 2022).

Figure 2- 7 Urban Street Canyons (Hülle, et. al., 2013; P.C: Diego Torres Silvestre)

A street canyon is the basic construction block of contemporary cities. It is made up of two normally parallel rows of buildings that are divided by a street. The "aspect ratio" of a street canyon, which includes the proportion of building height (H) to street width (W), describes the geometry of the structure. A uniform street canyon is one without significant apertures in the walls and with an aspect ratio of about equal to 1. Street canyons are defined as having an aspect ratio of less than 0.5 and greater than 2 respectively. The length of canyon (L) illustrates the road distance between two main intersections subdividing the street canyon into short (L/H= 3), medium (L/H = 5) and long (L/H = 7). It has been shown that the street canyon's shape and orientation have an impact on both the interior and outdoor climates, the permeability of the airflow for urban ventilation, and the possibility for cooling the entire urban system. Therefore, both the overall energy consumption of urban structures and the thermal comfort of pedestrians are influenced by street design. The fundamental challenge a designer has when planning a street from a climatic perspective is the distinction between seasonal internal and outward demands.

The height-to-width ratio (H/W) and street direction are the most important urban factors thought to be responsible for the microclimatic variations in a street canyon, according to the majority of related studies. These factors have a direct impact on the possibility of airflow at street level and, consequently, on the urban microclimate. Despite the fact that both traditional and modern architectures make great efforts to create urban streets in accordance with climate, quantitative data on the optimum street design, based on scientific methodologies, is still needed in order to manage the comfort of the climate. The interaction of an incoming wind with the built environment affects urban airflow patterns. For human health, outdoor and interior thermal comfort, air quality, building energy efficiency, and the creation of a comfortable urban microclimate, the development of airflow inside a roadway canyon is crucial. For instance, the consequences of the urban heat island phenomenon may be lessened by the cooling effect of airflow, especially at night. Numerous studies show that passing through a constructed area alters the pattern of an existing regional wind. Designing the physical environment, particularly street canyons, is crucial for creating urban airflow patterns (Shishegar, 2013).

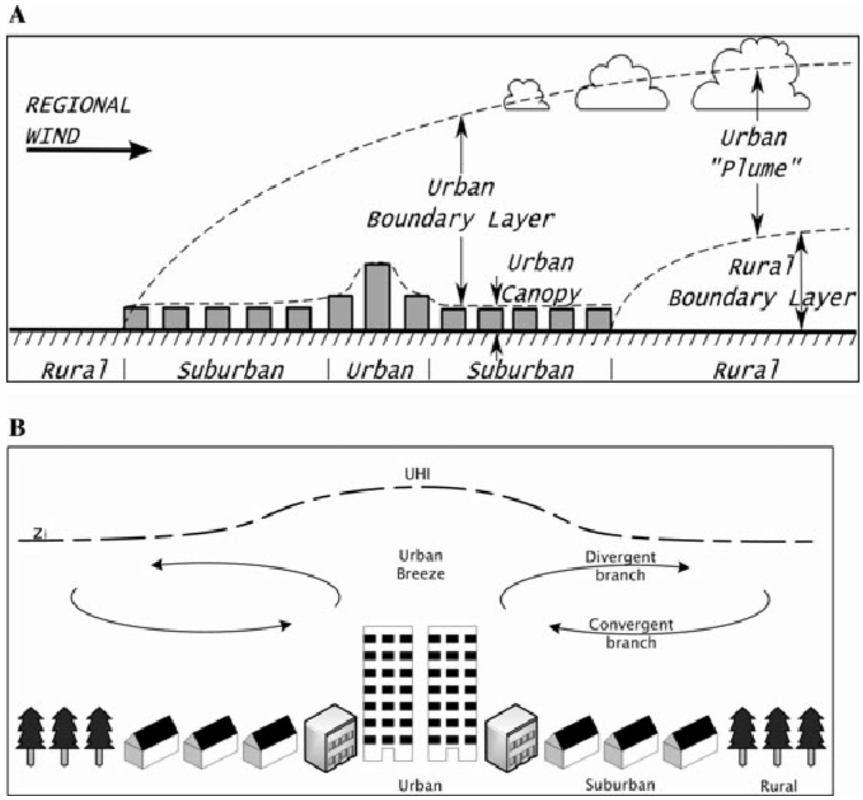


Figure 2- 8 Urban Canopy Layer and Urban Boundary Layer (Hidalgo, et.al., 2008)

Urban boundary layer and urban canopy layer are the two primary layers of the air that cover cities. The urban canopy layer, which is below the roofs in the gaps between buildings, is affected by convection as a result of solar radiation hitting the ground and building facades. The urban border layer is higher than the typical building height. The primary variables influencing air temperature in the urban border layer are heat transmission, pollution emission, evaporation and transpiration, and generally modern urban growth. 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, the urban canopy layer has a slower airflow than the nearby rural areas.

When the airflow in the urban boundary layer is approximately normal to street axis, three airflow regimes with different characteristics could occur based on the aspect ratio (H/W) and building ratio (L/W): (a) the isolated roughness regime; (b) the wake interference regime; and (c) the skimming regime. The conversion from one regime to another happens at critical combinations of H/W and L/W. The major flow characteristics associated with each of the airflow regimes over a variety of obstacles. When there is no interaction between windward and leeward flows, the isolated roughness regime occurs between evenly spaced structures, much as a wind movement around an isolated barrier. The wakes are disrupted when the H/W ratio rises, resulting in a wake interference regime. A constant circulatory vortex is produced in the street canyon as a result of the H/W ratio's continued rise isolating the canyon from the urban border layer's circulating air. This steady circulation vortex creates a skimming regime, which is most common in cities. Therefore, it could be concluded that there is slower airflow in deep street canyons in comparison with uniform or shallow ones (Shishegar, 2013).

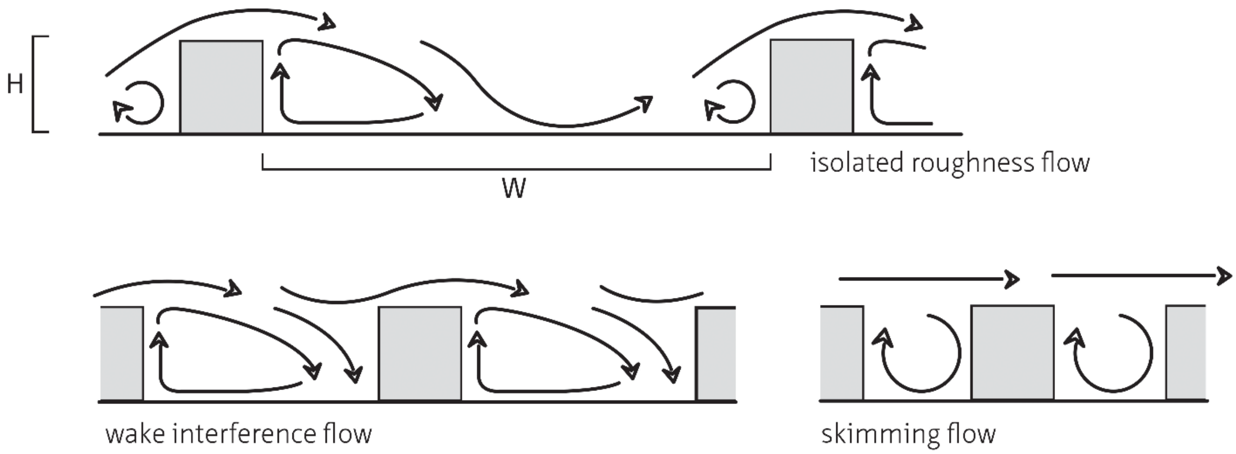


Figure 2- 9 Types of Wind Flow around buildings (urbanclimate.nl)

Numerous research has been conducted to ascertain how roadway layout affect air flow. As an illustration, Johansson (2006) investigated for more than 1.5 years the impact of street geometry on airflow in the cities of Morocco using case studies. 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 microclimate 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 summertime average wind speed in the shallow street canyon was 0.7 m/s, while in the winter it was 0.8 m/s.

Another research by Al-Sallal and Al-Rais in Dubai shown that wind speed may be increased through small roadway canyons (4 m and smaller), improving passive cooling performance while producing eddies at bending angles. Wind penetrated further into the conventionally small streets with higher wind speeds (5 m/s), improving the possibility for thermal comfort. The majority of locations (49–57% 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. In this term, it was found that strategically placing a few blocks of high-rise towers will improve the velocity within the street canyon when the airflow in parallel or perpendicular to the canyon.

Additionally, when tall skyscrapers were built in a street canyon, the temperature dropped. By putting up a few high-rise towers, the temperature may be lowered by up to 1°C while the velocity is enhanced for parallel flow by up to 90%. The temperature drops by 1.1°C when the velocity is raised up to 10 times for perpendicular flow. Additionally, creating a few tall structures among the nearby buildings constrained in the urban canopy layer might result in U-shaped vortices and extra wakes that increase air exchange. 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. Straight air flow causes extreme heat (in hot-dry climates) or cold (in cold-dry climates) wind to blow into the streets when there is a lack of vegetation and suitable coverings. Narrow and curving roadways lessen the impact of cold, hot, and severe winds (Shishegar, 2013).

2.4 Building Bylaws

2.4.1 Introduction

Building bylaws are essentially the guidelines that must be followed when designing and constructing structures in a community. These rules, which are municipal and federally defined, are intended to increase building use and safety both within and outside the structure. There is a risk of overdevelopment of real estate due to the dramatic increase in housing demand and population, and there is no indication that this trend will slow down soon. Building ordinances must thus be scrupulously adhered to in order to avoid haphazard growth and conflicts of interest among developers. To ensure that these regulations maintain the mobility and use of the buildings while maintaining their efficiency and safety, building bylaws are created by professional designers and engineers within a committee that includes politicians, public representatives, and other scientific specialists. These regulate the design and construction aspects of building activity. Building bye-laws also regulate the amount of open space that may be included in a project. This is done to avoid turning a city into a concrete jungle as a result of growth.

Ground Coverage

The ratio of the building area divided by the land area is ground coverage by building. Building area means the floor space of a building when looking down at it from the sky. It is calculated by

Ground coverage (%) = (Building area/Land area) x 100

Floor Area Ratio

The ratio of total floor area divided by land area is floor area ratio. Here, total floor area refers to the total amount of floor space in a building. It is calculated by:

FAR = Total floor area/ Land area

Height of building

Building height is the average maximum vertical height of a building or structure measured from finished grade to the highest point along each building elevation at a minimum of three evenly spaced locations (division).

2.4.2 Relationship between Ground coverage, FAR and Height of building

Ground coverage (%) = (Building area/Land area) x 100

Maximum ground coverage area = (Ground coverage (%) x Land area)/100

FAR = Total floor area/ Land area

Total floor area = FAR x Land area

Area of one floor = Total floor area/ allowable ground coverage Area

No . of storey = Total Floor Area/ Area of one floor

Setback

The term "building setback" means the required separation between a street line (and/or right-of-way line) and a building or structure (Development, n.d.).

2.4.3 Building Bylaws of Siddharthanagar Municipality

In order to facilitatte the design and arrangement of buildings for the modelling and simulation for the research, the bylaws of Siddarthanagar Municipality, Lumbini is taken as a reference to maintain scientific validity and accuracy of the data implemented in CFD simulation through ANSYS Fluent.

FAR and Ground Coverage

Table 2- 2 FAR and GC of Siddharthanagar

Building Type	**Land Area**	**Maximum ground coverage**	**FAR**
Residential	Upto 250 sq.m	70%	2.5
Residential	More than 250 sq.m	60%	2.5
Commercial cum Residential	Upto 250 sq.m	60%	3
Commercial cum Residential	More than 250 sq.m	50%	3
School, College		40%	1.25
Polyclinic, Nursing Home		35%	1.25
Hostel		50%	2

Setback

The minimum setback for highway road is 6m while while for inner roads the minimum setback is 1.5 m.

Table 2- 3 Setback proposed by Siddharthanagar

Building Type	Minimum Setback (m)	
	Front	Sides and back
Residential	1.5	1
Educational	5	3
Institutional	5	3
Theatre, community hall	15	6
Commercial	5	2
Hotel	5	2
Star Hotel	10	3

The minimum storey height for residential type is 2.9 m. The maximum permissible height of building is 17m.

Shading devices used for rain and sun protection must be less than 1m in length and must be placed under setback.

The height of the boundary wall shall not exceed 4 feet when built with brick or stone masonry. If necessary, a 3ft height may be added with wire mesh.

2.5 CFD Analysis

2.5.1 Introduction

Wind corridors or Urban Ventilation Corridors are specifically designed to optimized the wind flow within the settlement to induce proper air circulation and temperature reduction of the area. As a consequence, both inside and outside of buildings, thermal comfort for pedestrians is improved. Multiple simulation programs have been used to streamline the developing process, even though they have been experimentally created and studied in terms of Climatopes, which originated in Germany. In a virtual system, simulation is essentially a technique for testing theories in a controlled setting. Even very large-scale experiments may be carried out more easily than when several variables are genuinely controlled in the experiment. The use of computers and other computing devices speeds up the efficient and accurate simulation and modeling of projects.

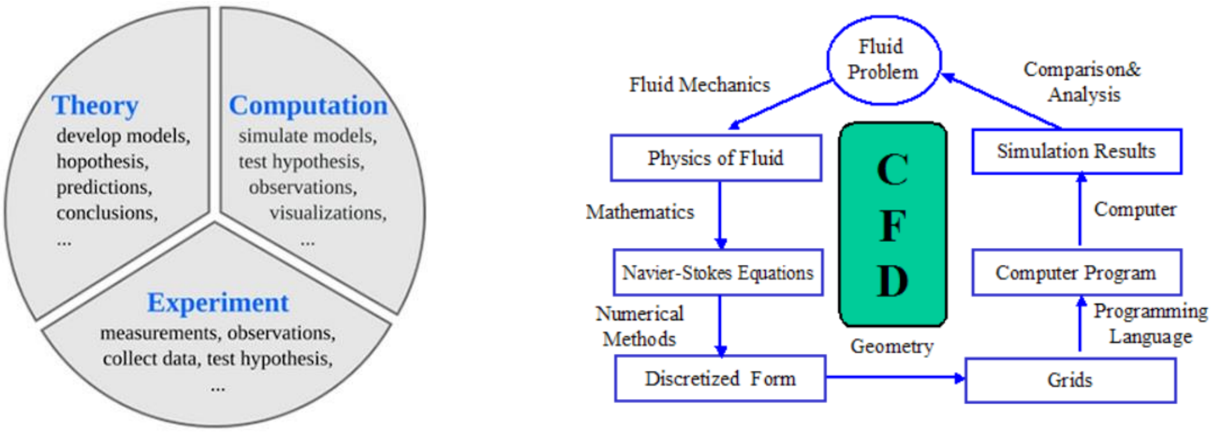


Figure 2- 10 CFD Analysis Chart (cfdflowengineering, 2018), (Studymafia, n.d)

Computation or CFD A system called fluid dynamics uses modeling (mathematical physical problem formulation) and numerical techniques to answer issues about fluid flow and its nature (discretization methods, solvers, numerical parameters, and grid generations, etc.) n.d. (Studymafia). You may analyze intricate issues involving fluid-fluid, fluid-solid, or fluid-gas interaction using CFD. Aerodynamics and hydrodynamics are two engineering disciplines where CFD analyses are routinely used to produce quantities like lift and drag or field characteristics like pressures and velocities. Physical laws are connected to fluid dynamics by partial differential equations (cfdflowengineering, 2018).

Lewis Fry Richardson made the first attempt to compute fluid flow, with applications for forecasting the weather. A "prediction factory" with 64,000 people "computers" was what he had in mind. Each "computer" was placed at varying heights around a spherical globe, occupying computational cells that matched specific places on maps, as is the case for northern Europe in the image below. His approach involves entering weather observation data at the appropriate grid places, then resolving the equations by forward-stepping.

CFD could not truly start to take off until electric computers were created, replacing Richardson's human computers with digital computing. With the introduction of the ENIAC programmable digital computer, this typically started in the 1940s. In the early 1950s, small-scale simulations started to appear in the scientific literature (Bhattacharyya, et.al., 2021).

- **Until 1910:** Improvements on mathematical models and numerical methods.
- **1910 – 1940:** Integration of models and methods to generate numerical solutions based on hand calculations[1].
- **1940 – 1950:** Transition to computer-based calculations with early computers (ENIAC)[3]. Solution for flow around a cylinder by Kawaguti with a mechanical desk calculator in 1953[8].
- **1950 – 1960:** Initial study using computers to model fluid flow based on the Navier-Stokes equations by Los Alamos National Lab, US. Evaluation of vorticity – stream function method[4]. First implementation for 2D, transient, incompressible flow in the world[6].
- **1960 – 1970:** First scientific paper "Calculation of potential flow about arbitrary bodies" was published about computational analysis of 3D bodies by Hess and Smith in 1967[5]. Generation of commercial codes. Contribution of various methods such as k-ε turbulence model, Arbitrary Lagrangian-Eulerian, SIMPLE algorithm which are all still broadly used[6].
- **1970 – 1980:** Codes generated by Boeing, NASA and some have unveiled and started to use several yields such as submarines, surface ships, automobiles, helicopters and aircrafts[4,6].
- **1980 – 1990:** Improvement of accurate solutions of transonic flows in the three-dimensional case by Jameson et. al. Commercial codes have started to implement through both academia and industry[7].
- **1990 – Present:** Thorough developments in Informatics: worldwide usage of CFD virtually in every sector.

Figure 2- 11 Brief History of CFD (Simscale, 2021)

2.5.2 Use of CFD

Applying the fundamental laws of mechanics to a fluid gives the governing equations for a fluid. The conservation of mass equation is:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{V}) = 0$$

And the equation of conservation of momentum is:

$$\rho \frac{\partial \vec{V}}{\partial t} + \rho(\vec{V} \cdot \nabla)\vec{V} = -\nabla p + \rho \vec{g} + \nabla \cdot \tau_{ij}$$

These equations along with the conservation of energy equation form a set of coupled, non-linear partial differential equations. It is not possible to solve these equations analytically for most engineering problems. However, it is possible to obtain approximate computer-based solutions to the governing equations for a variety of engineering problems. This is the subject matter of Computational Fluid Dynamics (CFD) (Studymafia, n.d).

CFD analyses are more affordable and quicker than traditional testing for data collecting since they offer a greater potential to speed up the design process. Additionally, in real-world testing, only a few variables are assessed at once, but in a CFD study, all necessary variables may be monitored simultaneously and with a high degree of spatial and temporal precision. It should be emphasized that these CFD studies cannot completely eliminate physical testing processes because they only approximate a true physical solution. However, tests should still be run for verification purposes (cfdflowengineering, 2018).

CFD is helpful because it provides insight into flow patterns that are challenging, costly, or impractical to analyze using conventional (experimental) methods (Kuzmin, 2017). Some differences between experiments and CFD simulations are presented as follows:

EXPERIMENTS	SIMULATIONS
Quantitative **description** of flow phenomena using measurements ■ for one quantity at a time ■ at a limited number of points and time instants ■ for a laboratory-scale model ■ for a limited range of flow problems and operating conditions	Quantitative **prediction** of flow phenomena using CFD software ■ for all desired quantities ■ with high resolution in space and time ■ for the actual flow domain ■ for virtually any problem and realistic operating conditions
Error sources: measurement errors, flow disturbances by the probes	Error sources: modeling, discretization, iteration, implementation

Figure 2- 12 Difference between Experiments and CFD Simulations (Kuzmin, 2017)

Table 2- 4 Comparison of Simulation and Experiments (Studymafia, n.d)

	Simulation (CFD)	**Experiments (Traditional)**
Cost	Cheap	Expensive
Time	Short	Long
Scale	Any	Small/Middle
Information	All	Measure Point
Repeatability	Yes	Some
Safety	Yes	Sometimes Dangerous

2.5.3 Process of conducting CFD Simulation Research:

CFD basically consists of three phases, namely:

1. **Pre-processing**

In this phase the problem statement is transformed into an idealized and discretized computer model. Assumptions are made concerning the type of flow to be modelled (viscous/inviscid, compressible/incompressible, steady/non steady). Other processes involved are mesh generation and application of initial- and boundary conditions.

2. **Solving**

The actual computations are performed by the solver, and in this solving phase computational power is required. There are multiple solvers available, varying in efficiency and capability of solving certain physical phenomena.

3. **Post-processing**

Finally, the obtained results are visualized and analysed in the post processing phase. At this stage the analyst can verify the results and conclusions can be drawn based on the obtained results. Ways of presenting the obtained results are for example static or moving pictures, graphs or tables (cfdflowengineering, 2018).

Specific process of solving CFD problems are as follows:

1. **Formulate the Flow Problem**

Here, we determine what are our objectives to be completed (Engineering quantities Performance Proof of concept), what are the possible variables identified (Freestream conditions Geometry Configuration) and what approach of analysis is the most suitable to process the problem (Steady or unsteady flow? Are viscous forces important? Are shocks present? What equations to solve? What other flow models are needed?).

2. **Model the Geometry**

Here, we identify what geometrical features are important, whether any simplification of geometry is required and what computational format are available for geometry.

3. **Model the Flow (Computational) Domain**

Here we start modelling the flow domain for the problem. The flow (computational) domain is the control volume (bounded by the control surface) in which the flow field is computed.

4. **Generate the Grid**

Now, a grid is generated within the flow domain for the next step of solution of the flow problems. The grid consists of finite-volume cells on which the CFD equations are approximated. The grids can be both structured and unstructured. The flow domain may be divided into zones for various reasons:

- Simplify grid generation.
- Reduce memory requirements.

- Divide grid for parallel computation.

5. **Specify the Boundary Conditions**

Numerical conditions need to be applied at the boundaries of the flow domain and zones. We specify the types of boundary condition: a. Viscous (no-slip) wall; b. Inflow / outflow; c. Reflection Coupled. Zone-to-zone boundaries and overlapped zones require topology and coupling specifications. Specification of boundary conditions may be part of grid generation package. Additional inputs may be required in the flow code input process (i.e., flow rates, pressures, etc.).

6. **Specify the Initial Conditions**

We determine the initial condition of the fluid flow to begin the solution process. One choice is to start with a uniform flow field with conditions of the freestream or inflow conditions.

7. **Set up the CFD Simulation**

Now, we setup the simulation file according to our need. The simulation requires several input files: Grid file, Initial solution file, Input data file, Auxiliary files (i.e., multi-processor, local boundary conditions, chemistry, etc.

8. **Conduct the CFD Simulation**

Now, we carry out the simulation and modelling process in the computer. The computer needs powerful CPU to carry out the program efficiently and smoothly. The researcher must make sure that his/her inputs are correctly entered to obtain feasible accurate outputs.

9. **Examine and Process the CFD Results**

After the simulation process, we obtain the results according to the boundary conditions and input entered. The obtained outputs are then analysed and processed in the software to be viewed by the user. We measure multiple View flow properties (Mach, pressure, vectors) to get overall view of flow.

10. **Further Analysis**

Once the program has undergone multiple iterations and the feasible accurate results are obtained. The researcher has to decide whether to carry out further analysis or the result obtained is accurate enough. There can be various reasons of carrying out the analysis again such as change in the physical model, change in the numerical algorithm model, changing or refining the boundary conditions and grids and changing initial condition to obtain different results in multiple iterations.

11. **Report the Findings**

Finally, after the results obtained are approved, then the overall process and results along with every variable used will be properly documented and reported. CFD results, like any other data, should be reported along with some idea of the level of error that it contains and indications of how much confidence one has in the data. Some points to be included in the reports are:

- What are engineering results and uncertainty of those results?
- How much error is there in the iterative and grid convergence?
- How sensitive are the results to model parameters (turbulence, etc…)
- How sensitive are the results to algorithm parameters
- How do the results compare to similar experimental or theoretical data? (Centre, n.d)

2.5.4 Application of CFD

CFD is a versatile tool which can be used to solve any problems regarding the fluid flow. From ever-growing transportation sector to design of hydropower turbines/wind turbines, CFD is a powerful ally to every researcher. The modelling of wind flow around buildings is also an exciting scope of CFD. CFD can be used to simulate the flow over a vehicle. For instance, it can be used to study the interaction of propellers or rotors with the aircraft fuselage.

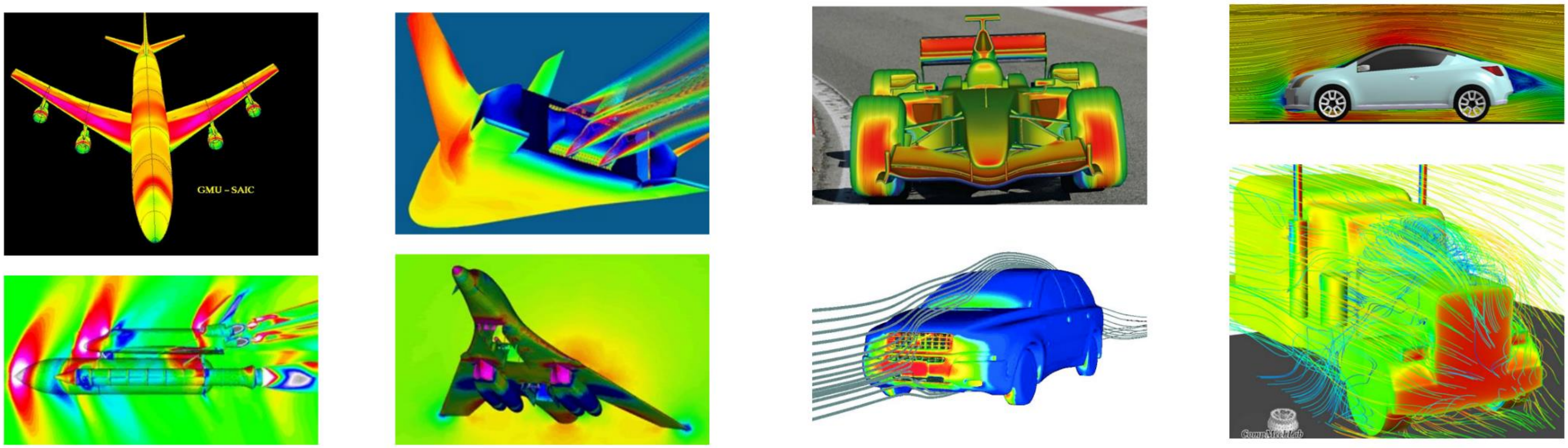


Figure 2- 13 Use of CFD in Automobile Industry (Kuzmin, 2017)

CFD can also be used in Chemical Industry to monitor and evaluate the motion of fluid (chemicals) in the production containers along with also the motion of fluids during combination. CFD has also been prominently used to design wind corridors for quite a few years now. Using CFD we can monitor the flow of wind in a settlement, through the buildings and around the buildings as well.

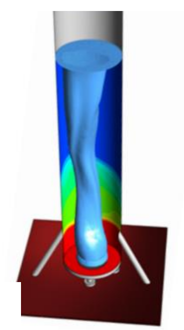


Figure 2- 14 Use of CFD in Chemical Industries (Kuzmin, 2017)

2.5.5 Limitation of CFD

Even though CFD is a powerful tool that can be used in multiple sectors of research, it also has some limitations within which researchers have to work. Some of the limitations are:

- The CFD solutions can only be as accurate as the physical models on which they are based.
- Solving equations on a computer invariably introduces numerical errors.
- Round-off error: due to finite word size available on the computer. Round-off errors will always exist (though they can be small in most cases).
- Truncation error: due to approximations in the numerical models. Truncation errors will go to zero as the grid is refined. Mesh refinement is one way to deal with truncation error.
- Boundary conditions: As with physical models, the accuracy of the CFD solution is only as good as the initial/boundary conditions provided to the numerical model (cfdflowengineering, 2018)**.**

2.5.6 Use of CFD in designing wind corridors

CFD is an effective method for designing and resolving issues with fluid flow in a regulated volume or zone. Since wind is likewise a fluid, we can simply use CFD methods to evaluate its velocity and makeup. With the use of CFD modeling, it is simple to calculate the direction and speed of a community's wind flow, which enables us to determine the most effective approach to plan wind corridors. We may examine a settlement to see how the wind moves through the structures and whether there are any obstructions or things that cause the wind to flow in an unusual direction. Monitoring the dispersion of contaminants in the atmosphere may also be done using CFD.

In this thesis research, we will also be using CFD simulation to design wind corridors to combat against UHI effect. There are many CFD software such as Solidworks, ANSYS Fluent, Simscale, etc. Each software is powerful in its own way but for the ease and accuracy of the objectives of the research, ANSYS Fluent will be used to monitor the flow of wind and thus model the wind corridors.

2.6 ANSYS FLUENT

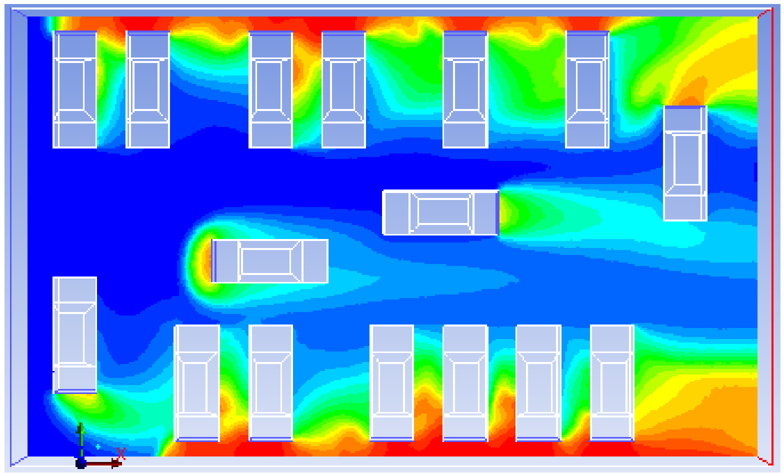


Figure 2- 15 Simulation in ANSYS Fluent (Ahmed, 2013)

1. What is ANSYS Fluent?

It is a CFD program that is specifically designed for modeling fluid flow and heat transfer. You may simulate and model various varieties of fluid processes with this CFD program, as well as fluid-structure multiphysics interactions. Additionally, ANSYS Fluent includes a wide range of physical modeling features that are necessary for industrial applications including fluid flow, heat transfer, turbulence, and reactions.

There are a total of five components in each Fluent system. For the entire Fluent system to work properly, each component must be configured appropriately. Each component has a unique set of functions. Each element has a unique meaning, which is succinctly defined as follows:

2. Components of ANSYS Fluent

a. Geometry

'Geometry' is the ANSYS Fluent workspace for creating 2D or 3D models that need to be processed further for their heat transfer or fluid flow.

b. ANSYS Meshing

Meshing is an integral part of all the analysis systems in ANSYS. Meshing allows the geometries to be broken into small polygons so that each polygon can be processed separately to generate the results. The finer the mesh, the better the results.

c. Setup and Solution

Double-clicking over setup launches the ANSYS Fluent. Before Fluent opens, a Fluent Launcher opens to set the pre-launch settings. It allows you to select your dimensions, display options, processing options and much more. One of the many great features of ANSYS Fluent is to its ability to use parallel processing or multiple processors of HPC.

d. Results CFD Post

The ‘results’ is a CFD post-processor for Fluent. Under CFD-post, the outputs from the variables that are set up in Fluent can be visually seen and analysed. CFD post is an important component and essential to the users as here they can see a simulated demonstration of how their product will behave in real life scenarios (Alam, n.d).

2.7 Key Theories

In the case of fluid dynamics, the whole concept is based upon multiple theories which are ultimately utilized in designing CFD software such as ANSYS Fluent. In order to understand and use CFD tools, one must be quite knowledgeable of multiple key theories upon which the tool is based upon. Some of the key theories are:

A. Fluid Flow and it’s types:

The motion of fluid over a surface can be defined as fluid flow. There are basically Six types of fluid flows according to their physical nature. They are:

1. **Laminar Flow and Turbulent Flow**
   Laminar fluid flow is defined as in which the fluid particles move to streamline or along well-defined paths, which means the flow is straight and parallel.

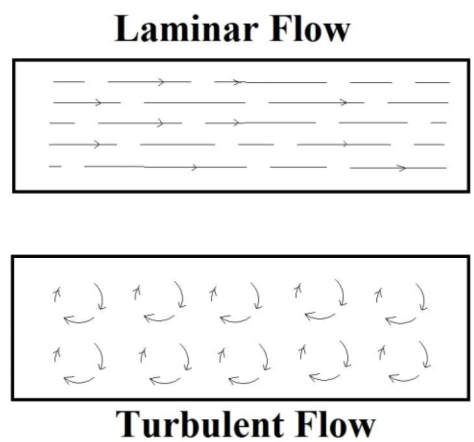


Figure 2- 16 Laminar and Turbulent Flow (Kumar, 2022)

The term "turbulent fluid flow" refers to a flow in which the fluid particles travel in a zigzag pattern rather than in a streamline or along clearly defined routes, i.e., the flow is not straight and parallel but rather zigzags instead. The Reynolds number (Re) is typically used to distinguish between laminar and turbulent flows. Laminar flow may become turbulent when certain factors, such as surface geometry, surface roughness, free-stream velocity, surface temperature, and fluid type, come into play (CFD Modelling, 2018).

$$Re = \frac{\text{Inertia forces}}{\text{Viscous}} = \frac{\mathcal{V} L_c}{\nu} = \frac{\rho \mathcal{V} L_c}{\mu}$$

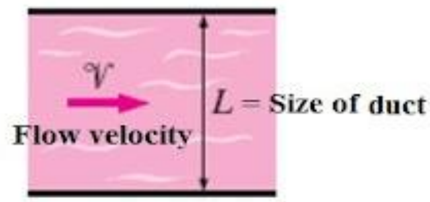


Figure 2- 17 Reynold's Number (CFD Modelling, 2018)

- **$Re < Re_{cr}$: Laminar flow**
- **$Re > Re_{cr}$: Turbulent flow**

2. **Rotational and Irrotational flow**

   Rotational fluid flow is defined as in which the fluid particles move to streamline or along well-defined paths, which means the flow is straight and also rotates about its own axis is known as rational flow.

   Irrational fluid flow is defined as in which the fluid particles move to streamline or along well-defined paths, but it does not rotate about its own axis is known as Irrational flow.

3. **Steady and Unsteady flow**

   The steady flow is defined as the flow in which the velocity, pressure, and density are constant at any point with respect to time.

   $$\frac{\partial V}{\partial t} = 0 \quad \frac{\partial p}{\partial t} = 0 \quad \frac{\partial j}{\partial t} = 0$$

   Whereas an unsteady flow is just opposite to steady flow. The unsteady flow is defined as the flow in which the velocity, pressure, and density are not constant and it is different with respect to time.

   $$\frac{\partial V}{\partial t} \neq 0 \quad \frac{\partial p}{\partial t} \neq 0 \quad \frac{\partial j}{\partial t} \neq 0$$

4. **Uniform and non-uniform flow**

   Uniform flow can be defined as the type of flow in which the velocity does not change with respect to space at any given time (i.e. length of direction of the flow).

   $$\left(\frac{\partial V}{\partial t}\right)_{\text{t is a constant}} = 0$$

   Non uniform flow can be defined as the type of flow in which the velocity does change with respect to space at any given time (I.e., length of direction of the flow).

$$\left(\frac{\partial V}{\partial t}\right)_{\text{t is a constant}} \neq \mathbf{0}$$

5. **One, two, and three-dimensional flow**

   In one-dimensional fluid flow, the name itself indicates fluid is moving in only one dimension (Either X, Z, or Y).

   u=f(x), v=0 and w=0

   In two-dimensional fluid flow, the name itself indicates two the fluid is moving in two dimensions (Either XY, ZX, or YX).

   u= f1(x,y,), v= f2(x,y,) and w= 0.

   In three-dimensional fluid flow, the name indicates three the fluid is moving in three dimensions (Either XYZ, ZYX, or YZX).

   u= $f_1(x,y,z)$***, v=*** $f_2(x,y,z)$ ***and w=*** $f_3(x,y,z)$***.***

6. **Compressible or Incompressible flow.**

   Compressible flow can be defined as the flow in which the density is not constant that means from point-to-point density is changing therefore it is known as compressible flow.

   J ≠ constant

   Incompressible flow can be defined as the flow in which the density is constant that means from point-to-point density is not changing therefore it is known as Incompressible flow.

   J = constant (Kumar, 2022)

B. Conservation Law

Navier-Stokes equations are the governing equations of Computational Fluid Dynamics. It is based on the conservation law of physical properties of fluid. The principle of conservational law is the change of properties, for example mass, energy, and momentum, in an object is decided by the input and output.

- **Conservation of Mass**

  It basically states that the amount of mass entering into a control volume (m_{in}) is equal to the amount of mass leaving the control volume (m_{out}).

  $$\frac{dM}{dt} = \dot{m}_{in} - \dot{m}_{out}$$

If $\dot{m}_{in} - \dot{m}_{out} = 0$, we have

$$\frac{dM}{dt} = 0$$

Which means

$$M = const$$

- **Conservation of Momentum**

  The momentum observation principle can be mathematically represented as:

  $m_1u_1 + m_2u_2 = m_1v_1 + m_2v_2$

  In the equation, m_1 and m_2 are masses of the bodies, u_1 and u_2 are the initial velocities of the body.

  v_1 and v_2 are the final velocities of the bodies.

C. Flow Rate

- **Mass flow rate**

$$\dot{m} = \frac{dm}{dt} = \frac{\text{mass}}{\text{time taken to accumulate mass}}$$

- **Volume flow rate – Discharge**

  Generally, the volume flow rate is also known as discharge and it is denoted by Q.

$$Discharge, Q = \frac{Volume}{Time}$$

D. Continuity Equation

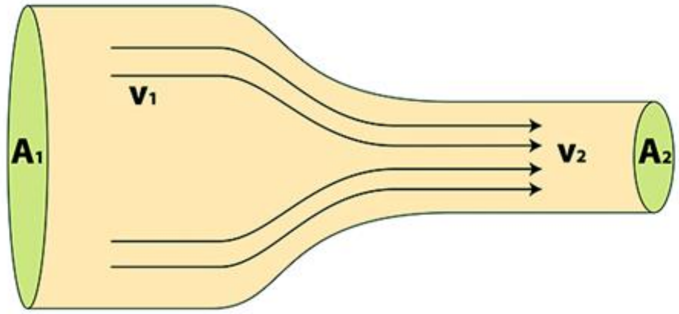


Figure 2- 18 Continuity Equation (Byjuus, n.d)

The continuity equation in fluid dynamics describes that in any steady state process, the rate at which mass leaves the system is equal to the rate at which mass enters a system.

The differential form of the continuity equation is:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho u) = 0$$

Where,

t = Time

p = Fluid density

u = flow velocity vector field (Byjuus, n.d).

E. Bernoulli's Equation

Bernoulli's principle states that for an inviscid flow of a non-conducting fluid, an increase in the speed of the fluid occurs simultaneously with a decrease in pressure or decrease in the potential energy. Bernoulli's principle can be applied to various types of liquid flow, resulting in what is denoted as Bernoulli's equation. The simple form of Bernoulli's principle is applicable for incompressible flows.

$$P + \frac{1}{2}\rho v^2 + \rho g h = constant$$

Where,

- P is the pressure
- v is the velocity of the fluid
- ρ is the density of the fluid
- h is the height of the pipe from which the fluid is flowing (Byjuus, n.d)

F. Navier-Stokes Equation

To simplify the Navier-Stokes equations, we can rewrite them as the general form.

$$\frac{\partial(\rho\Phi)}{\partial t} + \frac{\partial}{\partial x_i}\left(\rho U_i \Phi - \Gamma_\Phi \frac{\partial \Phi}{\partial x_i}\right) = q_\Phi$$

When Φ =1, U_j, T, we can respectively get continuity equation, momentum equation and energy equation (studymafia.org, n.d).

2.8 Previous Research Findings

Wong, et.al., (2010) studies the concept of designating of urban ventilation corridors in the peninsula of Kowloon in Hongkong to mitigate urban heat island formation. They used the concept of building frontal area index with GIS mapping to analyze and plot the least cost path. The concluded that the living environment in future urban renewal projects may easily be improved by a simple shift in location and orientation of buildings, while maintaining the same plot ratios, and the creation of ventilation corridors will not decrease ventilation elsewhere. They also suggested that in further research the type of surface should also be considered as cooling capacity of wind differs between vegetation and impervious surfaces.

Eldesoky, et.al. (2020) studied the use of Land based data, satellite imaging and GIS mapping system along with spatial analysis to study the UHI effect. Their study area was based in the city of Milan whose wind speeds are about 4m/s. They also utilised Least Cost Path (LCP) analysis to design urban ventilation corridors. They determined that the effect of green ventilation corridors is more significant at night, which is adequate for mitigating the UHI effect that is most developed at this time. Also, it has been demonstrated that even under very calm wind conditions, there is still an opportunity for maximizing the benefits of greening measures in terms of lowering high temperature. This research finding is very promising, especially for cities where wind is a marginal potentiality like Milan.

Wicht, et.al. (2017) studied the aid of remote sensing data in detecting ventilation corridors using a spatio temporal approach in Warsaw in east-central Poland. They found out that ventilation corridors played a major role in increasing thermal comfort in Warsaw by mitigating the UHI and pollutants disperse. They also suggested that preservation of green areas lead to better thermal conditions through increasing cooling capacity of wind corridors. They suggested that further researchers looked at investigating the influence of hight vegetation in circulation systems.

Wicht, et.al. (2017) studies the use of LIDAR in mapping Urban Ventilation Corridors in Warsaw, Poland. In this study, they determined the areas in Warshaw that could act as wind corridors in the future if designed properly by using roughness and porosity parameters but they failed to study the dimensions and continuity of those areas which might affect the cooling capacities.

Ren, et.al. (2018) studied the development of breathable cities by adopting urban ventilation corridors in the cities of China. Here they reviewed the plans and policies developed by China Government in the design and development of wind corridors.

Oke (1988) studies the effect of street design and urban canopy in the local climate of a region where he suggested that that H/W ~ 0.65 should ensure considerable pedestrian protection and, therefore, a minimum acceptable value may be somewhere in the middle of the wake interference regime at about 0.4.

Shishegar (2013) studied the relationship between street canyons on air flow and local urban microclimate. He concluded that since streets covers around a quarter of urban areas, designing streets is a key issue in a global approach for an environmental urban design. The geometry of streets (H/W and L/w ratios) and orientation directly influence the airflow and solar access in urban canyon and therefore thermal comfort at pedestrian level. A wider street provides better mixing of air and consequently better airflow in the urban canyon. In addition, better ventilation could be occurred in a street with various building heights.

Su, et.al. (2016) studied the application of Ventilation Corridor Planning in Urban New Area of Xixian New Area, China whose mean wind speed is around 1.7m/s. They used CFD along with studying the dimension of potential corridors to carry out the research. They concluded that urban ventilation corridor planning is very important in the ecological environment of Xixian City as they improve the wind and atmospheric environment. However, they suggested that further research on ventilation efficiency of the air corridors and improving the wind environment in the area outside the wind corridors should be carried out.

Wang, et.al. (2015) studied multiple strategies in mitigating Urban Heat Island Effect in Toronto, Canada where he also studied the use of Wind Corridors and its cooling capacity. He used ENVI-met as a CFD software tool to study the effects of cool surfaces. Utilizing three concepts of cool pavements, increased vegetations and wind corridors can reduce the temperature by around 5°C during summer. The research however failed to consider heat emissions from HVAC as well as Transportation related heat fluxes.

Zhang, et.al. (2021) studied the Urban Ventilation Assessment with Local Climate Zone (LCZ) parameters by using Mathematic Model on Simulation with the Porous Media Urban Canopy Model in the Luoyang City, China. The study revealed the effects of building height and surface cover fraction on the wind field within the urban canopy layer, including the upward and downward wind within the canopy layer model. On the other hand, the heat balance models could also be conducted on LCZ. Results showed the potential capabilities and advantages of LCZ and porous media method in urban wind assessment.

Gu, et.al. (2020) studied the use of spatial planning for urban ventilation corridors in urban climatology in Bozhou City in China whose average wind speed is around 2019 m/s. They concluded that effective construction of urban ventilation corridors plays an important role in improving urban climate. The study, based on the main urban area of Bozhou as the research object, showed that the ventilation corridor planning (VCP) model, integrating three commonly cited benefits of ventilation corridors (background wind environment, ventilation potential and heat island intensity), can be used as a generalizable approach to spatial planning. This approach can help the designer and local communities to make detailed value judgments and timely feedback adjustments, and finally propose a relatively comprehensive plan that incorporates a wide range of economic and environmental benefits.

Xie, et.al. (2020) studied the use of circuit theory to simulate urban ventilation corridors in Wuhan City, China. They used the concept of electricity flow to simulate wind corridors through the use of resistors. They interpreted current, voltage and resistance in the field of wind movement. They concluded that unlike accurate simulation of air flow around buildings in wind tunnel experiments and CFD simulations, the urban ventilation environment analysis method based on numerical simulation is an empirical model based on the relationship between urban morphological parameters and experimental wind data.

Chang, et.al. (2018) studied the use of CFD (WinAir) and GIS in the development of Urban Ventilation Corridors in Changchun City, China where the yearly average wind speeds is around 4m/s. They concluded that the combined use of GIS and CFD tools can help to develop and optimize accurate Urban Ventilation Corridors for research purposes.

Wang, et al. (2020) studied the use of large eddy simulations by using Parallelized LES Model (PALM) in identification of pedestrian-level ventilation corridors in downtown Beijing city, China whose average wind speed was taken as 1.5m/s. They suggested that the use of PALM and CFD can help in developing pedestrian-level wind corridors which help in reducing UHI effect but the limitation of the study is that only buildings were considered whereas focus should also be given to effect of urban greenery and thermal conditions of pedestrian level ventilation.

Liu, et.al. (2020) studied the detection of wind corridors based on climatopes in the city of central Ji'nan, China whose average wind speed is 3m/s. They utilised GIS technologies :VPC and CSL along with remove satellite imaging to develop new set of Climatopes. They conclued that the use of Climatopes can facilitate in mitigating UHI effect in an area and increase air circulation.

Rajagopalan, et.al. (2014) studied UHI and wind flow characteristics of a tropical city Muar, Malaysia which used numerical simulations IES Virtual Environment to examine the wind pattern. This research suggested that tall buildings and narrow streets are detreimental to urban ventilation. Step up configutation can distribute wind evenly and facilitate proper wind circulation. Application of numerical modeling in city planning can also help the urban planners to consider the town planning from the thermal comfort point of view.

Liu, et al. (2022) studied the planning of urban ventilation corridors in Kaifeng City, China which uses Landsat8OLI_TIRS remote sensing system along with GIS. They concluded that the existing ventilation corridor planning is the mostly researched topic but from a single scale (urban areas, blocks, buildings) and lack comprehensive consideration of multiple scales. Moreover, seasonal climate factors and variations (winter versus summer), street orientation, building layout, vegetation coverage, and water distribution are all key factors affecting the construction of ventilation corridors. Thus, in future planning and design city planners and designers should adopt a multi-standard method, comprehensively considering influential factors, and construct a reasonable and effective ventilation corridors.

Hsieh & Huang (2016) studied the method of identifying potential wind corridors for cooling and ventilation in Tainan City whose wind speed is around 3.5 m/s. They used WinPerfect software as CFD tools for simulation of wind flow in the city. They concluded that use of CFD alongwith GIS can determine least cost wind corridors in new plans and also suggest ways to develop wind corridors in already developed cities.

Tablada & He (2018) studied strategies to model city patterns for urban ventilation in high density areas of Singapore whose average wind speeds are 2.7m/s. Five strategies aiming to improve ventilation conditions at pedestrian and upper building levels in urban areas with the same GPR were tested using computational fluid dynamic (CFD) simulations. They suggested that the best improvement on urban ventilation at pedestrian and building levels is achieved by varying the breezeway widths in an alternated way. They also suggested that it is imperative that when road and landscape planning is conducted, planners should consider not only the traffic situations and landscape aesthetics, but also the airflow patterns; especially in hot and humid climates. A poorly ventilated breezeway network pattern may have a long-term negative effect since it is difficult to adjust in subsequent urban developments.

Hsieh & Wu (2012) studied climate sensitive urban design measures for improving the wind environment for pedestrians in a TOD area. They studied the importance of TOD in maintaining wind environment and air circulation in cities. This study examined the blocks around the main station of the Kaohsiung Railway Underground Project in a tropical city of Kaohsiung whose average wind speed is around 2.33m/s. They used CFD to show that the quality of the ventilation can be improved through land-use control measures. It was found that street-block-based development created a better pedestrian wind environment. Furthermore, considering the same FAR and the wind direction of prevailing winds, it was found to be possible to create a wind environment suitable for pedestrians by adjusting the building heights and layouts in the street-block-based development.

Liu, et.al. (2021) studied effective range and driving factors of the urban ventilation corridor effect on urban thermal comfort at unified scale with multisource data in Pearl River Delta, Guangzhou Province which used CFD simulation, Wind Tunnel Testing and LCP diagram algorithm. They concluded that the effective range of urban ventilation corridors on the urban surface temperature and urban comfort was ≤1000 m. The greater the distance, the smaller the mitigation effect, which was also related closely to the increasing building density and decreasing vegetation coverage within the buffer zone. Therefore, for cities with large and dense building layouts, attention should be paid to the effective range of urban ventilation corridors on the urban thermal comfort and the changing trend in building density and vegetation coverage in the buffer zone. The layout of urban buildings should be planned rationally and should utilize the environmental function of urban ventilation corridors effectively to save the amount of energy consumed by air conditioning in the summer and to create a liveable urban environment.

2.9 Methodological Review of previous research

In developing and reseraching about urban ventilation corridors, majority of the researchers utilised three different tools:

1. CFD Simulation Software
2. Wind Tunnel Experiment
3. GIS and Remote Sensing
4. Numerical Modelling and Simulation
5. LCP Algorithm
6. Research Article Review

Wong, et.al., (2010) used the concept of building frontal area index with GIS mapping to analyze and plot the least cost path.

Eldesoky, et.al. (2020) studied the use of Land based data, satellite imaging and GIS mapping system alongiwht spatial analysis to study the UHI effect. They also utilised Least Cost Path (LCP) analysis to design urban ventilation corridors.

Wicht, et.al. (2017) studied the aid of remote sensing data in detecting ventilation corridors using a spatio temporal approach in Warsaw in east-central Poland.

Wicht, et.al. (2017) studies the use of LIDAR in mapping Urban Ventilation Corridors in Warsaw, Poland.

Ren, et.al. (2018) reviewed the plans and policies developed by China Government in the design and development of wind corridors.

Shishegar (2013) studied the relationship between street canyons on air flow and local urban microclimate.

Su, et.al. (2016) used CFD along with studying the dimension of potential corridors to carry out the research.

Wang, et.al. (2015) used ENVI-met as a CFD software tool to study the effects of cool surfaces

Zhang, et.al. (2021) studied the Urban Ventilation Assessment with Local Climate Zone (LCZ) parameters by using Mathematic Model on Simulation with the Porous Media Urban Canopy Model in the Luoyang City, China.

Gu, et.al. (2020) studied the use of spatial planning for urban ventilation corridors in urban climatology in Bozhou City in

Xie, et.al. (2020) studied the use of circuit theory to simulate urban ventilation corridors in Wuhan City, China.

Chang, et.al. (2018) studied the use of CFD (WinAir) and GIS in the development of Urban Ventilation Corridors in Changchun City, China

Wang, et al. (2020) studied the use of large eddy simulations by using Parallelized LES Model (PALM) in identification of pedestrian-level ventilation corridors in downtown Beijing city, China who average wind speed was taken as 1.5m/s.

Liu, et.al. (2020) utilised GIS technologies :VPC and CSL along with remove satellite imaging to develop new set of Climatopes.

Rajagopalan, et.al. (2014) studied UHI and wind flow characteristics of a tropical city Muar, Malaysia which used numerical simulations IES Virtual Environment to examine the wind pattern.

Liu, et al. (2022) studied the planning of urban ventilation corridors in Kaifeng City, China which uses Landsat8OLI_TIRS remote sensing system along with GIS.

Hsieh & Huang (2016) used WinPerfect software as CFD tools for simulation of wind flow in the city.

Tablada & He (2018) tested using computational fluid dynamic (CFD) simulations.

Hsieh & Wu (2012) used CFD to show that the quality of the ventilation can be improved through land-use control measures.

Liu, et.al. (2021) used CFD simulation, Wind Tunnel Testing and LCP diagram algorithm

Chapter 3: Methodology

Methodology is the most important part of a research project which actually makes or breaks the completion of the project. Different research project requires different research methodology that enables the researchers to consider different variables and factors that affect the operation of the research. Methodology should be carefully developed which includes step by step process that need to be undertaken along the period of research execution for formal completion of research and achievement of favorable results. In order to reach the completion of the research, the methodologies adopted will be divided into 5 stages including analysis of literature to simulation of field variables resulting to the final recommendations for further research into the field.

There are three layers of knowledge in the field of academia, they are:

- The primary studies that researchers conduct and publish.
- The reviews of those studies that summarize and offer new interpretations built from and often extending beyond the original studies.
- The perceptions, conclusions, opinions, and interpretations that are shared informally that become part of the lore of the field.

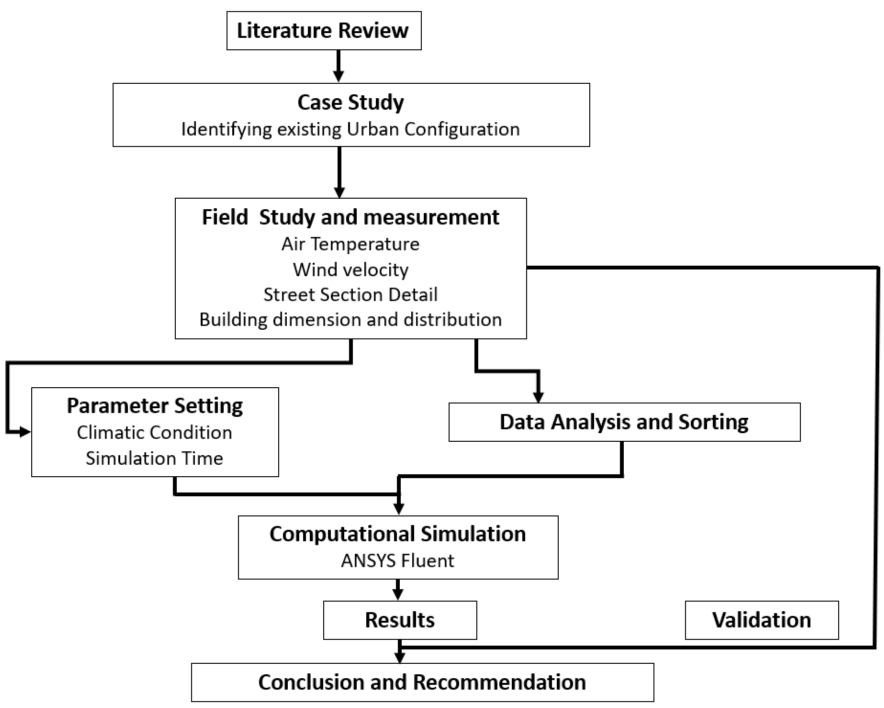


Figure 3- 1 Methodological Framework

In the first step, intensive literature review of research articles and historical data is carried out in order to get more in depth understanding of the theme of the research. In composing a literature review, it is important to note that it is often this third layer of knowledge that is cited as "true" even though it often has only a loose relationship to the primary studies and secondary literature reviews. Given this, while literature reviews are designed to provide an overview and synthesis of pertinent sources we have explored, there are several approaches to how they can be done, depending upon the type of analysis underpinning our study (Larabee, 2007)

3.1 Literature Review

The literature review is a crucial component of every research project and lays the groundwork for its effective conclusion. A literature review helps researchers comprehend the actions that must be followed to complete their research as well as the variables that must be measured in the field. The researcher can prioritize the areas of study that need to be prioritized and eliminate extraneous factors that could potentially skew the results by consulting the literature. The basis of the research issue must be understood through a chronological examination of the research literature in order for researchers to have a clear understanding of the type of research methodology that will yield the most accurate results. A literature review is an in-depth analysis of secondary data gathered by researchers from around the world, which may include factors influencing the research. The literature review has always been updated in order to help members of the academic community and the general public better comprehend the topic of the research due to the ongoing refinement and enhancement of the research process.

Different types of research require different types of literature review. Some of the types of literature review are:

3.1.1 Argumentative Review

This method selectively explores the literature to prove or disprove a claim, a fundamental presumption, or a philosophical conundrum that has already been established in the literature. The goal is to create a corpus of writing that promotes an opposing viewpoint. Argumentative approaches to literature analysis may be a valid and significant type of discourse given the value-laden character of some social scientific studies (e.g., immigration control; educational reform). However, keep in mind that when they are used to make summary statements similar to those found in systematic reviews, they might potentially pose issues with bias.

3.1.2 Integrative Review

It is thought of as a type of study that evaluates, analyzes, and integrates representative literature on a subject in order to produce fresh frameworks and viewpoints. The research that addresses similar or related theories are all included in the body of literature. In terms of clarity, rigor, and reproducibility, a well-done integrative review satisfies the same requirements as primary research.

3.1.3 Historical Review

Few things exist without reference to earlier events in history. Historical reviews are centered on assessing research through time, frequently beginning with the first time a topic, idea, theory, or phenomena appeared in the literature and then following the topic's development within a discipline's scholarly output. The goal is to put research in historical perspective in order to demonstrate knowledge with cutting-edge advancements and to determine the most likely pathways for future study.

3.1.4 Methodological Review

The technique of analysis used in a review is often more important than the speaker's actual words (content). This method allows researchers to draw from a wide variety of knowledge ranging from the conceptual level to practical documents for use in fieldwork in the areas of ontological and epistemological consideration, quantitative and qualitative integration, sampling, interviewing, data collection, and data analysis techniques. It also provides a framework of understanding at different levels (i.e. those of theory, substantive fields, research approaches, and data collection and analysis techniques).

3.1.5 Systematic Review

Using pre-specified and defined methodologies, this form provides a summary of the available research that is relevant to a clearly stated research topic. It also collects, reports, and analyzes data from the studies that are part of the review. Usually, it focuses on a fairly narrow empirical inquiry, which is frequently phrased in cause-and-effect form, such as "To what degree does A contribute to B?"

3.1.6 Theoretical Review

This form's goal is to specifically investigate the body of thought that has collected in relation to a problem, idea, theory, or phenomena. The theoretical literature review aids in identifying what ideas currently exist, their connections, the depth to which they have been explored, and the creation of new testable hypotheses. This format is frequently employed to demonstrate the absence of suitable theories or the inadequacy of present theories to account for novel or developing research issues. The theoretical notion, a full theory, or a framework may be the subject of the unit of analysis (Larabee, 2007).

3.1.7 Meta-analysis

Combines information from other separate research that deal with the same issue. need randomized controlled studies to provide scientific support. Groups of researchers do meta-analyses since there are so many articles to examine. Meta-analysis, which is utilized in the arts, sciences, and health, is helpful for giving a better approximation of the impact or success of an intervention.

3.1.8 Iterative

A somewhat thorough, algorithm-based literature review that compiles all publications in a field of study is known as an iterative review. After removing titles that are not essential, a human approach is utilized to determine their relevance to the study field. The remaining studies and papers are categorized, allowing iterative coding to be performed to the remaining data arrays to display pertinent and interesting data throughout an area of study.

3.1.9 Meta-synthesis

Examines, interprets and integrates findings of several qualitative studies using qualitative methods. Meta-synthesis is used in research looking at theory development. Meta-synthesis is used for clarifying concepts and patterns, and refining existing models and theories.

3.1.10 Rapid

Rapid reviews are undertaken to help support time-sensitive decision making. Standard systematic review procedures are adapted by removing or modifying some steps. These reviews are undertaken to quickly find information on a topic to support a project or decision making. Rapid reviews are useful in delivering answers in a shortened time frame.

3.1.11 Umbrella

Umbrella reviews bring together reviews that answer different questions which all relate to a shared topic. These reviews find, contrast and synthesise the findings from other systematic-style reviews. Umbrella reviews are developed to giver researchers and decision makers a clear understanding of a broad topic area in a shortened time frame (University, 2022).

3.2 Field Visit and Data measurements

After the completion of intensive literature review, the next step we undertake is the site visit where we select the most feasible site upon which we perform our research experiments. In the context of this thesis research, the situation demands a settlement with proper street allocation in and enough space between buildings for people to traverse. A well-planned city with proper spaced buildings is the main focus of the research thus we selected the Lumbini Masterplan Site. Here the buildings are well spaced and proper street distribution network is available. After analysis of all the variables that need to me measured to initiate further steps in the research from the Literature Review, we then utilize instruments or oral survey to obtain the necessary data from the site.

In terms of simulation in CFD, we require the following variables:

1. **3D model of the residential settlement plan**
   Here we need the dimension of the streets, the space between the houses, no of houses in a row and column in the direction of win and away from the wind direction. We also need to measure the dimension of the buildings in the area in order to model in accurately in the CFD tool. If the number of houses is not enough

in a row or column, we then add similar sized buildings in rows and columns according to the need of the simulation range.

2. **Temperature of the street**

   Since, UHI effect is basically dealing with the rise in temperature of the area due to the storage of heat by the building and pavements during the day, we need to measure the temperature of the region during the day and night. We also need to measure the temperature of the area before, during and after the wind blows through it to validate our simulation at the end of the research. This validity is an important part of research as simulation alone is not taken as the only answer to the problem. So, we use HOBO MX2300 series data logger to measure the temperature and humidity of the region for about 7 days continuously.

3. **Measurement of wind speed**

   For the simulation, wind speed is one of the most important variables in CFD. We input the wind speed and direction in the CFD tool to measure and monitor the flow of wind through the settlement. The direction of flow of wind as well as its speed is quite important in measuring the behavior of wind flow and the areas around the building it covers while circulating. This helps us to understand the best orientation and design of buildings to be made for best circulation.

3.3 Data sorting and analysis

After the field visit and data collection, we then sort the necessary data and the discard the unnecessary ones to easy the analysis. Then, from the data measured by the instruments and obtained from DOHM, Nepal, we then draw charts and graphs for temperature curve and wind curve. In the temperature curve we plot the time of day and temperature range where we also mark the hours where wind was blowing in the area. We also plot the wind rose diagrams to show the direction and magnitude of the wind flowing in the site. This helps us to validate our theory after the simulation is completed.

3.4 Modelling and Simulation

After the collection and sorting of data, we then start the process of simulation. Firstly, we model the buildings and the required area of settlement in the CFD tool and enter the required climate data for simulation. We then begin the simulation and run it couple of times for accuracy. After the completion of simulation, we obtain the results and then we recheck the input variables and boundary conditions. A final run of the simulation takes place and the result is collected. The software to be used are ANSYS Fluent as CFD Tool and Climate Consultant to develop wind charts & wind rose diagrams.

3.4.1 ANSYS Simulation

3.4.1.1 ANSYS Fluent Graphical User Interface (GUI)

Ansys fluent primarily presents itself as a combined interface called the Workbench where multiple steps of the simulation can be individually accessed after the completion of the previous steps.

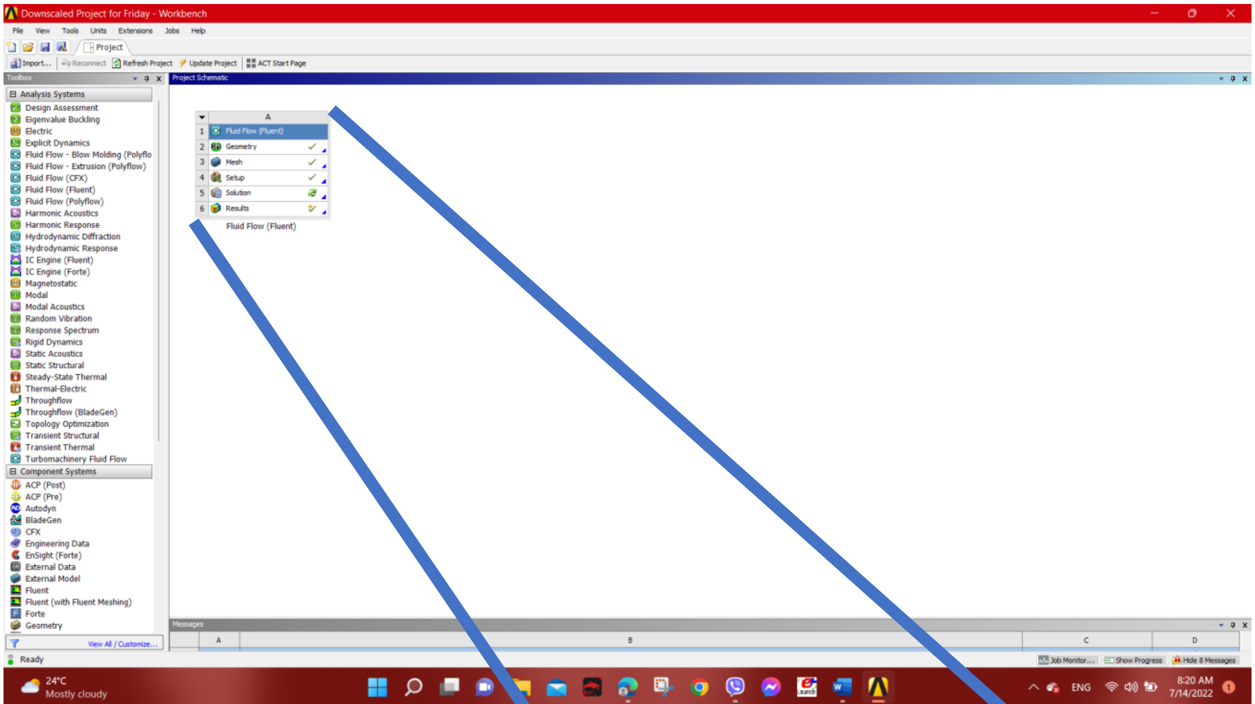
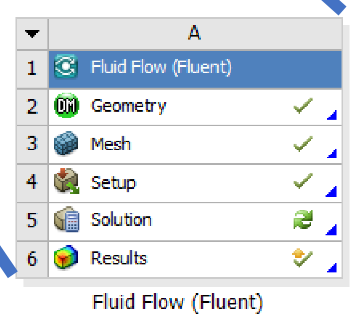


Figure 3- 2 ANSYS User Interface



3.4.1.2 ANSYS Settings

3.4.1.2.1 Geometry

Here, we model and draw the required objects for simulation including its fluid flow domain. For this thesis, a section of the site is selected as a sample simulation model to understand the basics of the software and its results. The site was selected to make sure that the buildings were oriented towards the wind flow direction with no prior blocks ahead. The building dimensions were used according to the field measurements.

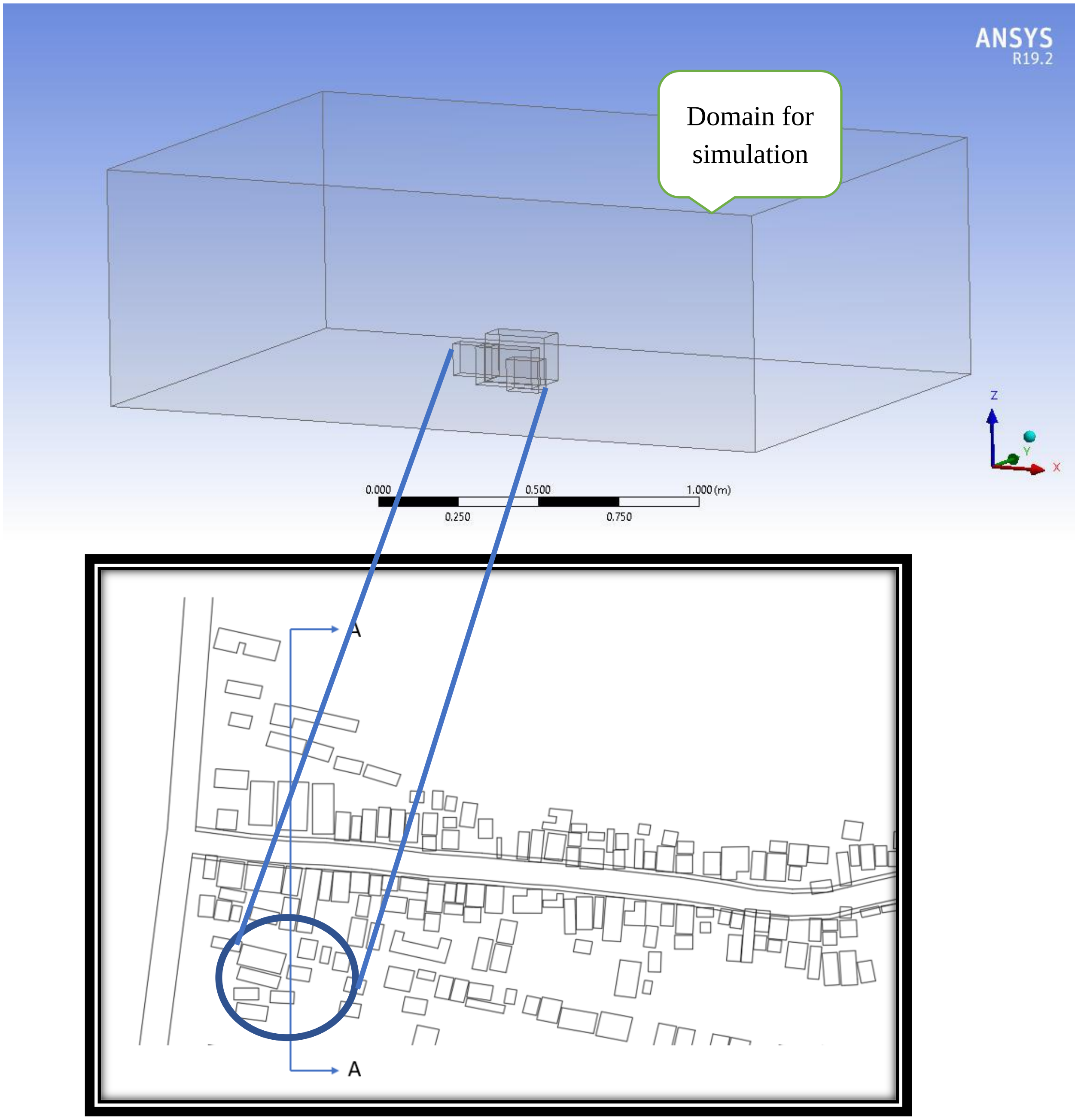


Figure 3- 3 Site Section for Sample Simulation

The domain was created using the ANSYS literature where the offsets from the extreme building surfaces are to be kept around 5H (H= height of the tallest building) to the left, right and to the inlet while 15H to the outlet with height also equals to 5H. The buildings were downscaled and few were only selected to reduce the computation time and increase the accuracy of the fluent solver as big buildings cannot be simulated in a basic laptop and require supercomputers for solving.

3.4.1.2.2 Meshing

This is the second and the most important aspect of CFD simulation where the geometry and domain are divided into meshes for solution. Mesh is basically small net like elements which are used for finite elements analysis. The main concept of meshing is to reduce the computation area into smaller sections to reduce computation effort and increase accuracy with the development of nodes.

Details of "Mesh"	
Display	
Display Style	Use Geometry Setting
Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Element Order	Linear
Element Size	1.e-002 m
Export Format	Standard
Export Preview Surface Mesh	No
Sizing	
Use Adaptive Sizing	No
Growth Rate	Default (1.2)
Max Size	Default (2.e-002 m)
Mesh Defeaturing	Yes
Defeature Size	Default (5.e-005 m)
Capture Curvature	Yes
Curvature Min Size	Default (1.e-004 m)
Curvature Normal Angle	Default (18.0°)
Capture Proximity	No
Bounding Box Diagonal	2.8062 m
Average Surface Area	0.48447 m^2
Minimum Edge Length	4.879e-002 m

Figure 3- 4 Mesh Setting in ANSYS Fluent

The mesh was generated in a default manner as much as possible due to the generic shape of the model and the domain (Cubical). The Element size was kept at 1 mm with a growth rate of 1.2.

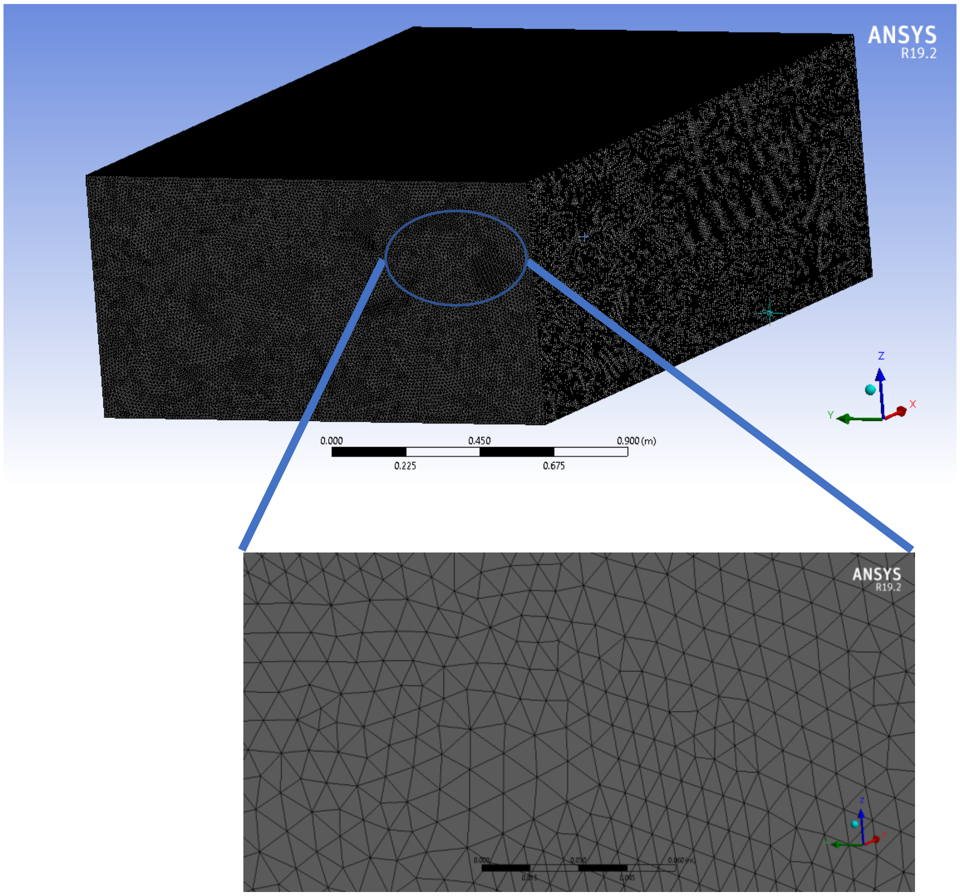


Figure 3- 5 Mesh and Enlarged Mesh units

3.4.1.2.3 Setup and Solution

Here we provide the boundary condition for the simulation which includes the material of fluid and solid objects which was kept as air and aluminum as defaults. The k-ε solver with Pressure-Velocity coupler was used for solving the steady state problem. The inlet was kept as velocity inlet while the outlet was provided as pressure-outlet for convection flow. The boundary of the domain and the ground layer was provided as no slip walls along with top layer as dynamic layer. Radiation loading was turned on for determining wind and temperature relation for the simulation. The number of iterations was kept at 500 for accurate results.

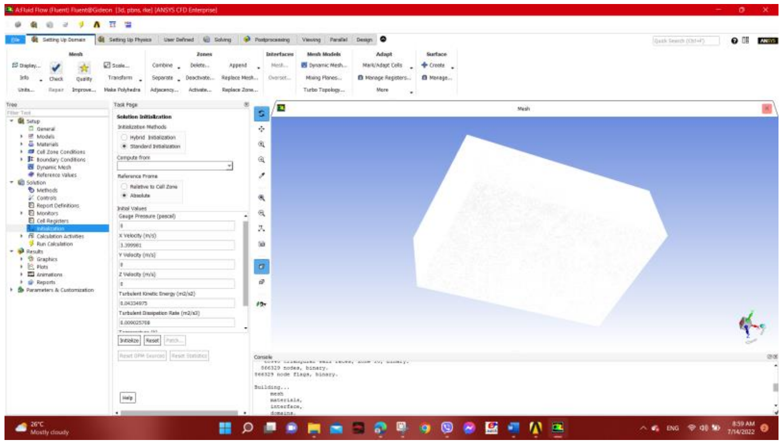


Figure 3- 6 Setup and Solution GUI

3.4.1.2.4 Post Processing

After the solution undergoes 500 iterations, the results are then obtained in the form of charts and animations. The different types of charts can be obtained in the form of contours, streamlines, vectors, etc. Here, we can also create an animation of the simulation for ease of understanding and presentation purpose if necessary.

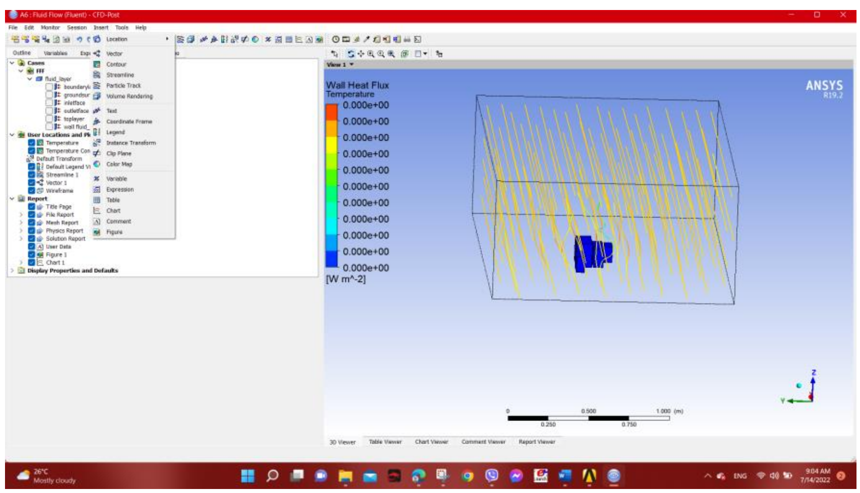


Figure 3- 7 Post Processing

3.5 Analysis and Conclusion

After the completion of simulation and obtaining the result, we then compare the results of simulation with the data obtained from the field. If our data validates the simulation then we are successful in proving our theory else we recheck the simulation and being modelling again. After validation of our results by field data, we then create a favorable conclusion to our research and begin the reporting stage.

3.6 Reporting

After the analysis of results and developing a conclusion, formal inclusion and completion of report is executed.

Chapter 4: Project Area

The site selected for the research is in the Lumbini Province, Rupandehi District of Nepal which lies in the Hot and Humid Climatic region of Nepal. The research will be based upon a section of settlement near the Lumbini Masterplan Area where a row of 4 houses will be selected in the direction of the wind as shown in the figure. In the case of lack of buildings to fulfil the selection criteria, hypothetical buildings will be modelled in the CFD simulation tool.

Figure 4- 1 Case Study Site at Lumbini (Google Earth)

4.1 Data Source

The weather information for Lumbini was obtained from **Meteonorm Software,** an expensive tool that is used to download meteorological data and irradiation data files of places using their GPS coordinates. Various climate aspects are shown, understood, and studied using Climate Consultant software. The Climate Consultant program uses the yearly TMY 8760-hour EPW format climate data to produce charts and graphs for the climatic research.

4.2 Climate Summary

Elevation: 150m (490 ft)

Co-ordinates: 27°28′53″N 83°16′33″E

The warmest time of year is generally early June where highs are regularly around 104.2°F (40.1°C) with temperatures rarely dropping below 80.2°F (26.8°C) at night. Lumbini has some extremely humid months, and slightly dry months in the opposite season. The least humid month is April (36.4% relative humidity), and the most humid month is January (81.2%).

Wind in Lumbini is usually calm. The windiest month is May, followed by June and July. May's average wind speed of around 6.4 knots (7.3 MPH or 11.8 KPH) is considered "a light breeze." Maximum sustained winds (the highest speed for the day lasting more than a few moments) are at their highest in late April where average top sustained speeds reach 15 knots, which is considered a moderate breeze (ChampionTraveler, n.d).

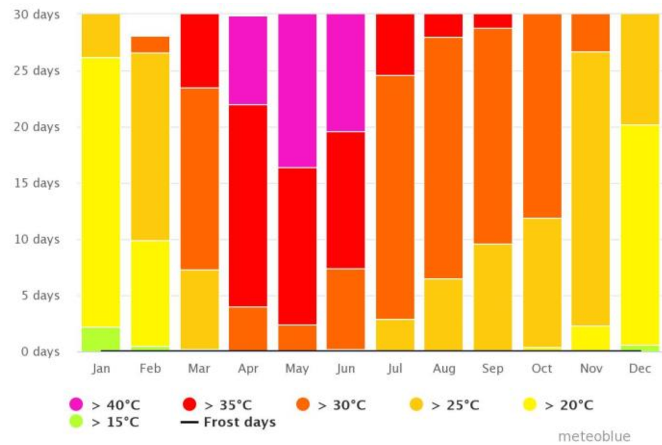


Figure 4- 2 Monthly temperature of Lumbini (Meteoblue)

According to figure 4.2, we can see that from the months of April to June, the temperature in the region exceeds 40℃ while it remains around 20℃ even during winter months of December, January and February. This shows that Lumbini is a predominantly tropical climate region with excessive heat during the summer which definitely is enhanced by the presence of UHI effect in the cities. So, development and design of proper wind corridors is imperative.

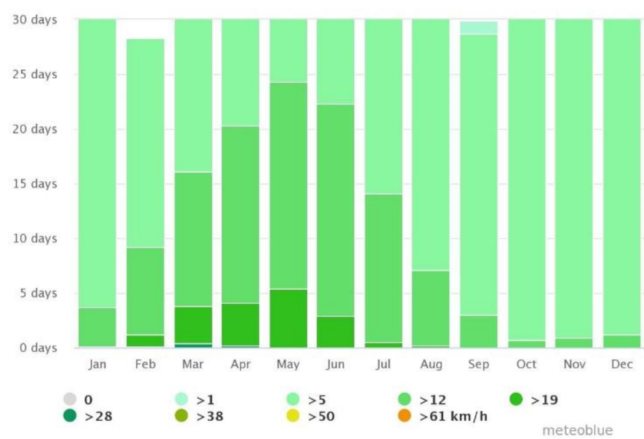


Figure 4- 3 Monthly Wind Speed of Lumbini (Meteoblue)

From figure 4.3, we can conclude that the wind speed in the site is around 5kmph and rarely exceed 20 kmph. So, there is a big potential of reducing UHI effect in the region by using the wind flow and air circulation properly. Thus, this natural phenomenon becomes a gift to the people of the region who have to face the hardships of excess temperature each summer.

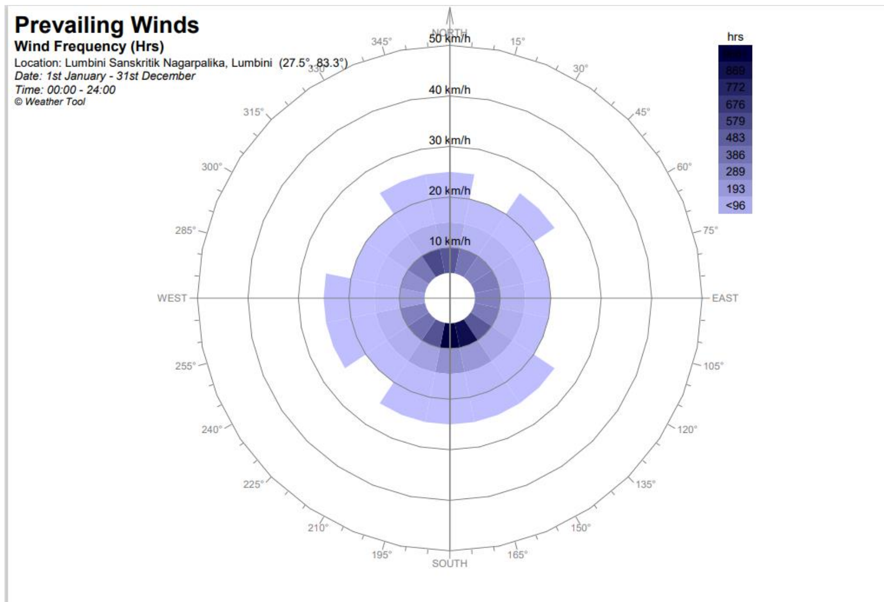


Figure 4- 4 Monthly Windrose Diagram of Lumbini (Ecotect)

According to the wind rose diagram sourced from the Autodesk Ecotect 2011 for the region of Lumbini we can see that the direction of wind flow is primarily in the N direction and least in the South. So, developing wind corridors orientation towards the north or in the nearby direction is beneficial for the optimum use of the wind circulation within the settlements.

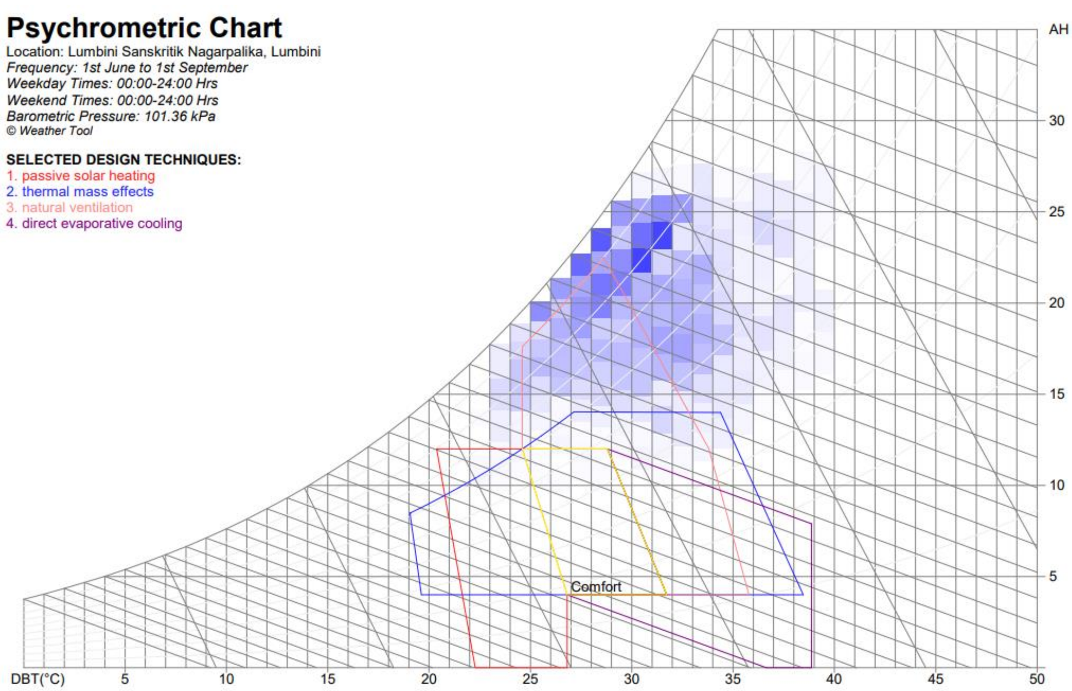


Figure 4- 5 Psychrometric Chart during Summer (Ecotect)

From figure 4.5, we can observe the psychrometric chart for the site during the summer where majority of the time, passive and active cooling systems are to be used to maintain thermal comfort as the average temperature reaches mor than 35℃ during the day which is more than the thermal comfort region for a human (26 ℃) whereas from figure 4.6, we can see that during the winter, the temperature normally falls in the average human comfort zone with slight focus towards passive solar heating can be done for days where the temperature falls below the thermal comfort zone (18℃).

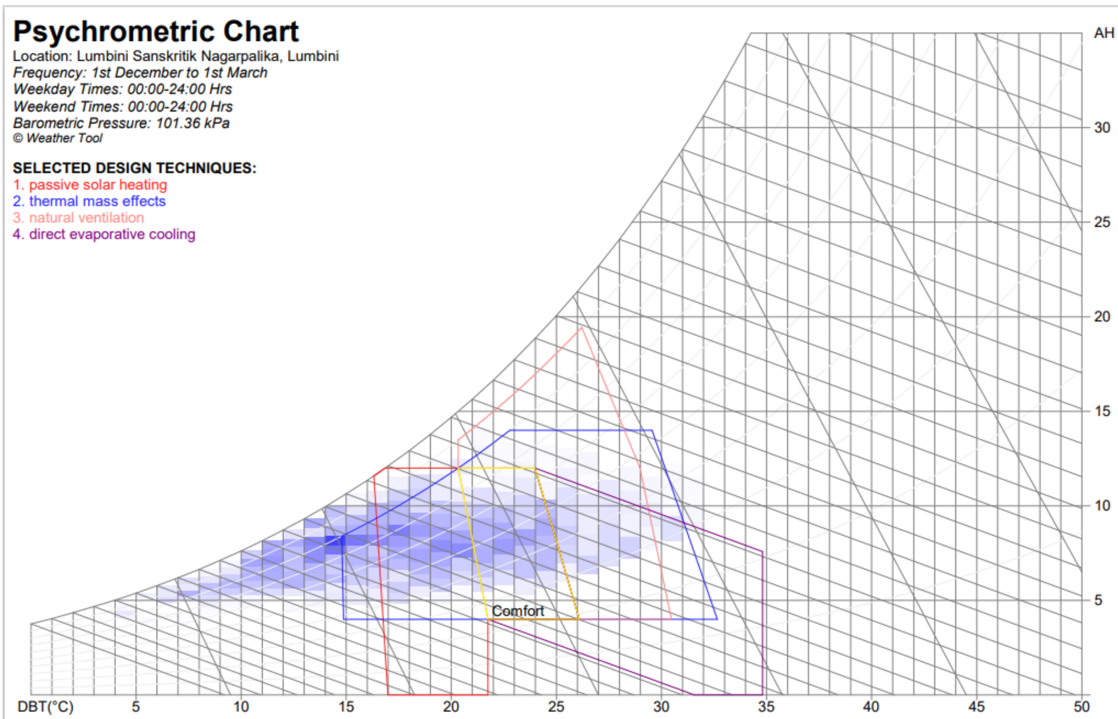


Figure 4- 6 Psychrometric Chart during Winter (Ecotect)

4.3 Field Visit and measurements

For the next stage of the report completion, a thorough field visit was carried out during the month of June 16th to 25th in the road section along the Lumbini Sanskritic Municipality beside the Lumbini Gate no 5. The field measurement primarily consisted of measurement of road section (length, width and topography) with the help of laser measure (NOYAFA NF 271) and a standard 50 m measuring tape (Fibreglass Measuring Tape). The temperature and humidity were measured by a standard HOBO data external datalogger (Onset MX2302A HOBO Data logger) which was calibrated from the Department of Hydrology and Meteorology. The data logger was placed in the premise of a grocery shop due to safety and security region at a height of around 3m with the sensor hanging around 15 cm below the device in the air for measurement of the required atmospheric data. The datalogger was placed for 15 days starting from June 17 to July 1.

The air velocity was measured in a stretch of about 100m each along the road section in 3 points covering the required site section. The air velocity was measured by a standard anemometer (ERICKHILL HT625C) for about 7 days facing the west where the street section began and to the North as well.

Figure 4- 7 Wind Velocity Measurement (PC: Skanda Rimal)

4.3.1 Street View and Section

The location is on the eastern edge of the sacred garden area. The roadway runs from Gate No. 8 to Mahilwar Chowk and goes to Lumbini's villages. Lumbini Sanskritik Municipality was founded by merging the existing Lumbini Adarsha Village Development Committee with six additional Village Development Committees, namely Bhagawanpur, Tenahawa, Ekala, Khudabazar, Madhuwani, and Masina.

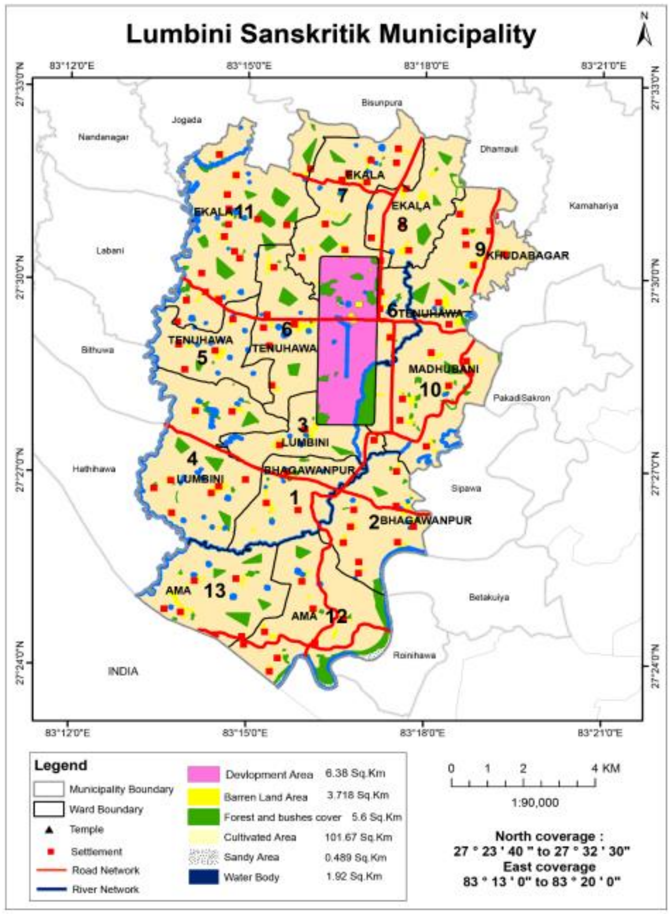


Figure 4- 8 Lumbini Sanskritik Municipality Map (Source: GIS Map)

Lumbini Sanskritik Municipality is one of the municipalities that is predicted to grow fast in the future due to the development potential of housing the world heritage site of Buddha's birthplace. The street near the Lumbini masterplan is made up of hotels and lodges with very few residences. The street width is 45', with a 6.5' pedestrian sidewalk, while the surrounding buildings have 1 to 5 storeys. Because the average height of a one-story structure is 10', the maximum height of a building on the street is 50'.

Figure 4- 9 Street view towards east (PC: Biplav Pokhrel)

Figure 4- 11 Street view towards south (PC: Apekshya Ghimire)

Figure 4- 10 Street view towards north (PC: Apekshya Ghimire)

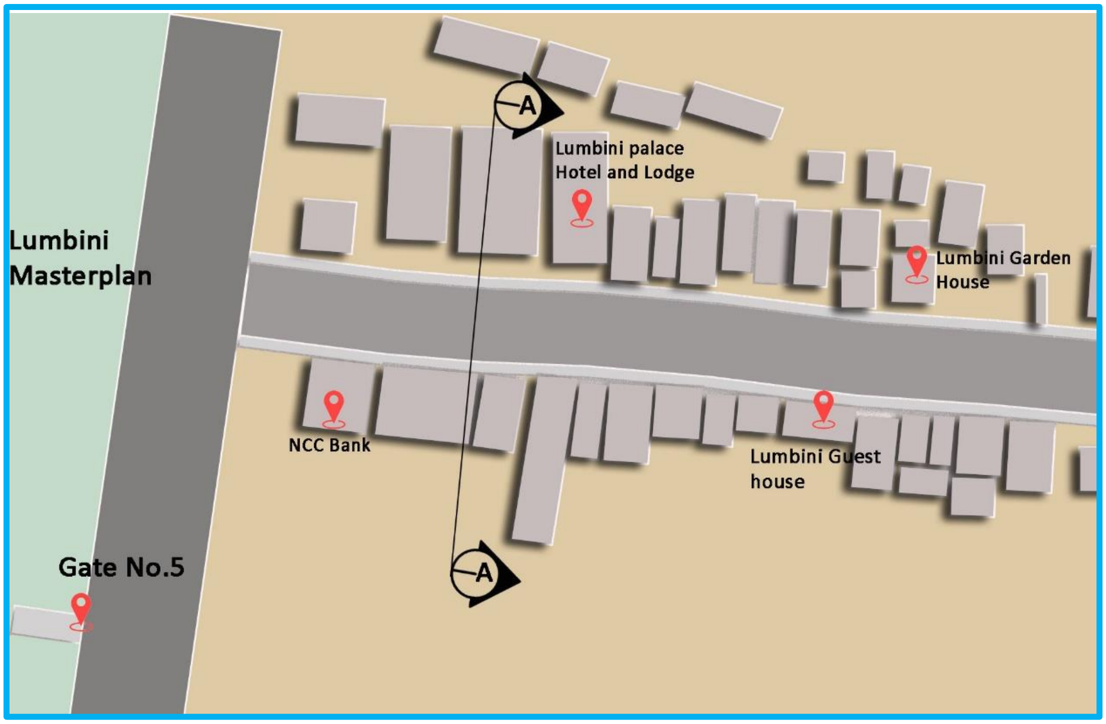


Figure 4- 12 Street map

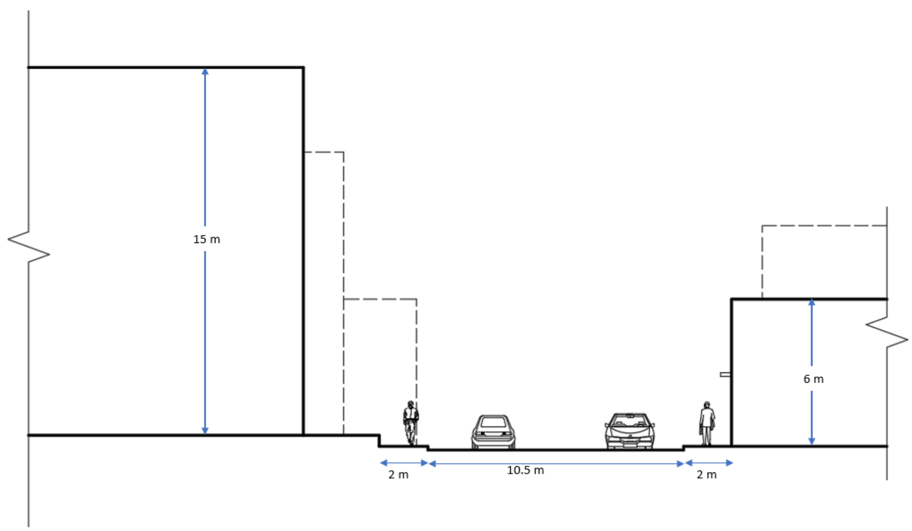


Figure 4- 13 Section at A-A

Chapter 5: Findings and Analysis

As per the site survey carried out at Lumbini Sanskritik Municipality Road section from Mahilwar to Majhediya road section, it was found that 100% of the interviewees preferred more wind flow in the area than the present scenario. They believed that increased wind flow would significantly make the road section more comfortable and pleasant for mobility and interaction which is the major objective of this research project.

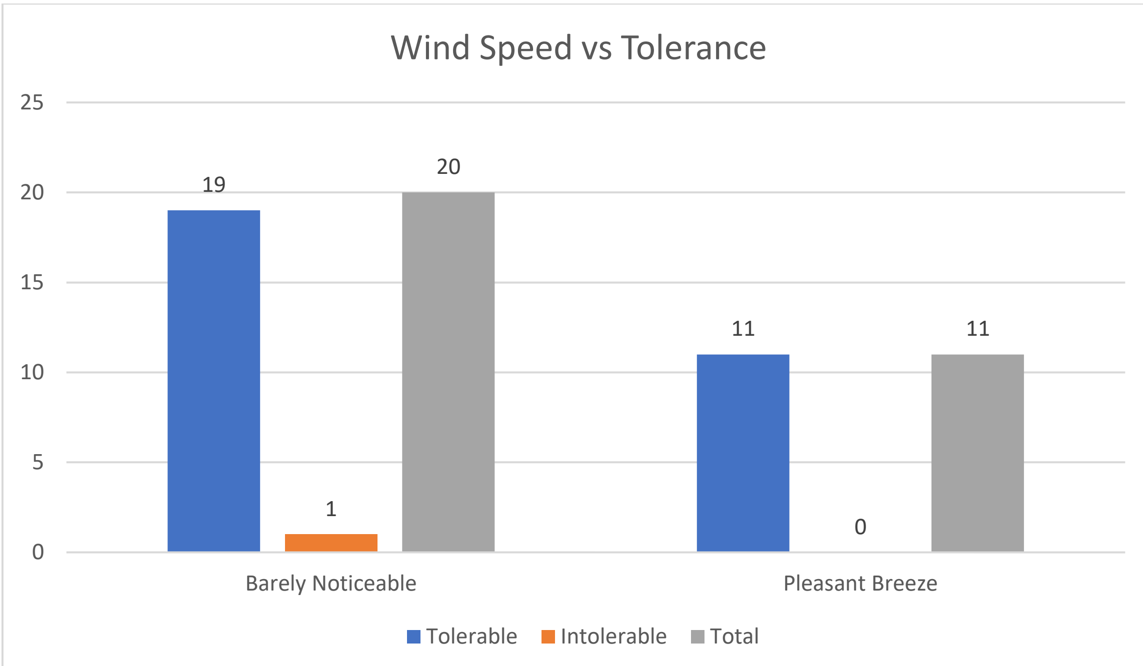


Chart 5- 1 Thermal comfort survey result for wind flow

From chart 5.1 we can see that total people surveyed was 31 in the road section in multiple days and different points of time. 20 people were surveyed when the breeze was barely noticeable where 19 people accepted the condition as tolerable while single person found it intolerable. Next, 11 people were surveyed when the breeze was pleasant and 11 of them found the condition perfectly tolerable which only proves that increased wind flow along the region increased thermal comfort for the people living there.

5.1 Findings from ANSYS Simulation

5.1.1 Sample Simulation

For the completion of sample simulation, multiple results were obtained to understand the basic condition of the part of the site in the real time scenario as a practice run for simulation purpose. Two different charts were obtained for analysis.

They are:

1. Pressure contours on the buildings due to wind flow
2. Streamlined movement of the wind particles along the buildings

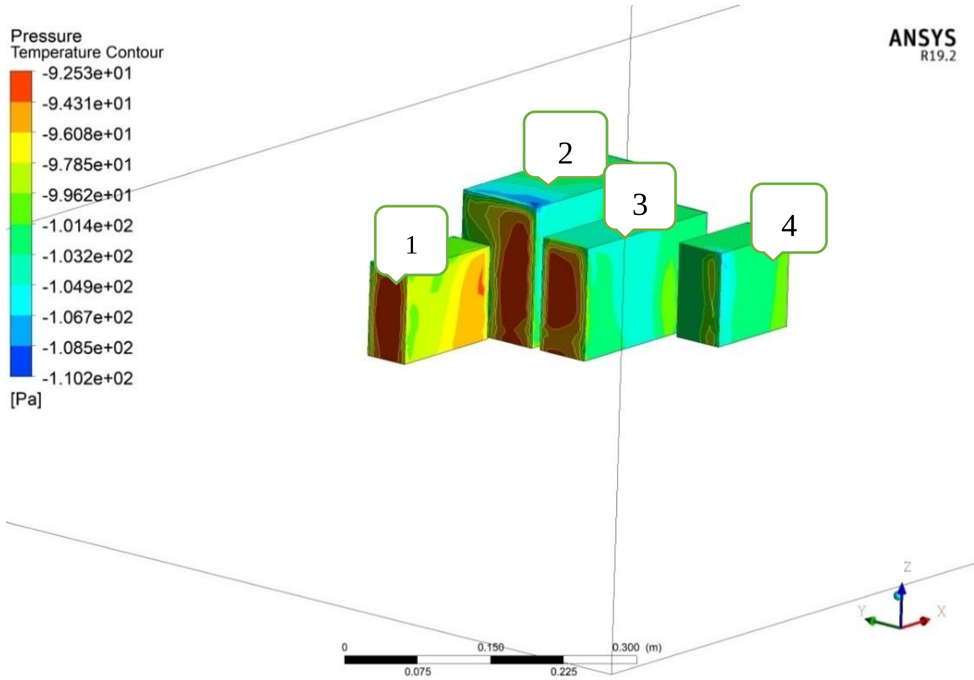


Figure 5- 1 Pressure Temperature Contours on the buildings

From figure 5.1, we can see that maximum pressure is obtained in the wind ward face of the buildings directly in contact with the wind (Buildings 1, 2 and 3) while the building shaded receives very little wind pressure as the buildings in front act as a wind barrier. The top surface receives less pressure due to tangential contact with the wind while the perpendicular faces are under maximum pressure due to direct contact.

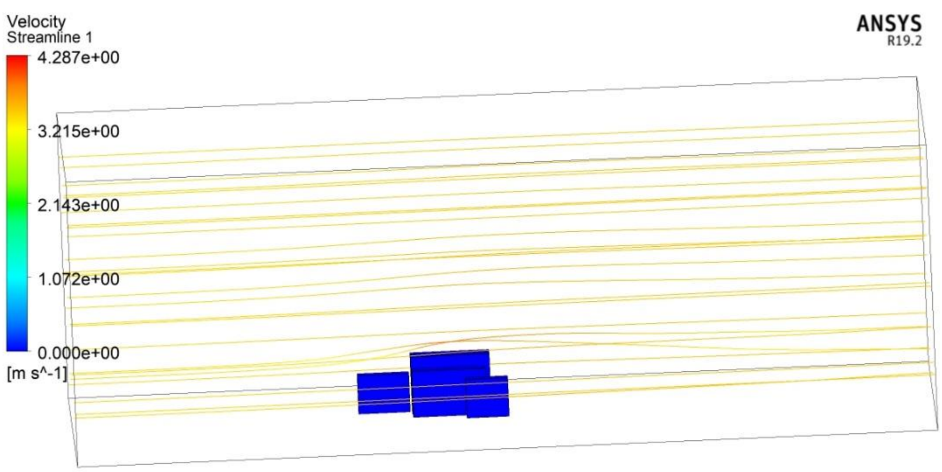


Figure 5- 2 Velocity Streamlines of Wind

From figure 5.2, we can see the movement of wind along the domain and interacting with the objects (buildings) in contact. We can see the lines moving through as well as curving over the buildings as a normal fluid would in contact with a porous element.

5.1.2 Simulation of the Site

5.1.2.1 Site Selection and division

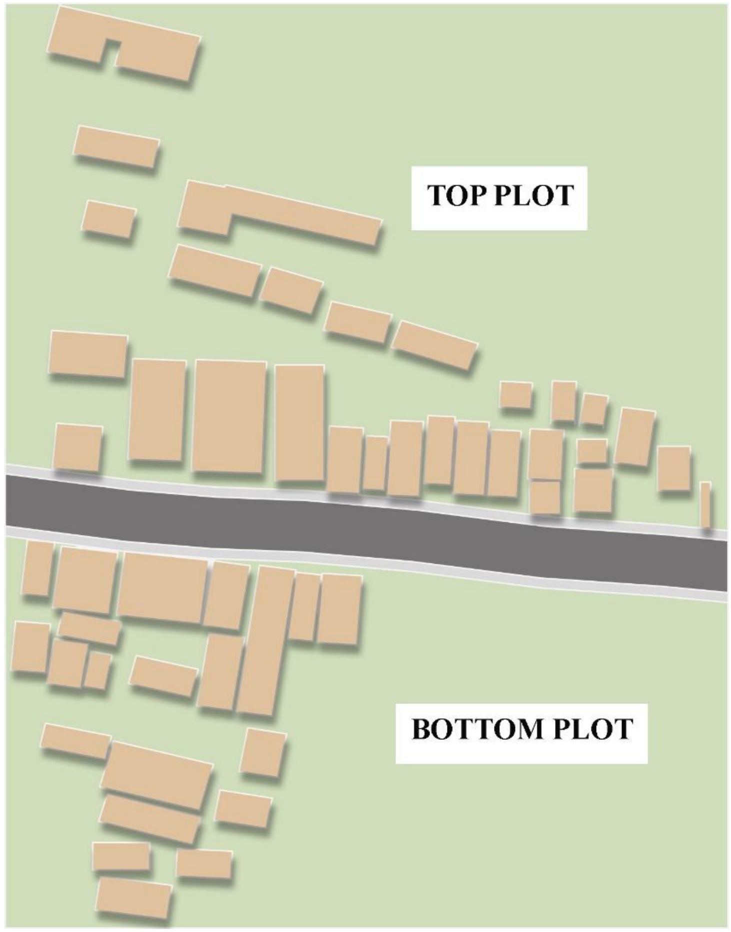


Figure 5- 3 Site selection and division for simulation

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 as follows:

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 simulation was done as a 2D format to increase computation efficiency and reduce computation time. 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 steps that were followed were same as that of the sample simulation as shown in 5.2.2 with different entry parameters in setup section of the simulation. The buildings were measured in AutoCAD along with their spacing and orientation and then modelled accordingly in Solidworks. 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. The height of the building was not utilized in modeling as it does match with the research objective and is irrelevant to the result that we intended to obtain in the end.

While the heigh to the building may result in draft movement of the wind which increases the wind speed in the pedestrian level, the buildings weren't that high enough to be considered as the maximum height was measured to be around 15m from the GL of a single building in that vicinity while majority of the building were of around 10m in height. Since, this research primarily focuses 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. In the presence of higher computing power, the whole site could be selected for simulation thus giving higher factor of accuracy but as a limitation, limited computing power has enabled the researcher to selected such plots.

5.1.2.2 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 (Liu, et.al., 2017). The final sketch was then exported as IGIS (.IGS) format that is readable by ANSYS Fluent Geometry interface.

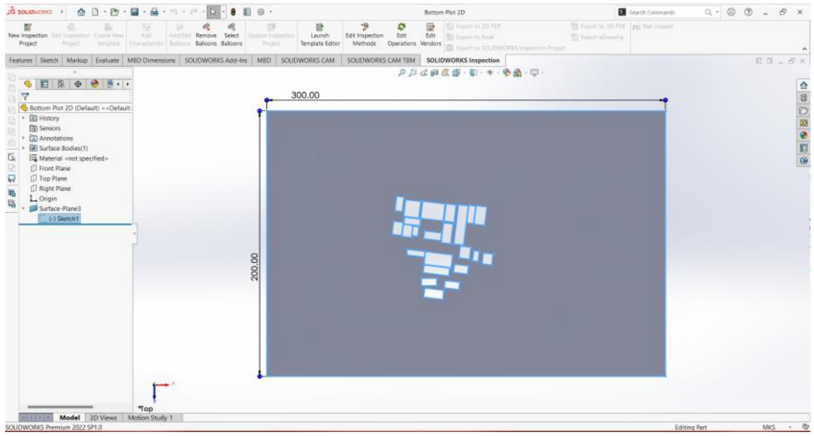


Figure 5- 4 Solidworks Interface for modelling

5.1.2.3 Simulation in ANSYS Fluent

Similar to the sample simulation, the meshing and setup process were carried out with different parameters as shown below:

Table 5- 1 Simulation Parameters in ANSYS Fluent

Model	Inviscid
Fluid Type	Air
Solid Type	Aluminum
Geometry Named Selection of Domain Sides	
North	Velocity Inlet (6m/s)
West	Velocity Inlet (4m/s)
South	Pressure Outlet (1 atm)
East	Pressure Outlet (1 atm)
Mesh	
Meshing Element Size	1m
Refinement	1 step
Solution Setup Parameters	
Streamline Points	10,000 nos
Contour lines	1000 nos
Vector factor	1
No of iterations of calculations	500

The parameters were set according to the requirement of the simulation and modelling. The model of fluid flow was set to inviscid. Viscosity's impact on the flow is ignored in inviscid flow analyses, which are suitable for high Reynolds-number applications because inertial forces typically outweigh viscous forces. An aerodynamic study of a high-speed projectile is a good illustration of a situation where an inviscid flow calculation is applicable. In this scenario, the body's pressure forces will outweigh its viscous forces. An inviscid analysis will thus provide you with a fast estimation of the main forces affecting the body. You may do a viscous analysis to take into account the impact of the fluid viscosity and turbulent viscosity on the lift and drag forces after the body shape has been altered to optimize the lift forces and minimize the drag forces.

For issues requiring complex flow physics and/or complex flow geometry, a suitable initial solution may often be found using inviscid flow analysis. In this situation, the viscous forces are significant, but the viscous elements in the momentum equations will be disregarded in the early phases of the computation. Once the computation has begun and the residuals are getting smaller, you may enable turbulent or laminar flow to activate the viscous terms and continue the solution to convergence. This is the only method available for some really complex flows to begin the computation (ANSYS, 2009).

The fluid was kept as air because we are dealing with wind flow while the buildings were supposed as aluminum to ignore the friction loss in the building surface. The velocity inlet is the region of domain where the wind flow is provided while pressure outlet is the region of domain where the wind exits due to pressure gradient. This difference in pressure gradient causes the flow of wind in the region because the force was natural and not forced. Meshing size was kept at 1m to increase computation accuracy and reducing computation time. Even though smaller mesh sizes would mean greater accuracy, the computation memory was not enough to reduce the size anymore and 1m was selected as the optimum size.

The mesh refinement was preferred to reduce the mesh size by half to 0.5m thus increasing the mesh arrangement on the object surface to increase accuracy. The no of contours was kept at 1000 for easy analysis while the no of streamline points was kept at 10,000 to increase the accuracy of wind flow analysis. The vector factor was kept at 1 while the no of iteration for calculation was kept at 500 to increase the chances of convergence of variables calculated (Velocity x, y and z).

5.1.3 Results and Findings of Site Simulation

For the final results of site simulation in ANSYS Fluent, there are mainly 4 results and their charts that need to be developed and analyzed in ANSYS. The charts are Pressure Contour, Velocity Contour, Velocity Streamline and Velocity Vector. 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.

a. Pressure Contour

The pressure contour is a chart that determines the pressure exerted in multiple part of the domain including the building surfaces by the wind flow from the desired direction in setup. This helps us to understand the wind pressure present in the required area of analysis which provides us with the knowledge whether the pressure is appropriate in removal of humidity, dusts and other pollutants if required. This also enables us to observe the aerodynamic loading in buildings especially in high rise buildings during structural analysis phase of building construction. In this research, the wind pressure factor is included to determine whether the wind flow is appropriately present in between and around the buildings. Lower wind pressure means low presence of wind flow in the area which is detrimental to the thermal comfort of that region.

i) Top Plot

As we can see from figure 5.5, 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 pressure ranges from 265 Pascal high to as low as -462 Pascal which is not what we desire. We desire uniform pressure distribution in majority of the region for proper wind circulation.

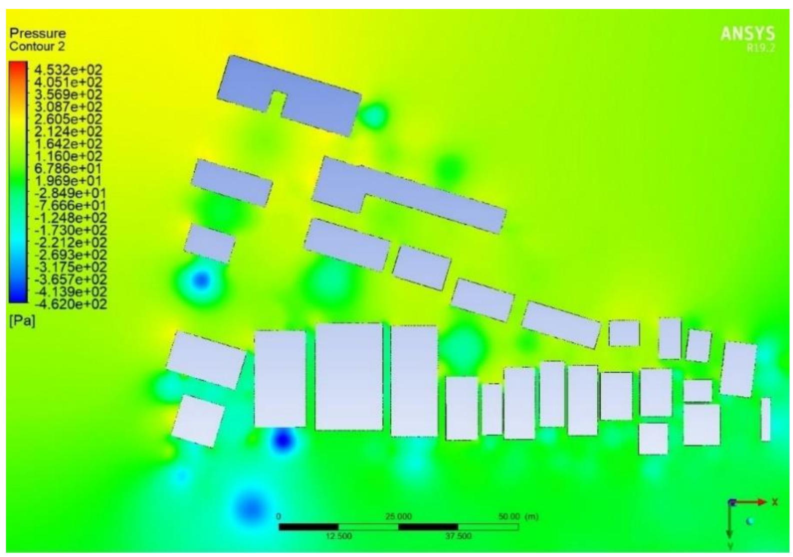


Figure 5- 5 Pressure contour of the Top Plot in ANSYS Fluent

ii) Bottom Plot

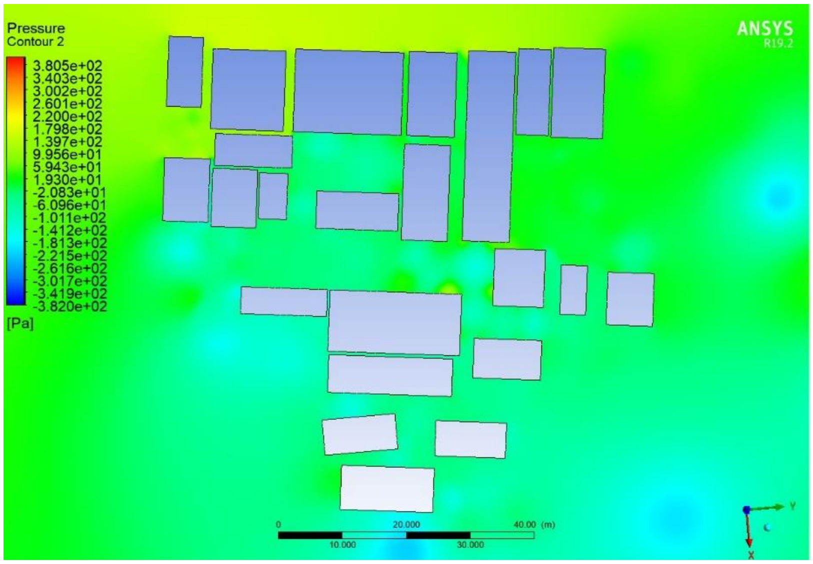


Figure 5- 6 Pressure contour of Bottom Plot in ANSYS Fluent

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 Pa high to as low as -18 Pa 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 5.5 and 5.6. Even though this much pressure difference isn't detrimental to the thermal comfort of the overall macroclimate, microclimate analyzing, this might create some stagnation points around the buildings which causes discomfort to the people.

b. Velocity Contour

i) Top Plot

As seen from figure 5.7, 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 (0.01895m/s) which is quite detrimental to the thermal comfort for the pedestrians and people with occasional velocity reaching 2m/s.

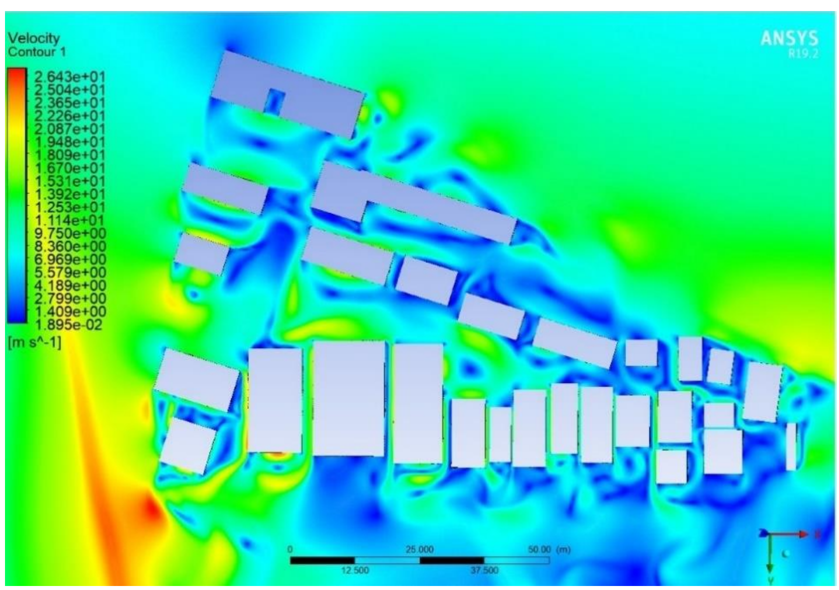


Figure 5- 7 Velocity of Top Plot in ANSYS Fluent

This low velocity of air flow around the buildings suggests rise in temperature that in the areas with considerably higher wind flow velocity.

ii) Bottom Plot

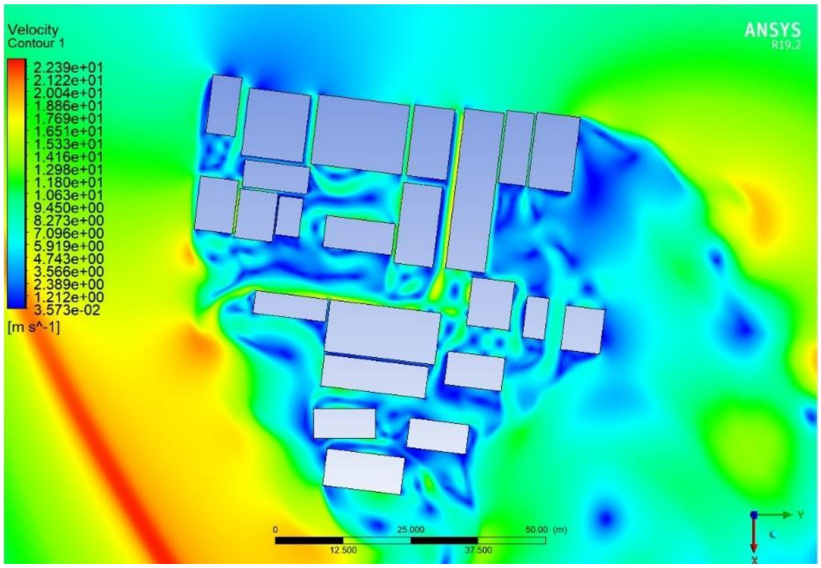


Figure 5- 8 Velocity contour of Bottom Plot in ANSYS Fluent

From figure 5.8, we can observe that 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 0.03573m/s with occasional speed of 1.65m/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 which can be observed clearly from the high thermal discomfort to the pedestrians in the streets during the day.

c. Velocity Streamline

i) Top Plot

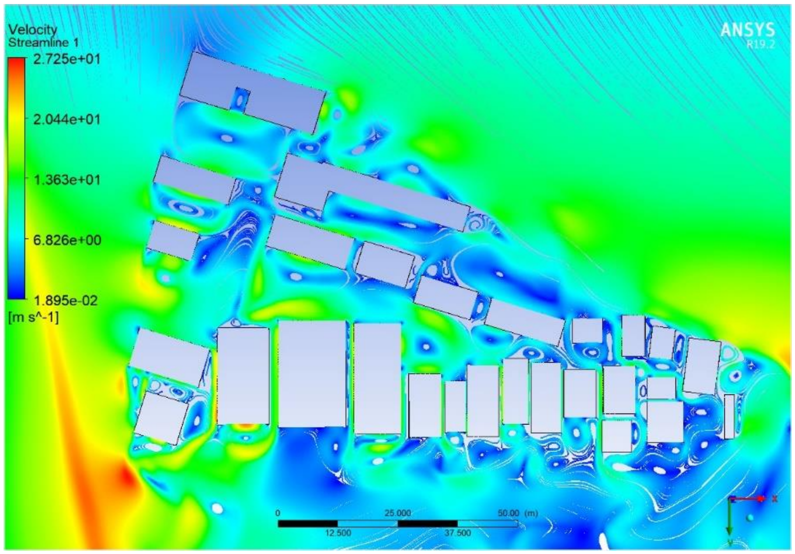


Figure 5- 9 Velocity Streamline in ANSYS Fluent

From figure 5.9, we can observe the wind flow routes in the region and around the buildings. As we can see, the flow isn't 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). This results in the circulation of the same air in the region which is high in pollutants and CO_2 which is expelled by the users in the area. The consumption of this recirculated air results in multiple diseases as well as very high thermal discomfort due to rise in embodied energy and thus the temperature of the air pockets. We must avoid development of such air pockets around the buildings to maintain air quality in the outdoor regions.

ii) Bottom Plot

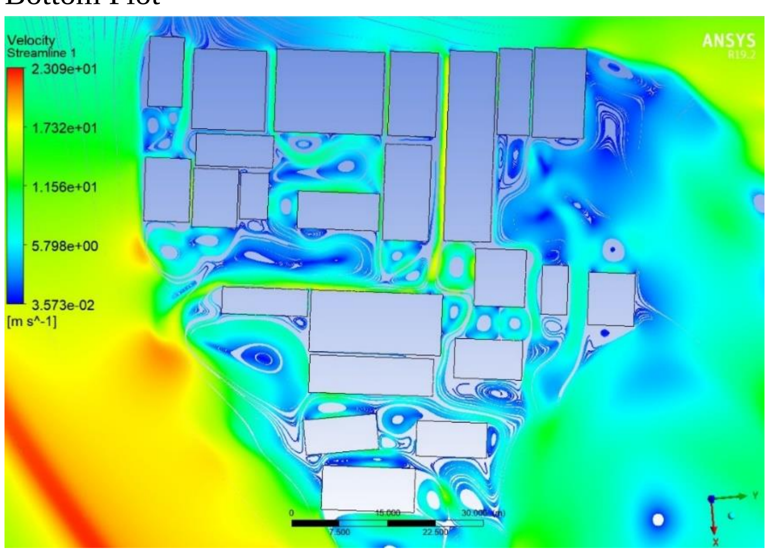


Figure 5- 10 Velocity Streamline in ANSYS Fluent

From figure 5.10, we can see that 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.

d. Velocity Vector

i) Top Plot

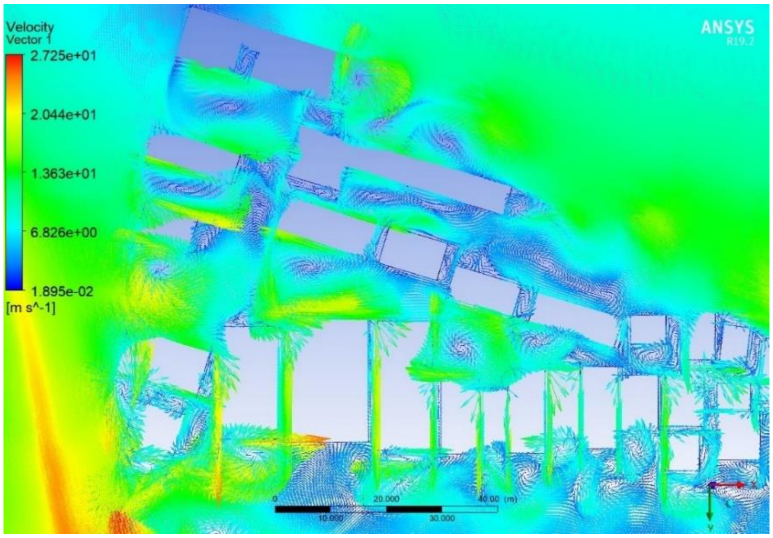


Figure 5- 11 Velocity Vector of Top Plot in ANSYS Fluent

Figure 5.11 and 5.12 present the velocity vector of the top and bottom plot as developed by ANSYS. This chart basically shows the motion of wind particles around the region which is very much similar to the streamlines. The circular white portions are the stagnation points.

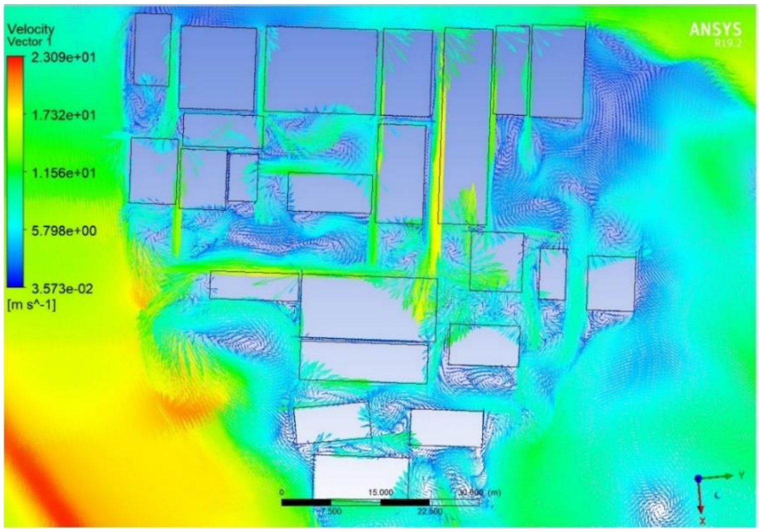


Figure 5- 12 Velocity contour of Bottom Plot in ANSYS Fluent

5.1.4 Proposed Solutions

5.1.4.1 Setback 1m & 2.5m

a) Planning and design

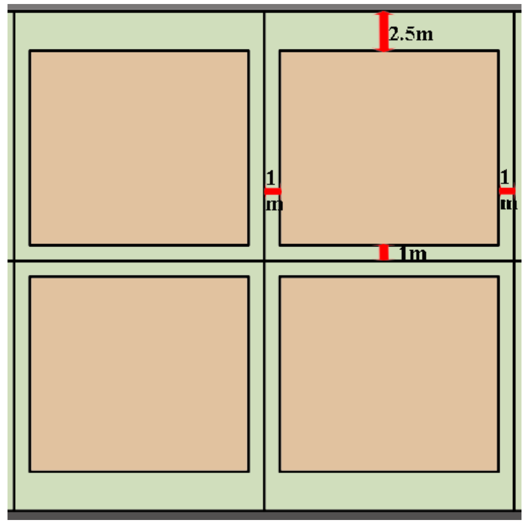


Figure 5- 13 Building arrangement for setback of 1m

Here, we have firstly analyzed the available By-laws of Siddharthanagar to redevelop the area starting from land allocation and planning. As, this region lies in a residential cum commercial region, the land plots that were used were around 256 sq. m (8 aana) with Ground Coverage of 67.9% and deducting the offsets we get the average building size to be used of 175 sq. m (5 aana). The offsets were 2.5m towards the road and 1m towards other buildings.

b) Simulation in ANSYS Fluent

i) Pressure Contour

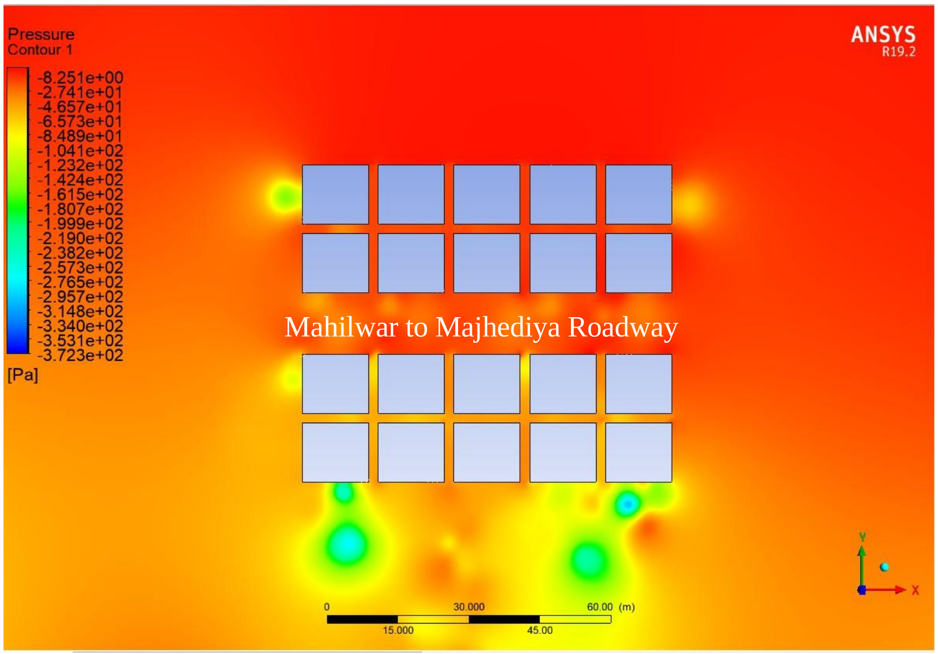


Figure 5- 14 Pressure Contour for Setback 1m in ANSYS Fluent

As we can see this basic land division and building arrangements has significantly improved the pressure distribution in the region with uniformity in the majority of the site. The pressure ranges from majority area with -8.25 Pascal to rarely present pressures of -180.7 Pascal.

ii) Velocity Contour

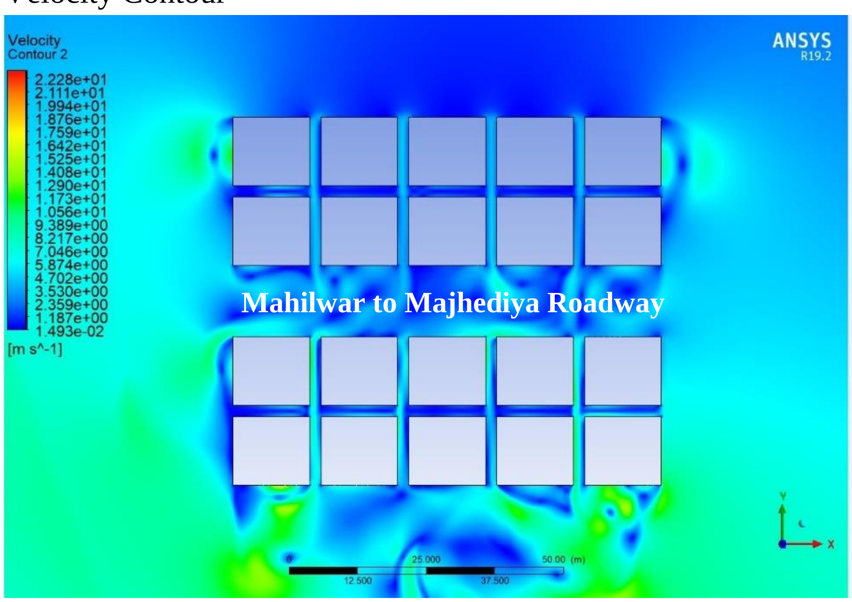


Figure 5- 15 Velocity Contour for Setback 1m in ANSYS Fluent

This solution has increased wind circulation around the buildings even more efficiently with velocity of 14m/s in each building face in the arrangement which is a significant improvement than the pre-existing condition in the area as shown in figure 5.15. This massive difference is due to the element size of the mesh which is kept around 1m. This basically proves that the solution presented has increased wind circulation to each and every house.

iii) Velocity Streamline

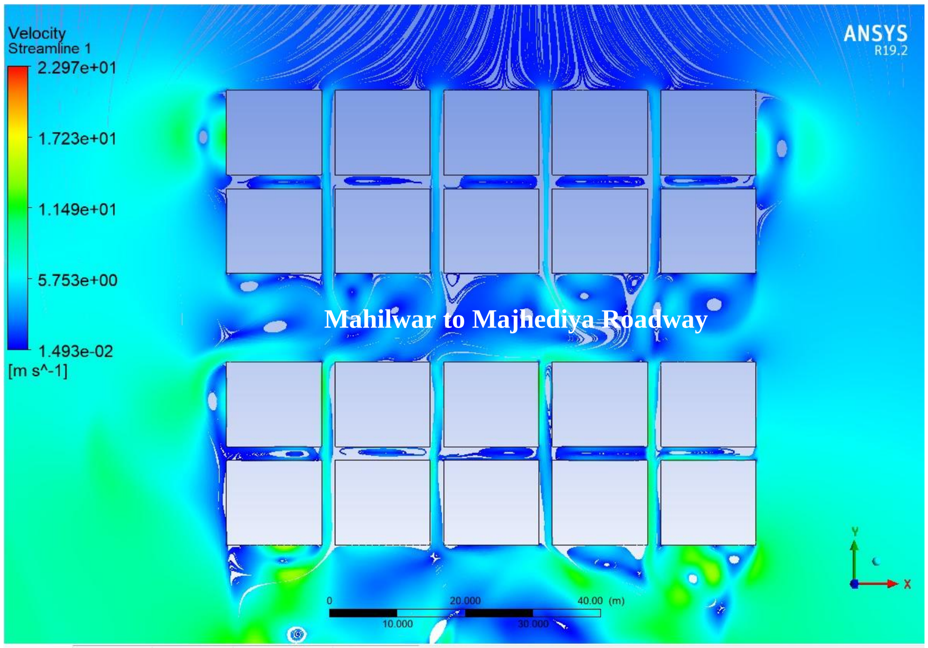


Figure 5- 16 Velocity Streamline of Setback 1m in ANSYS Fluent

As we can see from figure 5.16, the wind streamlines are well circulated and divided along the building walls in all 4 directions. This results in proper air circulation in the area causing reduction of stagnation points and recirculation in the regions resulting in fresh air circulation and enhanced thermal comfort in the region. This building arrangement and orientation is a basic planning practice practiced in western nations but still is quite effective in working as an effective wind corridor to the buildings. The well arrangement of buildings in the top plot and the bottom plot has led to proper air circulation in the bottom plots as well which was a hindrance in the real case scenario.

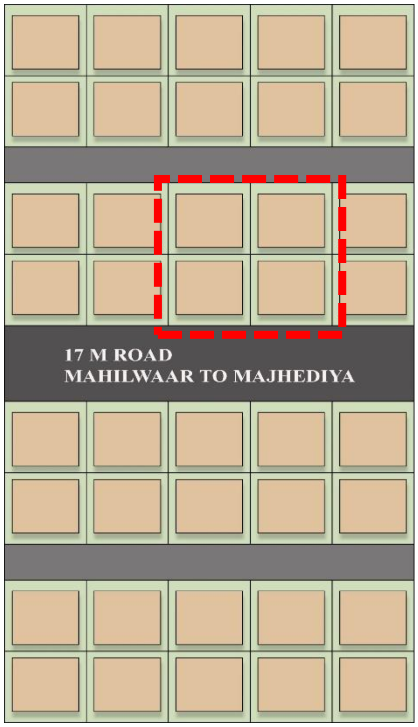
iv) Velocity Vector

Figure 5- 17 Velocity Vector of Setback 1m in ANSYS Fluent

From figure 5.17, with the help of vectors, we can see each and every wind particles movement in and around the region thus validating the streamline charts. This also supports the claim that this planning designs is much more efficient in terms of utilizing wind as a cooling element than the current base case scenario.

5.1.4.2 Setback 1.5m & 2.5m

a) Planning and design



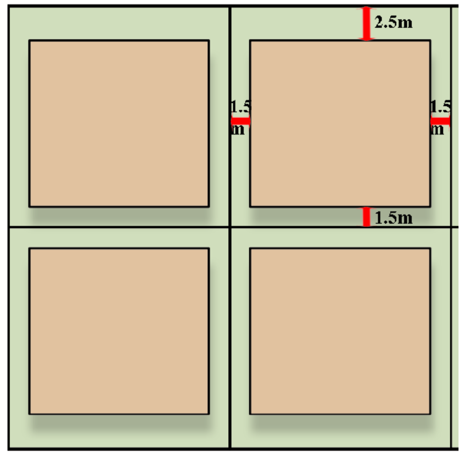


Figure 5- 18 Building arrangement for setback of 1.5m and 2.5m

Here, we have firstly analyzed the available By-laws of Siddharthanagar to redevelop the area starting from land allocation and planning. As, this region lies in a residential cum commercial region, the land plots that were used were around 256 sq. m (8 aana) with Ground Coverage of 60% and deducting the offsets we get the average building size to be used of 156 sq. m (5 aana). The offsets were 2.5m towards the road and 1.5 towards other buildings.

b) Simulation in ANSYS Fluent

i) Pressure Contour

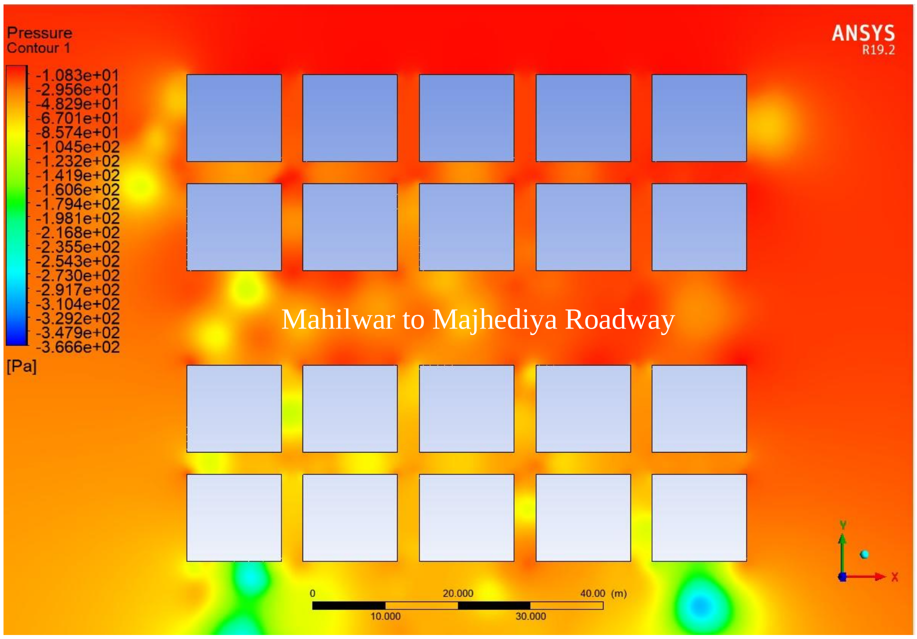


Figure 5- 19 Pressure Contour for Setback 1.5m in ANSYS Fluent

As we can see this basic land division and building arrangements has significantly improved the pressure distribution in the region with uniformity in the majority of the site. The pressure ranges from majority area with -10.83 Pascal to rarely present pressures of -370 Pascal.

ii) Velocity Contour

Figure 5- 20 Velocity Contour for Setback 1.5m in ANSYS Fluent

This solution has increased wind circulation around the buildings even more efficiently with velocity of 14m/s in each building face in the arrangement which is a significant improvement than the pre-existing condition in the area as shown in figure 5.20. This massive difference is due to the element size of the mesh which is kept around 1m. This basically proves that the solution presented has increased wind circulation to each and every house.

iii) Velocity Streamline

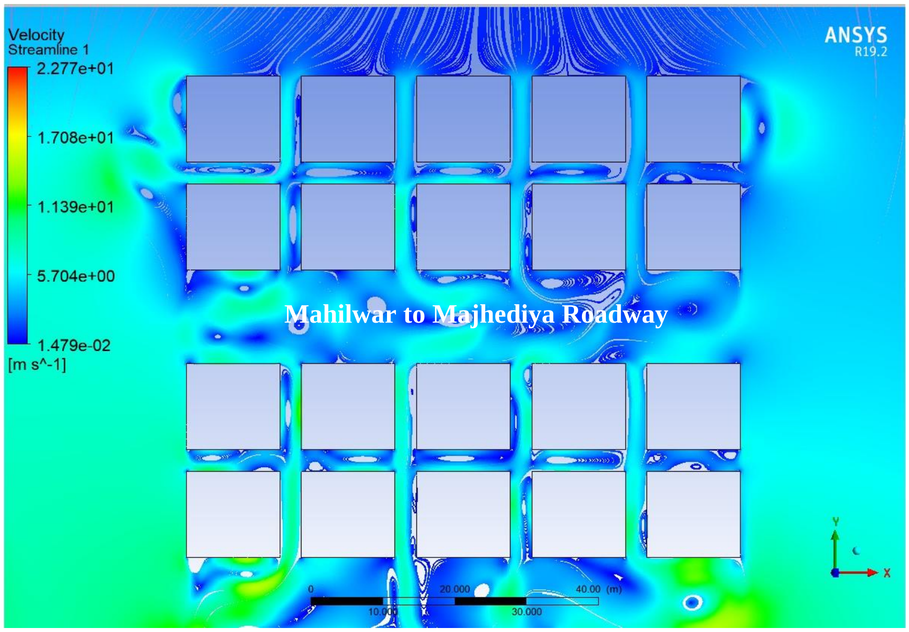


Figure 5- 21 Velocity Streamline of Setback 1.5m in ANSYS Fluent

As we can see from figure 5.21, the wind streamlines are well circulated and divided along the building walls in all 4 directions. This results in proper air circulation in the area causing reduction of stagnation points and recirculation in the regions resulting in fresh air circulation and enhanced thermal comfort in the region. This building arrangement and orientation is a basic planning practice practiced in western nations but still is quite effective in working as an effective wind corridor to the buildings. The well arrangement of buildings in the top plot and the bottom plot has led to proper air circulation in the bottom plots as well which was a hindrance in the real case scenario.

iv) Velocity Vector

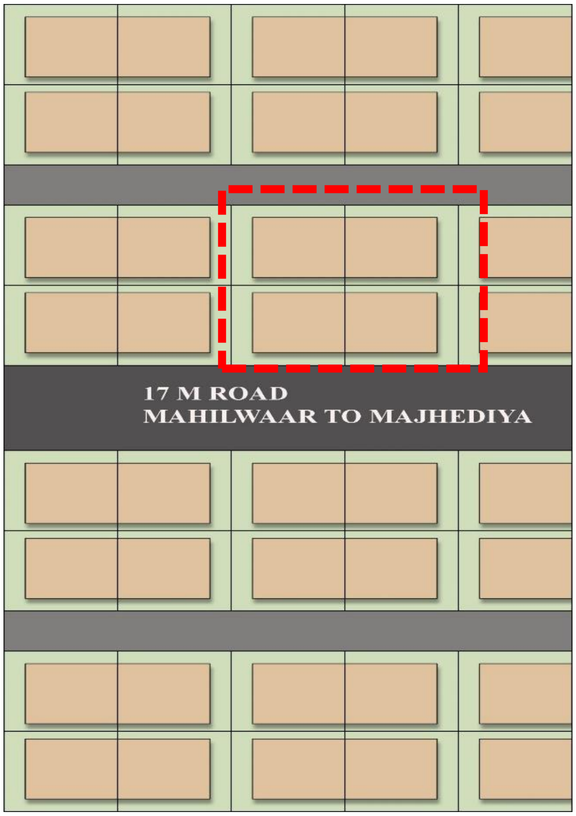
Figure 5- 22 Velocity Vector of Setback 1.5m in ANSYS Fluent

From figure 5.22, with the help of vectors, we can see each and every wind particles movement in and around the region thus validating the streamline charts. This also supports the claim that this planning designs is much more efficient in terms of utilizing wind as a cooling element than the current base case scenario.

5.1.4.2 Setback 1.5m, 3m & 2.5m

a) Planning and design

Here, we have firstly analyzed the available By-laws of Siddharthanagar to redevelop the area starting from land allocation and planning. As, this region lies in a residential cum commercial region, the land plots that were used were around 256 sq. m (8 aana) with Ground Coverage of 60% and deducting the offsets we get the average building size to be used of 156 sq. m (5 aana). The offsets were 2.5 m towards the road and 3m towards other buildings. Here the modern building arrangement that are adopted by housing developers in Nepal is adopted where 2 building are joined in one side to increase the spaces between them and the remaining adjoining building. The combined object (houses) is taken as a single rectangle of length 26m (sum of two building lengths) and breadth as 12m (individual buildings).



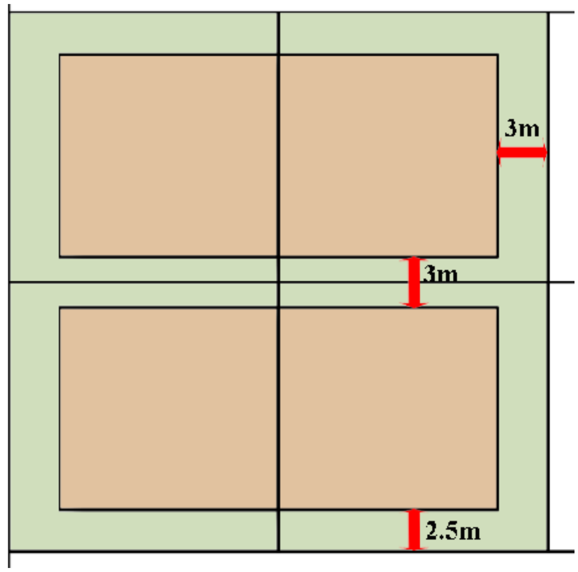


Figure 5- 23 Building Arrangement of Setback 1.5m, 3m and 2.5m

b) Simulation in Ansys Fluent

i) Pressure Contour

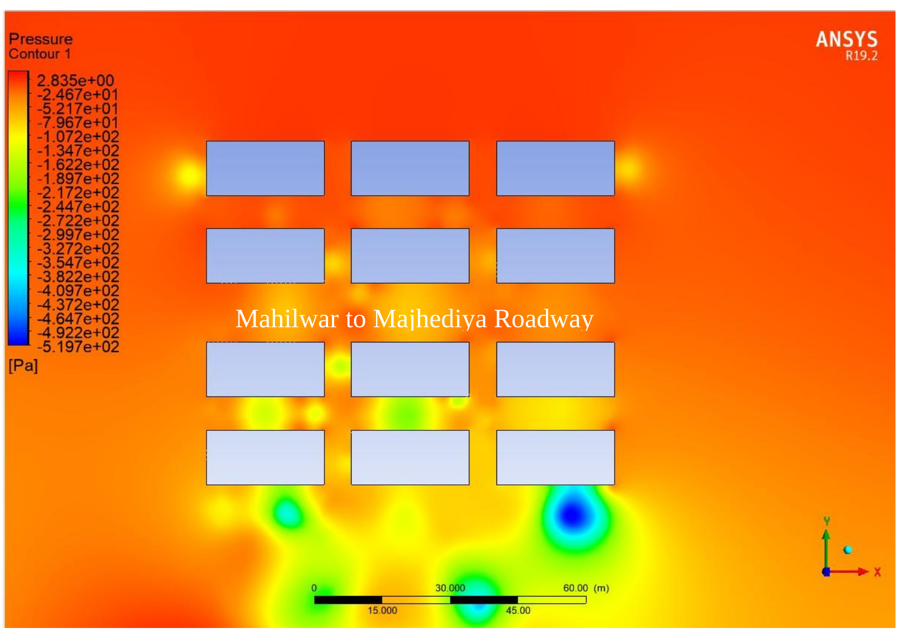


Figure 5- 24 Pressure contour of Setback 1.5m, 3m and 2.5m in ANSYS Fluent

This second type of building arrangement has also significantly improved the pressure distribution of the wind flow around the buildings with pressure of 3 Pa around building surfaces and a development of negative pressure zone in the east and south for pressure-density flow of the fluid as set up during solution section. Overall, the presence of proper wind pressure distribution leads to proper wind flow around the region due to convection as well.

ii) Velocity Contour

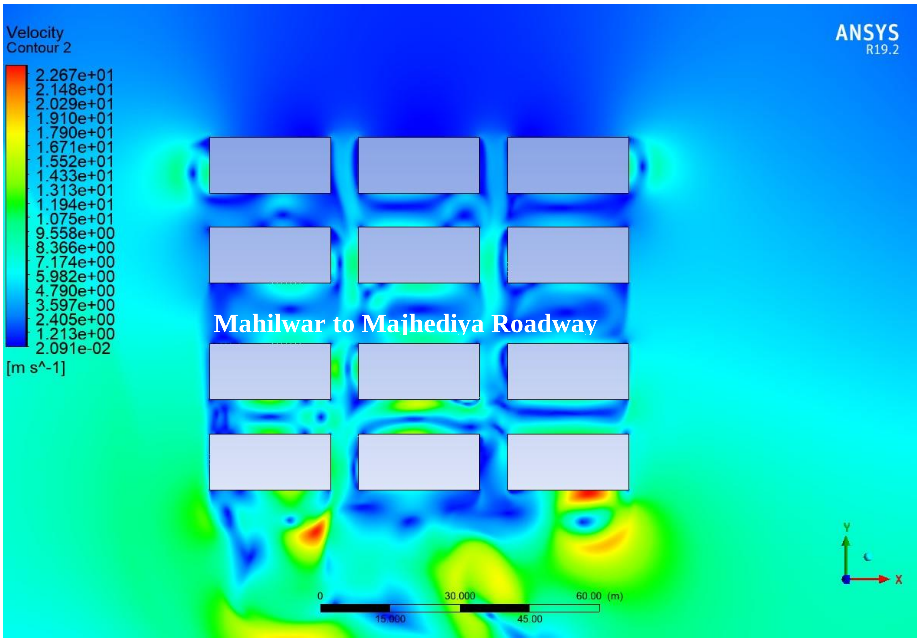


Figure 5- 25 Velocity Contour of setback 1.5m, 3m and 2.5m in ANSYS Fluent

From the figure 5.25 of velocity contour, we can see that the wind velocity along the corridors in the building faces is around 13m/s while it is around 2.5 m/s farther away from the face. This massive difference is due to the element size of the mesh which is kept around 1m. The wind corridors create a funnel effect which increases the wind speed considerably thus creating circulation of fresh air around the region.

iii) Velocity Streamline

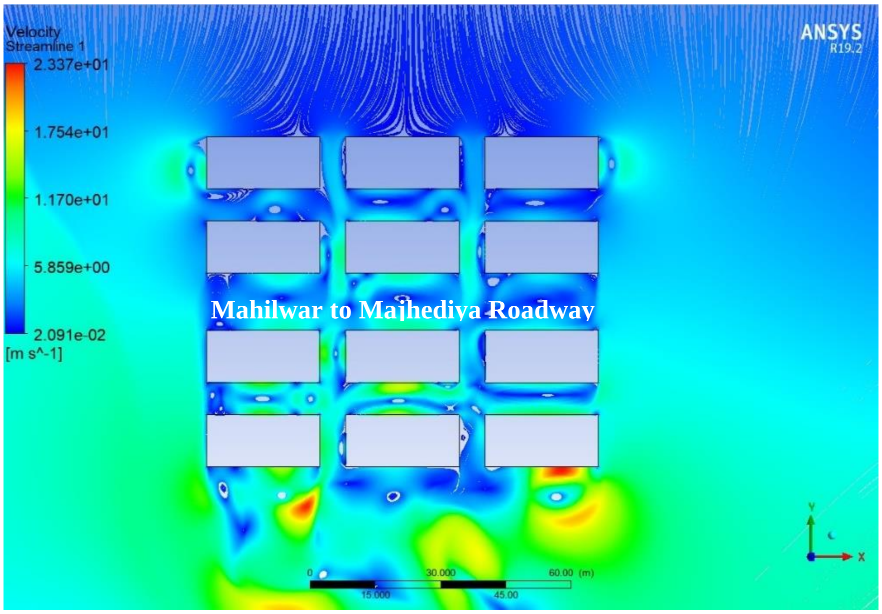


Figure 5- 26 A Velocity Streamline of setback 1.5m, 3m and 2.5m in ANSYS Fluent

We can clearly observe the increase of wind circulation in the region as compared to base case scenario with increased wind flow velocity thus creation of less amount of Stagnation points. This results in increased fresh air circulation in each building and results in enhanced thermal comfort to the pedestrians and residents of the area. This arrangement has allowed proper wind circulation in both the top as well as bottom plots which the desirable results of this research.

iv) Velocity Vector

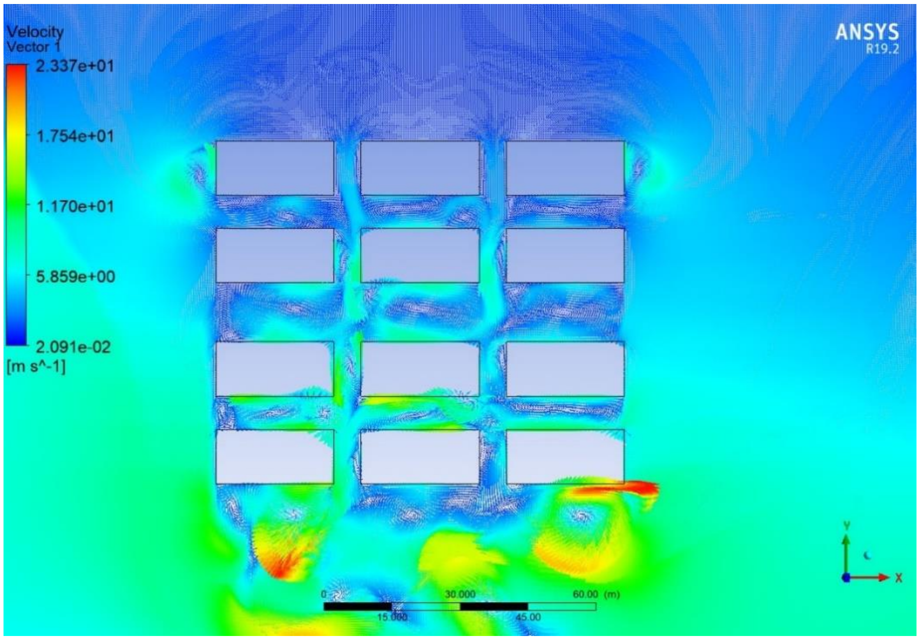


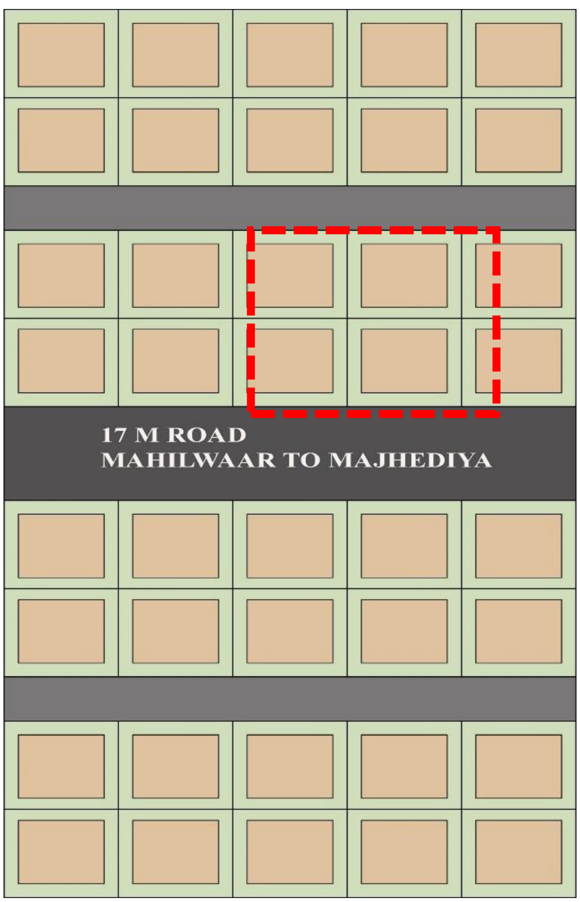
Figure 5- 27 Velocity Vector of setback 1.5m, 3m and 2.5m in ANSYS Fluent

From figure 5.27, with the help of vectors, we can see each and every wind particles movement in and around the region thus validating the streamline charts. This also supports the claim that this planning designs is much more efficient in terms of utilizing wind as a cooling element than the current base case scenario.

5.1.4.3 Setback 2m & 2.5m

a) Planning and design

Here, we have firstly analyzed the available By-laws of Siddharthanagar to redevelop the area starting from land allocation and planning. As, this region lies in a residential cum commercial region, the land plots that were used were around 256 sq. m (8 aana) with Ground Coverage of 54% and deducting the offsets we get the average building size to be used of 138 sq. m (4 aana). The offsets were 2.5 m towards the road and 2m towards other buildings.



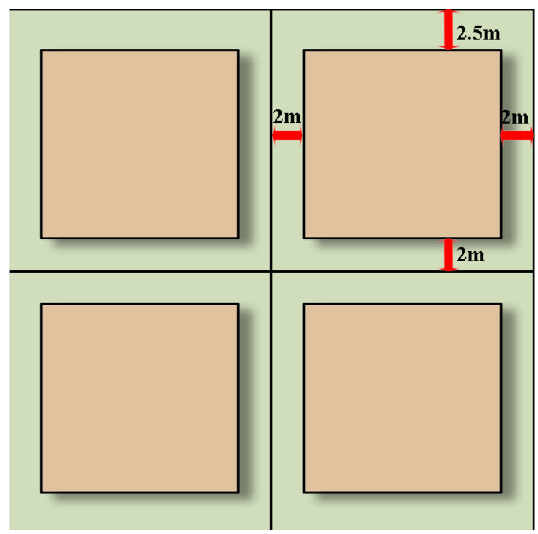


Figure 5- 28 Building arrangement for Setback 2m & 2.5m

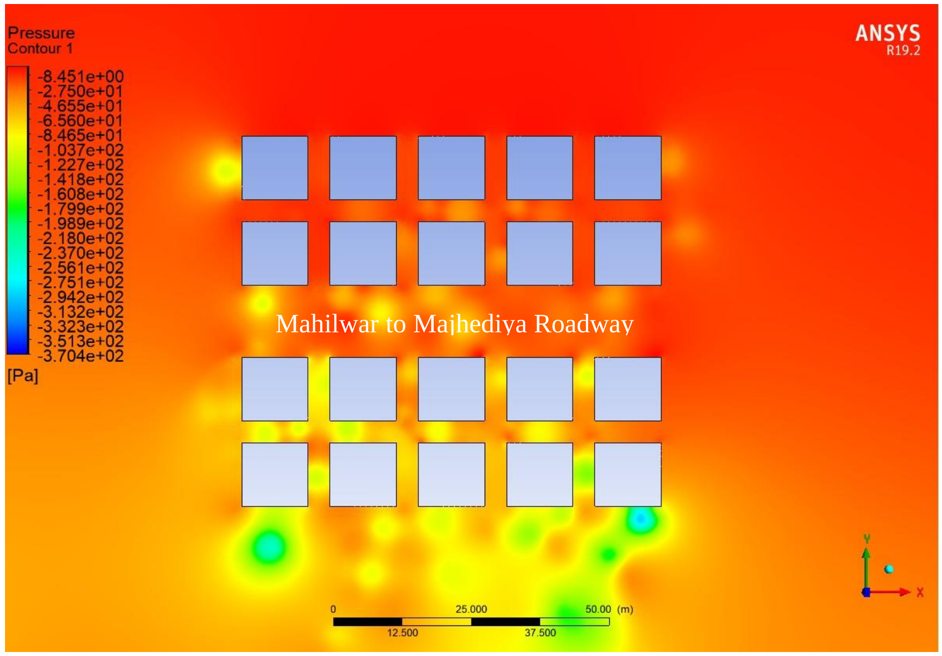


Figure 5- 29 Pressure contour of Setback 2m in ANSYS Fluent

This second type of building arrangement has also significantly improved the pressure distribution of the wind flow around the buildings with pressure of -8 Pa around building surfaces and a development of higher negative pressure zone in the east and south for pressure-density flow of the fluid as set up during solution section. Overall, the presence of proper wind pressure distribution leads to proper wind flow around the region due to convection as well.

ii) Velocity Contour

Figure 5- 30 Velocity Contour of Setback 2m in ANSYS Fluent

From the figure 5.30 of velocity contour, we can see that the wind velocity along the corridors in the building faces is around 14m/s. The wind corridors create a funnel effect which increases the wind speed considerably thus creating circulation of fresh air around the region.

iii) Velocity Streamline

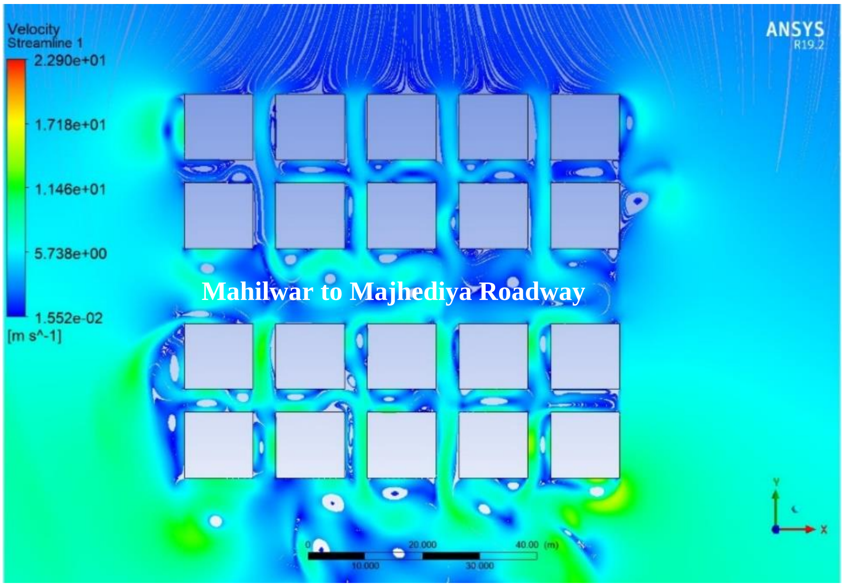


Figure 5- 31 A Velocity Streamline of Setback 2m in ANSYS Fluent

We can clearly observe the increase of wind circulation in the region as compared to base case scenario with increased wind flow velocity thus creation of less amount of Stagnation points. This results in increased fresh air circulation in each building and results in enhanced thermal comfort to the pedestrians and residents of the area. This arrangement has allowed proper wind circulation in both the top as well as bottom plots which the desirable results of this research.

iv) Velocity Vector

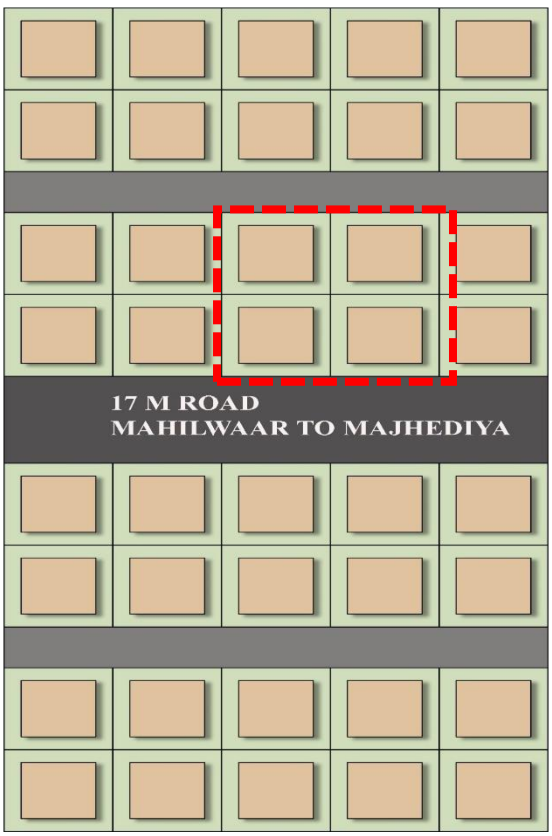
Figure 5- 32 Velocity Vector of Setback 2m in ANSYS Fluent

From figure 5.32, with the help of vectors, we can see each and every wind particles movement in and around the region thus validating the streamline charts. This also supports the claim that this planning designs is much more efficient in terms of utilizing wind as a cooling element than the current base case scenario.

5.1.4.4 Setback 2.5m

a) Planning and design

Here, we have firstly analyzed the available By-laws of Siddharthanagar to redevelop the area starting from land allocation and planning. As, this region lies in a residential cum commercial region, the land plots that were used were around 256 sq. m (8 aana) with Ground Coverage of 48% and deducting the offsets we get the average building size to be used of 121 sq. m (4 aana). The offsets were 2.5 m towards the road and 2.5m towards other buildings.



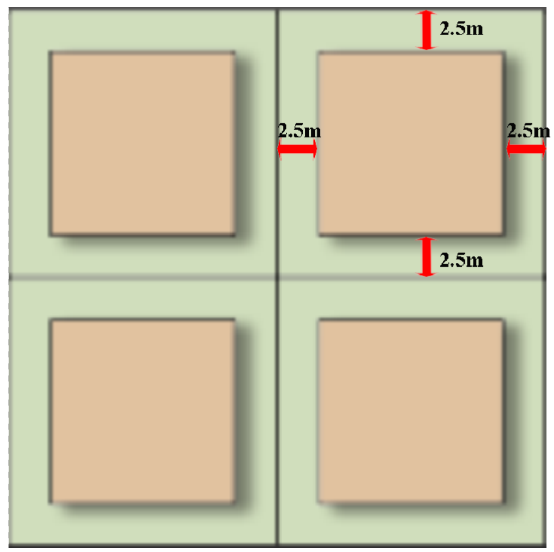


Figure 5- 33 Building arrangement for Setback 2.5m & 2.5m

b) Simulation in Ansys Fluent

i) Pressure Contour

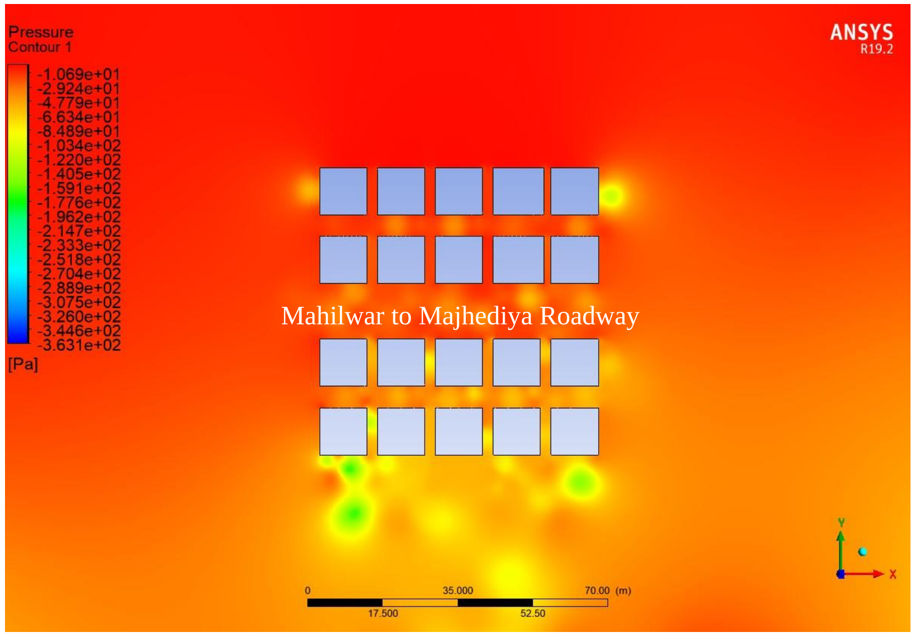


Figure 5- 34 Pressure contour of Setback 2.5m in ANSYS Fluent

This second type of building arrangement has also significantly improved the pressure distribution of the wind flow around the buildings with pressure of -10 Pa around building surfaces and a development of higher negative pressure zone in the east and south for pressure-density flow of the fluid as set up during solution section. Overall, the presence of proper wind pressure distribution leads to proper wind flow around the region due to convection as well.

ii) Velocity Contour

Figure 5- 35 Velocity Contour of Setback 2.5m in ANSYS Fluent

From the figure 5.35 of velocity contour, we can see that the wind velocity along the corridors in the building faces is around 13m/s. The wind corridors create a funnel effect which increases the wind speed considerably thus creating circulation of fresh air around the region.

iii) Velocity Streamline

Figure 5- 36 A Velocity Streamline of Setback 2.5m in ANSYS Fluent

We can clearly observe the increase of wind circulation in the region as compared to base case scenario with increased wind flow velocity thus creation of less amount of Stagnation points. This results in increased fresh air circulation in each building and results in enhanced thermal comfort to the pedestrians and residents of the area. This arrangement has allowed proper wind circulation in both the top as well as bottom plots which the desirable results of this research.

iv) Velocity Vector

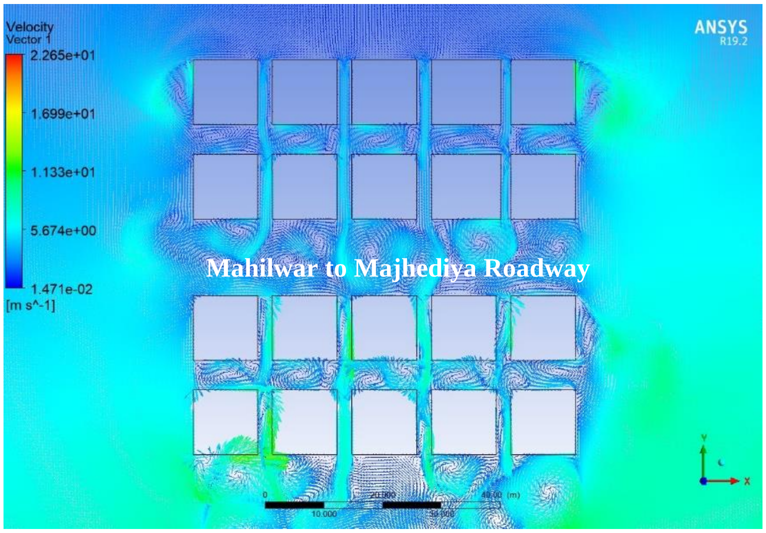


Figure 5- 37 Velocity Vector of Setback 2.5m in ANSYS Fluent

From figure 5.37, with the help of vectors, we can see each and every wind particles movement in and around the region thus validating the streamline charts. This also supports the claim that this planning designs is much more efficient in terms of utilizing wind as a cooling element than the current base case scenario.

5.1.4.5 Setback 2.5m and 3m

a) Planning and design

Here, we have firstly analyzed the available By-laws of Siddharthanagar to redevelop the area starting from land allocation and planning. As, this region lies in a residential cum commercial region, the land plots that were used were around 256 sq. m (8 aana) with Ground Coverage of 41% and deducting the offsets we get the average building size to be used of 105 sq. m (3 aana). The offsets were 2.5 m towards the road and 3m towards other buildings.

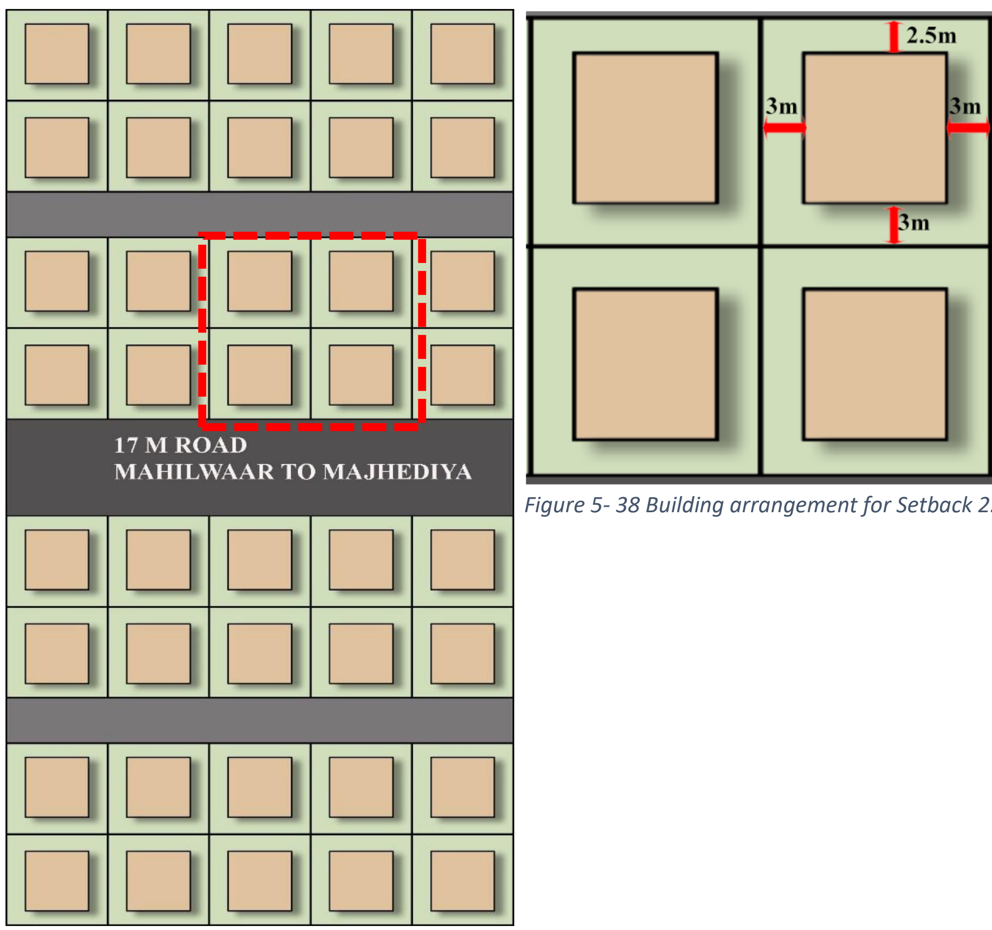


Figure 5- 38 Building arrangement for Setback 2.5m & 3m

b) Simulation in Ansys Fluent

i) Pressure Contour

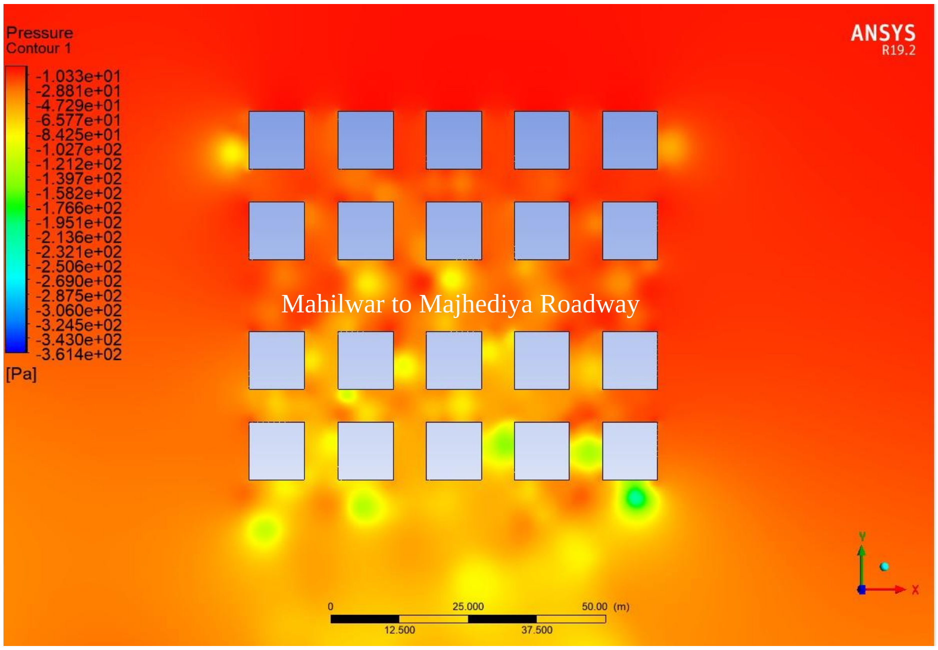


Figure 5- 39 Pressure contour of Setback 3m in ANSYS Fluent

This second type of building arrangement has also significantly improved the pressure distribution of the wind flow around the buildings with pressure of -10 Pa around building surfaces and a development of higher negative pressure zone in the east and south for pressure-density flow of the fluid as set up during solution section. Overall, the presence of proper wind pressure distribution leads to proper wind flow around the region due to convection as well.

ii) Velocity Contour

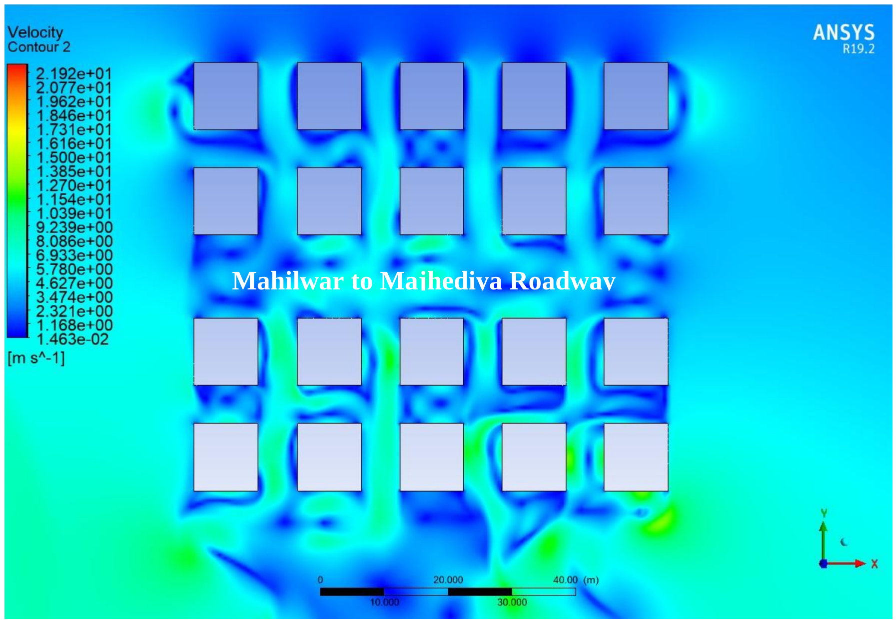


Figure 5- 40 Velocity Contour of Setback 3m in ANSYS Fluent

From the figure 5.40 of velocity contour, we can see that the wind velocity along the corridors in the building faces is around 12m/s. The wind corridors create a funnel effect which increases the wind speed considerably thus creating circulation of fresh air around the region.

iii) Velocity Streamline

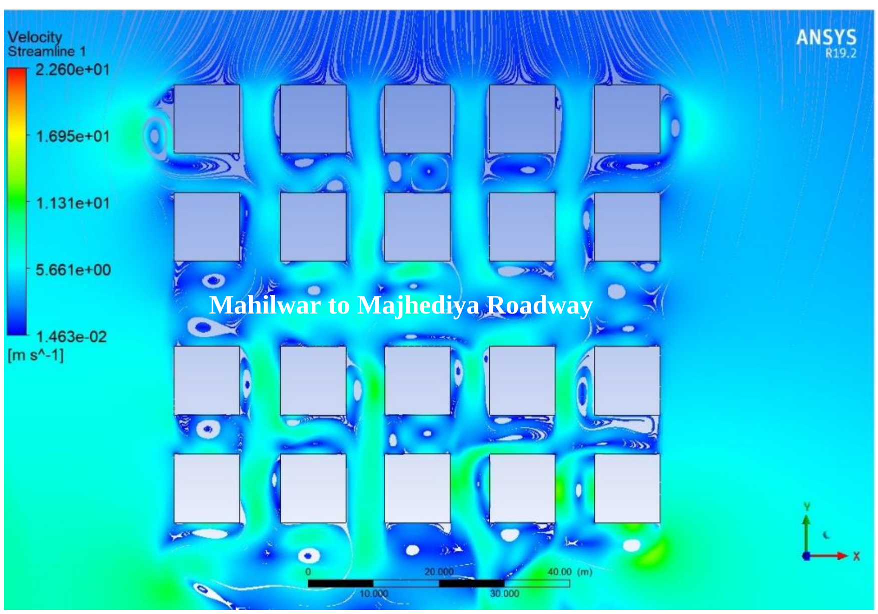


Figure 5- 41 A Velocity Streamline of Setback 3m in ANSYS Fluent

We can clearly observe the increase of wind circulation in the region as compared to base case scenario with increased wind flow velocity thus creation of less amount of Stagnation points. This results in increased fresh air circulation in each building and results in enhanced thermal comfort to the pedestrians and residents of the area. This arrangement has allowed proper wind circulation in both the top as well as bottom plots which the desirable results of this research.

iv) Velocity Vector

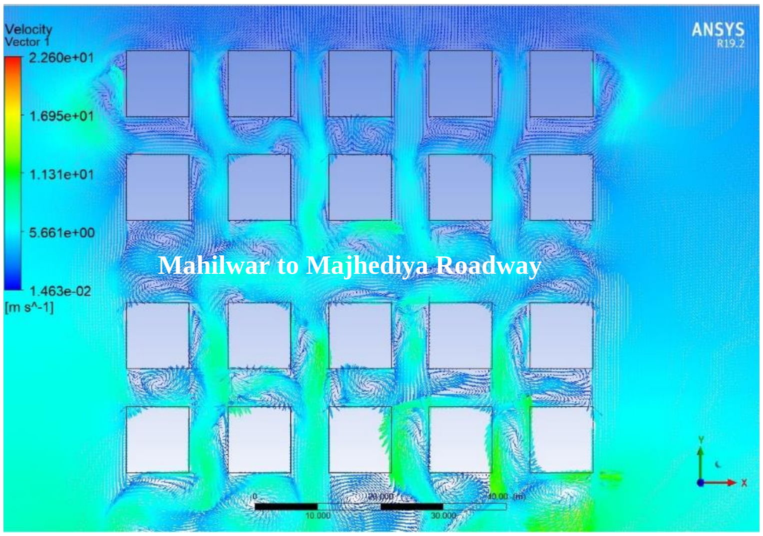


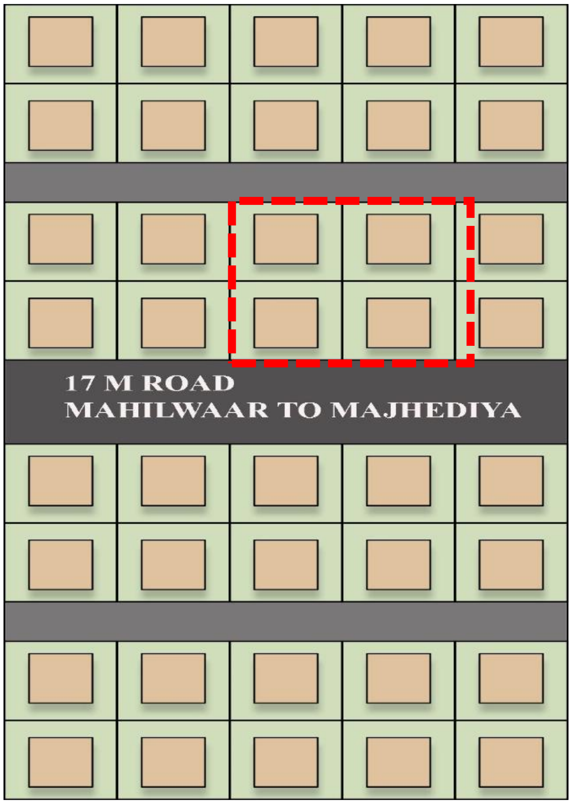
Figure 5- 42 Velocity Vector of Setback 3m in ANSYS Fluent

From figure 5.42, with the help of vectors, we can see each and every wind particles movement in and around the region thus validating the streamline charts. This also supports the claim that this planning designs is much more efficient in terms of utilizing wind as a cooling element than the current base case scenario.

5.1.4.6 Setback 2.5m and 3.5m

a) Planning and design

Here, we have firstly analyzed the available By-laws of Siddharthanagar to redevelop the area starting from land allocation and planning. As, this region lies in a residential cum commercial region, the land plots that were used were around 256 sq. m (8 aana) with Ground Coverage of 35% and deducting the offsets we get the average building size to be used of 90 sq. m (2.7 aana). The offsets were 2.5 m towards the road and 3.5m towards other buildings.



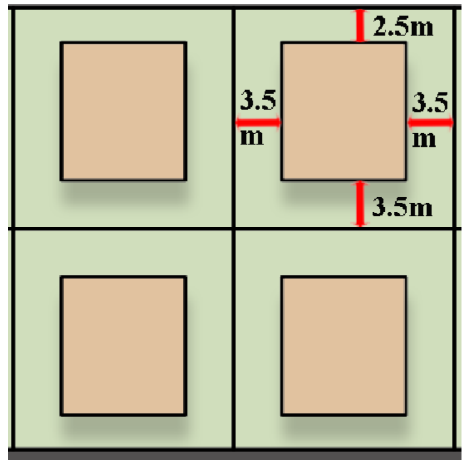


Figure 5- 43 Building arrangement for setback 2.5m and 3.5m

b) Simulation in Ansys Fluent

i) Pressure Contour

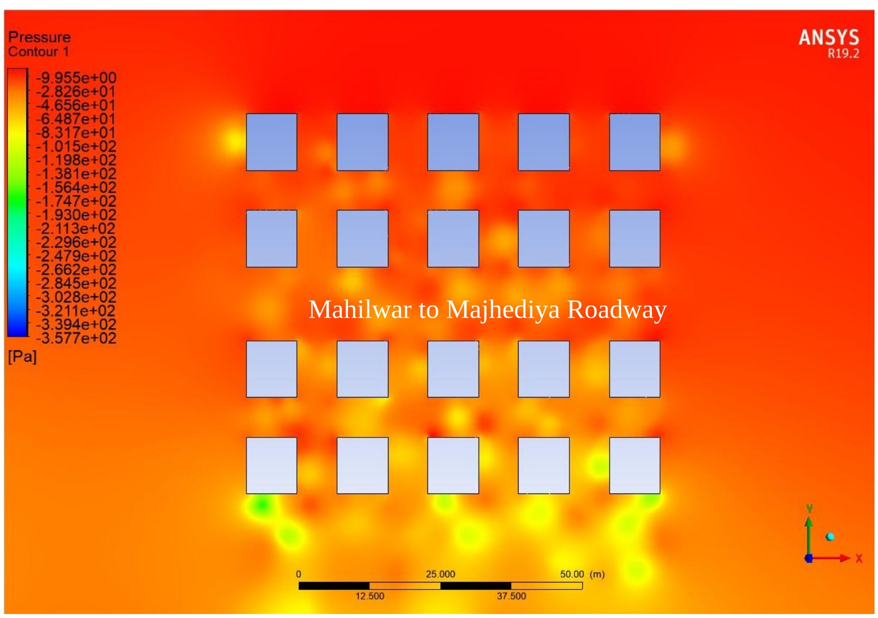


Figure 5- 44 Pressure contour of Setback 3.5m in ANSYS Fluent

This second type of building arrangement has also significantly improved the pressure distribution of the wind flow around the buildings with pressure of -10 Pa around building surfaces and a development of higher negative pressure zone in the east and south for pressure-density flow of the fluid as set up during solution section. Overall, the presence of proper wind pressure distribution leads to proper wind flow around the region due to convection as well.

ii) Velocity Contour

Figure 5- 45 Velocity Contour of Setback 3.5m in ANSYS Fluent

From the figure 5.45 of velocity contour, we can see that the wind velocity along the corridors in the building faces is around 10m/s. The wind corridors create a funnel effect which increases the wind speed considerably thus creating circulation of fresh air around the region.

iii) Velocity Streamline

Figure 5- 46 A Velocity Streamline of Setback 3.5m in ANSYS Fluent

We can clearly observe the increase of wind circulation in the region as compared to base case scenario with increased wind flow velocity thus creation of less amount of Stagnation points. This results in increased fresh air circulation in each building and results in enhanced thermal comfort to the pedestrians and residents of the area. This arrangement has allowed proper wind circulation in both the top as well as bottom plots which the desirable results of this research.

iv) Velocity Vector

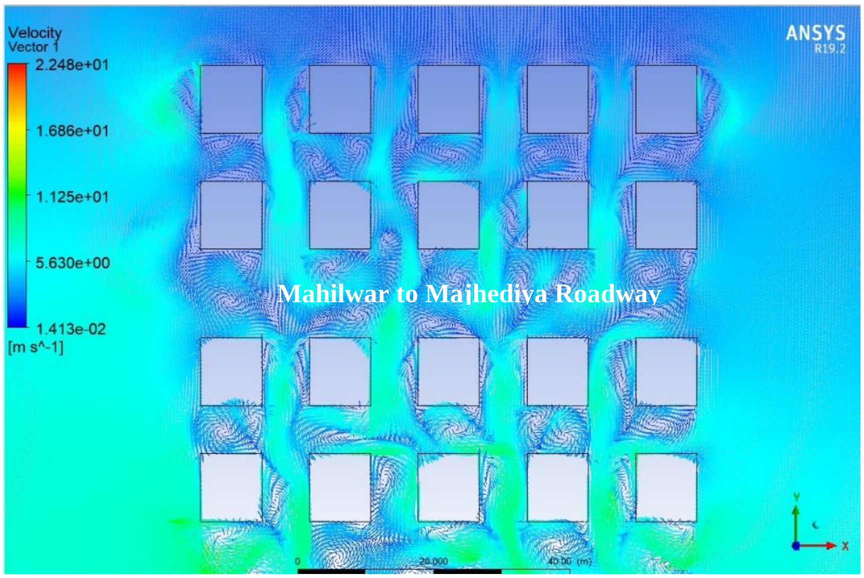


Figure 5- 47 Velocity Vector of Setback 3.5m in ANSYS Fluent

From figure 5.47, with the help of vectors, we can see each and every wind particles movement in and around the region thus validating the streamline charts. This also supports the claim that this planning designs is much more efficient in terms of utilizing wind as a cooling element than the current base case scenario.

5.1.4.7 Setback 2.5m and 4m

a) Planning and design

Here, we have firstly analyzed the available By-laws of Siddharthanagar to redevelop the area starting from land allocation and planning. As, this region lies in a residential cum commercial region, the land plots that were used were around 256 sq. m (8 aana) with Ground Coverage of 30% and deducting the offsets we get the average building size to be used of 76 sq. m (2.3 aana). The offsets were 2.5 m towards the road and 4m towards other buildings.

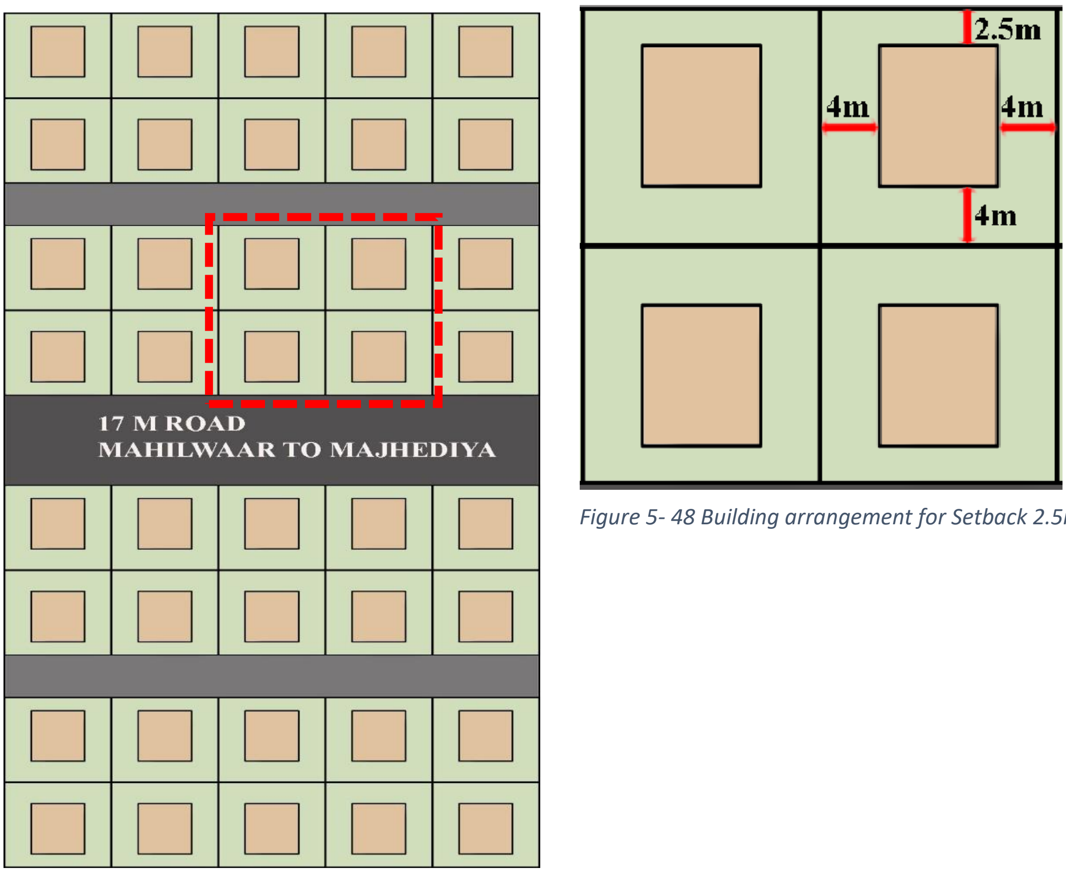


Figure 5- 48 Building arrangement for Setback 2.5m & 4m

b) Simulation in Ansys Fluent

i) Pressure Contour

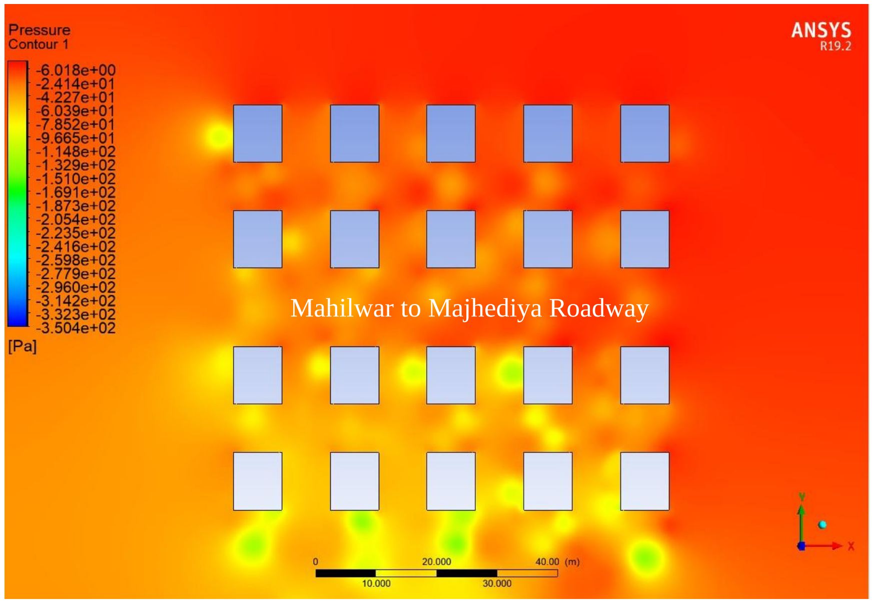


Figure 5- 49 Pressure contour of Setback 4m in ANSYS Fluent

This second type of building arrangement has also significantly improved the pressure distribution of the wind flow around the buildings with pressure of -6 Pa around building surfaces and a development of higher negative pressure zone in the east and south for pressure-density flow of the fluid as set up during solution section. Overall, the presence of proper wind pressure distribution leads to proper wind flow around the region due to convection as well.

ii. Velocity Contour

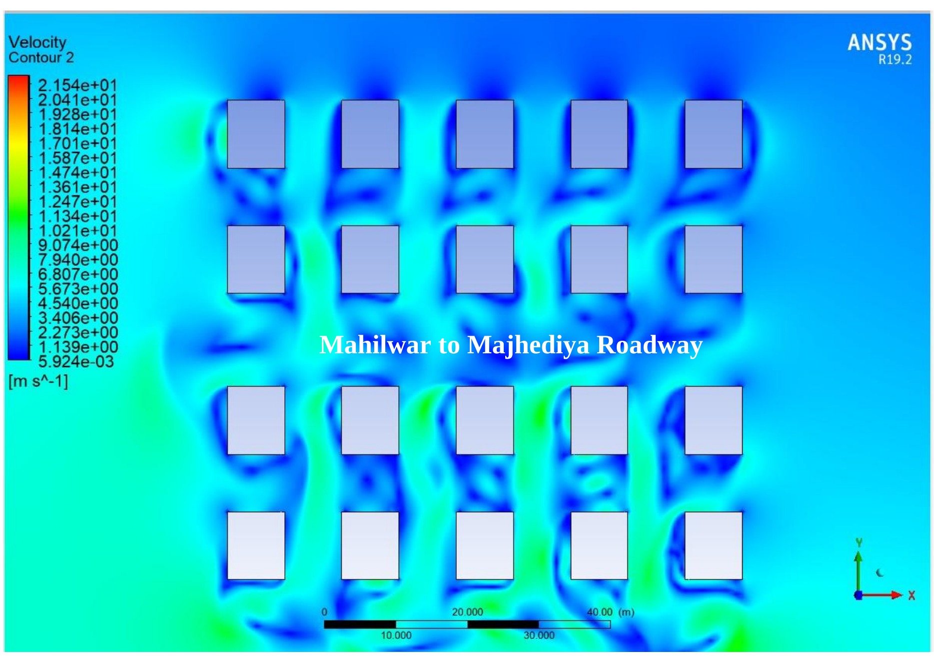


Figure 5- 50 Velocity Contour of Setback 4m in ANSYS Fluent

From the figure 88 of velocity contour, we can see that the wind velocity along the corridors in the building faces is around 8m/s. The wind corridors create a funnel effect which increases the wind speed considerably thus creating circulation of fresh air around the region.

iii) Velocity Streamline

Figure 5- 51 A Velocity Streamline of Setback 4m in ANSYS Fluent

We can clearly observe the increase of wind circulation in the region as compared to base case scenario with increased wind flow velocity thus creation of less amount of Stagnation points. This results in increased fresh air circulation in each building and results in enhanced thermal comfort to the pedestrians and residents of the area. This arrangement has allowed proper wind circulation in both the top as well as bottom plots which the desirable results of this research.

iv) Velocity Vector

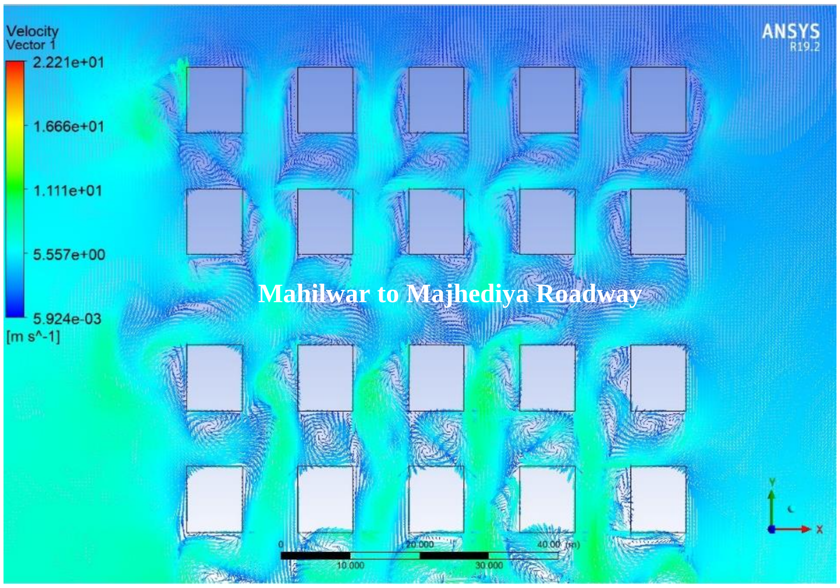


Figure 5- 52 Velocity Vector of Setback 4m in ANSYS Fluent

From figure 5.52, with the help of vectors, we can see each and every wind particles movement in and around the region thus validating the streamline charts. This also supports the claim that this planning designs is much more efficient in terms of utilizing wind as a cooling element than the current base case scenario.

5.1.5 Comparison between Base Case Scenario and Solutions

Table 5- 2 Comparison between Cases

Case	Velocity (m/s)	Stagnation Points	Pressure Distribution
Base Case (Top Plot)	4	Maximum amount	Non-Uniform
Base Case (Bottom Plot)	8	Less than Top Plot	Non-Uniform
Setback			
1m	13	Less than Base case	Uniform
1.5m & 3m	13	Least amount	Uniform
1.5m	14	Less than Base case	Uniform
2m	14	Less than Base case	Uniform
2.5m	13	Less than Base case	Uniform
3m	12	Less than Base case	Uniform
3.5m	10	Less than Base case	Uniform
4m	8	Less than Base case	Uniform

From Table 5.2, we can see the difference in variables that have been simulated in ANSYS Fluent among the Base case and the presented solutions. The base case has non-uniform pressure distribution among the buildings with extremely high amount of stagnation points due to uneven distribution of wind flow. The narrow passages between buildings have high velocity wind due to funnel effect but the remaining area of the region have very less wind velocity due to uneven orientation and arrangements. The increased amount of stagnation points is also the result of haphazard urban planning and development. The presented solutions are simple and very much possible to implement in the near future in the same site. The solutions were presented by just following the nation verified locally implemented Building Bylaws 2072 of Siddharthanagar (reference).

The land plot size was selected as the average land size in the base region of 256 sq.m (8 aana) with the maximum ground coverage take of 60% with the setbacks of 1.5m towards the land boundary and 2.5m towards the main road. The subsequent setbacks were taken in an increment of 0.5m without changing the setback towards the road. The minimum ground coverage was taken 30% at 76 sq.m (2.3 aana). The proposed solutions have proper pressure distribution around each building with reduced stagnation points than the base case. The uniform arrangement and orientation of buildings towards the wind direction incorporates well developed wind corridors between them. This development of uniform wind corridors increases the wind velocity all over the region with improved air circulation. With a minimum velocity achieved at 8m/s with setback 4m, this wind speed is more than good enough to reduce the air temperature around the region considerably as any wind speed more than 2m/s can reduce the temperature by 2℃ (Serteser and Ok, 2009).

Chapter 6: Conclusion

With the rise in population and housing demands, the use of low albedo materials has increased which have high thermal capacity. This increases the temperature of the micro climate which is ultimately called Urban Heat Island effect. Random planning of buildings without appropriate implementation of building bylaws of the area actually degrades the thermal comfort of the region by blocking winds which reduces the cooling potential of earth’s natural element. Building bylaws are designed to increase the aesthetics and functionality of the buildings in the area without compromising the structural safety. So, designers and planners must use the bylaws and architectural ideas in such a way that the thermal comfort of the region is not ignored.

Development of wind corridors once done through thumb rules can be made more scientific and accurate through the assistance of CFD simulation among which ANSYS Fluent is a major tool. The use of CFD helps to understand the properties of the fluid flow along with its pathway which helps us to plan the placement and orientation of any structures in its pathway without compromising its flow. ANSYS Fluent with its easy user interface helps researcher to carry out multiple research projects related to fluid flow thus enabling the research accuracy to be increased. This tool is a supplementary tool which should be used after understanding the fundamentals of fluid motion by the user. The interactive interface along with easy graphical output helps us to understand and analyze the results even more accurately and efficiently.

Findings in the base case scenario show that improper arrangement of buildings along with their orientation led to improper pressure distribution in the region along with reduced circulation of fresh air within it which causes increase in temperature and reduction of thermal comfort. The irregular pressure variation increases the recirculation of the same air which creates stagnation point resulting in uncomfortable breathing conditions. The irregular arrangement of buildings actually blocks the air flow to the leeward regions which again creates thermally uncomfortable situation for the people interacting with the space.

In the proposed solutions, we can see that by just following basic building bylaws i.e., Ground Coverage and setbacks, we can increase the wind flow to all buildings present in the area. The pressure distribution also becomes uniform which reduces the formation of air recirculation and stagnation points. The buildings are arranged in such a way that the wind corridors are uniformly divided which increases the wind velocity in them due to funnel effect. Any increase of velocity past 2m/s is found to have reduced the temperature by 2℃ which means that an average velocity of 12m/s will reduce the temperature significantly.

Multiple setbacks were selected from 1.5m to 4m with an increment of 0.5m each where the analysis was done. The velocities were quite different in each case where the velocity decreased with increase in setbacks which was evident as the area of wind corridors increased. The rule of continuity states that "Velocity of flow increases with decrease in flow area" which has also been verified by ANSYS Fluent.

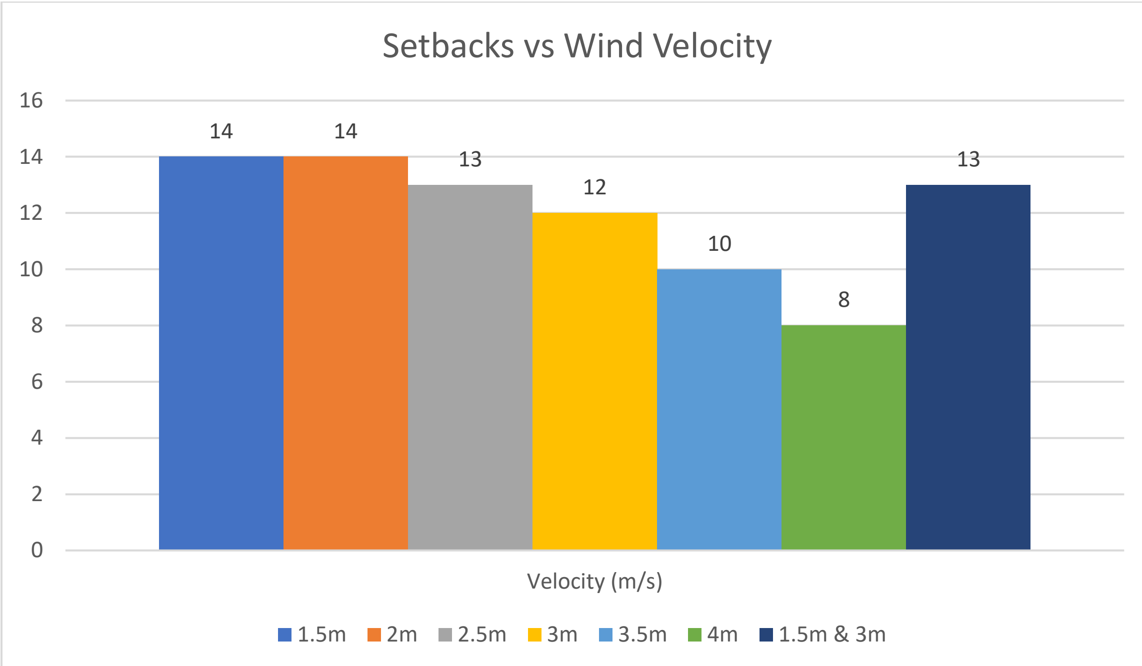


Chart 6- 1 Setback vs Wind velocity

From the above figure we can see that a setback of 1.5m-2m was found to be the best that can be incorporated in the field in future developments of similar building structures and arrangements.

To summarize, the development of wind corridors through the CFD analysis helps to increase the pedestrian and residential thermal comfort in the region significantly by guiding the wind flow to each and every building in the arrangement. Increased wind flow also helps the person to perceive the same temperature differently (a bit cooler) which also helps to increase mobility within the region as proven by the field survey carried out during the research in the site. The increase in pedestrian mobility contributes positively to the economy of the area which again helps.

Further research has to be done to the hot and humid regions of the nation to further validate the conclusion. Since the research was carried out in limited time during the peak summer season, the wind corridors could actually affect the thermal comfort of the area during winter which is again not acceptable. The research was carried out to the climate specific to Lumbini region which might differ to another region in the same climatic zone of the nation. So, further research can be carried out in other cities in the same climatic zone within the nation for additional validation to the concept.

Chapter 7: Limitations

1. The study was carried out in a single city of Lumbini region within a certain climatic condition
2. The weather data obtained from DOHM cannot be fully relied upon due to multiple inaccuracies
3. The availability of limited computation memory has certainly reduced the accuracy of the results
4. Weather data of the site was only recorded for 15 days due to time limitation of the research as well as due to safety concerns
5. Survey answers cannot be fully claimed as accurate due to subjective nature of answers
6. The simulation is carried out in 2D without considering the height of the buildings which also can play a major role in wind flow
7. Multiple external variables such as vegetations and water bodies have been ignored which can also play a major role in reduction of street temperature of the region
8. The surface roughness of the buildings and pavements was not considered which can affect the wind flow due to drag coefficient.

References

1. *Alam, S. (n.d). Ansys Fluent Tutorial. Retrieved from Sunglass: https://sunglass.io/ansys-fluent-tutorial/#Moving-forward*
2. *Al-Sallal, K.A., & Al-Rais, L. (2011). Outdoor airflow analysis and potential for passive cooling in the traditional urban context of Dubai. Renewable Energy, 36, 2494-2501.*
3. *Bhattacharyya, S., Abraham, J. P., Cheng, L., & Gorman, J. (2021). Introductory Chapter: A Brief History of and Introduction to Computational Fluid Dynamics. Intechopen.*
4. *Byjuus. (n.d). Continuity Equation. Retrieved from Byjuus: https://byjus.com/physics/continuity-equation/*
5. *Centre, G. R. (n.d). CFD Analysis Process. Glenn Research Process.*
6. *CFD Modelling. (2018). Retrieved from cfdflowengineering.com: https://cfdflowengineering.com/basics-of-cfd-modeling-for-beginners/*
7. *cfdflowengineering. (2018). CFD Modelling. Retrieved from cfdflowengineering.com: https://cfdflowengineering.com/basics-of-cfd-modeling-for-beginners/*
8. *ChampionTraveler. (n.d). The Best Time to Visit Lumbini, Nepal for Weather, Safety, & Tourism. Retrieved from ChampionTraveler: https://championtraveler.com/dates/best-time-to-visit-lumbini*
9. *Chang, S., Jiang, Q., & Zhao, Y. (2018). Integrating CFD and GIS into the Development of Urban Ventilation Corridors: A Case Study in Changchun City, China. MDPI.*
10. *Eldesoky, A., Colaninno, N., & Morello, E. (2020). MAPPING URBAN VENTILATION CORRIDORS AND ASSESSING THEIR IMPACT UPON THE COOLING EFFECT OF GREENING SOLUTIONS. The International Archives of the Photogrammetry, Remote Sensing and Spatial Information Sciences, 665-672.*
11. *Engineering, F. (2020). https://www.femto.eu/stories/what-is-cfd/. Retrieved from www.femto.eu.*
12. *Fanger, P.O. (1970) Thermal Comfort, Copenhagen, Danish Technical Press.*
13. *Fanger, P.O. and Toftum, J. (2002) Extension of the PMV model to non-air-conditioned buildings in warm climates, Energy Build., 34, 533–536.*
14. *Gu, K., Fang, Y., Qian, Z., Sun, Z., & Wang, A. (2020). Spatial planning for urban ventilation corridors by urban climatology. Ecosystem Health and Sustainability.*
15. *Hong, D.-L., & Chien, S.-S. (2018). 'Summoning' Wind for Urban Cooling: Urban Wind Corridor Projects in China. Designing Cooler Cities, 137-151.*
16. *Hsieh, C., & Wu, K. (2012). Climate-Sensitive Urban Design Measures for Improving the Wind Environment for Pedestrians in a Transit-Oriented Development Area . Journal of Sustainable Development , 46-58.*
17. *Hsieh, C.-M., & Huang, H.-C. (2016). Mitigating urban heat islands: A method to identify potential wind corridor for cooling and ventillation. Elsevier, 130-143.*
18. *Ibrahim, S. H., Ibrahim, N. I., Wahit, J., Goh, N. A., Koesmeri, D. R., & Nawi, M. N. (2018). The impact of Road Pavement on Urban Heat Island (UHI) Phenomenon. IJTech.*

19. *Johansson, Erik. (2006). Influence of urban geometry on outdoor thermal comfort in a hot dry climate: A study in Fez, Morocco. Building and Environment. 41. 1326-1338. 10.1016/j.buildenv.2005.05.022.*
20. *Kubota, T., Miura, M., Tominaga, Y., Mochida, A., 2008. 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. Build. Environ. 43 (10), 1699–1708*
21. *Kumar, E. A. (2022). Types of Fluid Flow. Retrieved from The Mechanical Engineering.com: https://themechanicalengineering.com/types-of-fluid-flow/*
22. *Kuzmin, D. (2017). Introduction to Computational Fluid Dynamics. Dortmund: TU Dortmund. Retrieved from https://www.mathematik.tu-dortmund.de/sites/cfdintro*
23. *Larabee, D. R. (2007, April). Literature Review: Types of Literature Reviews. Retrieved from uscupstate.libguides: https://uscupstate.libguides.com/c.php?g=627058&p=6601225*
24. *Liu, D., Zhou, S., Wang, L., Chi, Q., Zhu, M., Tang, W., . . . Cui, Y. (2022). Research on the Planning of an Urban Ventilation Corridor Based on the Urban Underlying Surface Taking Kaifeng City as an Example. MDPI.*
25. *Liu, W., Zhang, G., & Jiang, Y. W. (2021). Effective Range and Driving Factors of the Urban Ventilation Corridor Effect on Urban Thermal Comfort at Unified Scale with Multisource Data. MDPI.*
26. *Liu, Y., Cheng, P., Chen, P., & Zhang, S. (2020). Detection of wind corridors based on "Climatopes": a study in central Ji'nan. Theoretical and Applied Climatology.*
27. *Nations, U. (2014). World Urbanization Prospects: The 2014 Revision. Geneva: United Nations.*
28. *Ng, E. (2009). Policies and technical guidelines for urban planning of high-density cities—Air ventilation assessment (AVA) of Hong Kong. Building and Environment, 44, 1478–1488.*
29. *Oke, T. (1988). Street Design and Urban Canopy Layer Climate. Energy and Buildings, 103-113.*
30. *PE, P., K, K., M, M., J, G., B, P., & H, S. (2015). Urban heat island: mechanisms, implications and possible remedies. Annu Rev Environ Resour, 285-307.*
31. *Pomerantz, M., Akbari, H., Chen, A., Taha, H., & Rosenfeld, A. (1997). Paving materials for Heat Island Mitigation. Orlando: U.S. Environmental Protection Agency.*
32. *Rajagopalan, P., Lim, K. C., & Jamei, E. (2014). Urban heat island and wind flow characteristics of a tropical city. Elsevier, 159-170.*
33. *Ren, C., Yang, R., Cheng, C., Xing, P., Fang, X., Zhang, S., . . . Ng, E. (2018). Creating breathing cities by adopting urban ventilation assessment and wind corridor plan – The implementation in Chinese cities. Elsevier, 170-188.*
34. *Ren, C., Yang, R., Cheng, C., Xing, P., Fang, X., Zhang, S., . . . Ng, E. (2018). Creating breathing cities by adopting urban ventilation assessment and wind corridor plan – The implementation in Chinese cities. Elsevier, 170-188.*

35. *Shishegar, N. (2013). Street Design and Urban Microclimate: Analyzing the Effects of Street Geometry and Orientation on Airflow and Solar Access in Urban Canyons . Journal of Clean Energy Technologies.*

36. *Simscale. (2021, September 2). What is the History of CFD? Retrieved from simscale: https://www.simscale.com/docs/simwiki/cfd-computational-fluid-dynamics/what-is-cfd-computational-fluid-dynamics/*

37. *Studymafia. (n.d). CFD . Studymafia.org.*

38. *studymafia.org. (n.d). Computational Fluid Dynamics (CFD).*

39. *Su, N., Zhou, D., & Jiang, X. (2016). Study on the Application of Ventilation Corridor Planning in Urban . Elsevier, 340-349.*

40. *Sumei, L., Wuxuan, P., Hao, Z., Xionglei, C., Zhengwei, L., & Qingyan, C. (2017). CFD simulations of wind distribution in an urban community with a full-scale geometrical model. Elsevier, 11-23.*

41. *Tabada, A., & He, Y. (2018). Modeling City Patterns for Urban Ventilation: Strategies in High Density Areas of Singapore. Designing Cooler Cities, Energy, Cooling and Urban Form: The Asian Perspective, 119-135.*

42. *Tablada, A., & He, Y. (2018). Modeling City Patterns for Urban Ventilation: Strategies in High Density Areas of Singapore. Designing Cooler Cities, 119-137.*

43. *The Best Time to Visit Lumbini, Nepal for Weather, Safety, & Tourism. (n.d.). Retrieved from Champion Traveler: https://championtraveler.com/dates/best-time-to-visit-lumbini-np/*

44. *TR, O. (1982). The energetic basis of the urban heat island. QJR Meteorol Soc, 1-24.*

45. *University, G. (2022, April 19). Systematic style literature reviews for education and social sciences. Retrieved from Library: https://libraryguides.griffith.edu.au/systematic-literature-reviews-for-education*

46. *Wang, W., Yang, T., Li, Y., Xu, Y., Chang, M., & Wang, X. (2020). Identification of pedestrian-level ventilation corridors in downtown Beijing using large eddy simulation. Elsevier.*

47. *Wang, Y., Berardi, U., & Akbari, H. (2015). Comparing the effect of Urban Heat Island Mitigation Strategies for Toronto, Canada. Energy and Buildings.*

48. *Wicht, M., & Wicht, A. (2017). LiDAR-Based Approach for Urban Ventilation Corridors Mapping. IEEE JOURNAL OF SELECTED TOPICS IN APPLIED EARTH OBSERVATIONS AND REMOTE SENSING.*

49. *Wicht, M., Wicht, A., & Osinska-Skotak, K. (2017). Detection of ventilation corridors using a spatio-temporal approach aided by remote sensing data. European Journal of Remote Sensing, 254-267.*

50. *E. Willemsen, J.A. Wisse, Design for wind comfort in The Netherlands: procedures, criteria and open research issues, J. Wind Eng. Ind. Aerodyn. 95 (2007) 1541e1550.*

51. *Wong, M. S., Nichol, J. E., To, P. H., & Wang, J. (2010). A simple method for designation of urban ventilation corridors and its application to urban heat island analysis. Elsevier, 1880-1889.*

52. *Xie, P., Yang, J., Wang, H., Liu, Y., & Liu, Y. (2020). A New method of simulating urban ventilation corridors using circuit theory. Elsevier.*
53. *Zhang, Y., Zhang, X., Xu, W., & Jiao, B. (2020). A Case Study on Urban Ventilation Assessment with Local Climate Zone (LCZ) Parameters. NEPEE.*
54. *Chen, L.; Wen, Y.; Zhang, L.; Xiang, W.-N. Studies of thermal comfort and space use in an urban park square in cool and cold seasons in Shanghai. Build. Environ. 2015, 94, 644–653.*
55. *Yoshida, A.; Hisabayashi, T.; Kashihara, K.; Kinoshita, S.; Hashida, S. Evaluation of effect of tree canopy on thermal environment, thermal sensation, and mental state. Urban Clim. 2015, 14, 240–250.*
56. *Taleghani, M. Outdoor thermal comfort by different heat mitigation strategies-A review. Ren. Sustain. Energy Rev. 2018, 81, 2011–2018.*
57. *Tsitoura, M.; Theocharis, T.; Tryfon, D. Evaluation of comfort conditions in urban open spaces. Application in the island of Crete. Energy Convers. Manag. 2014, 86, 250–258.*
58. *Li, L.; XiaoQing, Z.; Lixiu, Y. The analysis of outdoor thermal comfort in Guangzhou during summer. Proc. Eng. 2017, 205, 1996–2002.*
59. *Jin, H.; Siqi, L.; Jian, K. The thermal comfort of urban pedestrian street in the severe cold area of northeast china. Energy Proc. 2017, 134, 741–748.*
60. *Mahmoud, A.H.A. Analysis of the microclimatic and human comfort conditions in an urban park in hot and arid regions. Build. Environ. 2011, 46, 2641–2656.*
61. *Morris, C. J. G., Simmonds, I., & Plummer, N. (2001). Quantification of the influences of wind and cloud on the nocturnal urban heat island of a large city. Journal of Applied Meteorology, 40(2), 169–182.*

Appendix A: IOE GC RESEARCH PAPER

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

Analysis of wind flow in Lumbini Sanskritik Municipality through ANSYS Fluent

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

[a] *Department of Architecture, Institute of Engineering*
[b] *Associate Professor, Department of Architecture, Institute of Engineering*
[c] *Assistant Professor, Department of Mechanical Engineering, Kathmandu University*
[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

Pages: 1 – 8

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 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

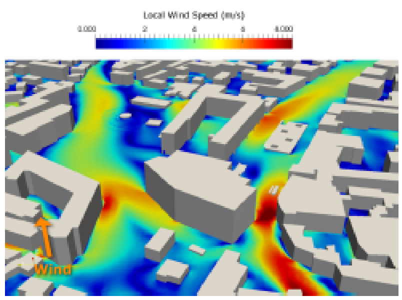


Figure 1: Wind Corridor design in CFD (fyfluiddynamics.com)

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].
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].
Many studies have been carried out on determining

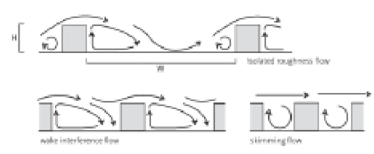


Figure 2: Type of wind flow around buildings (urbanclimate.nl)

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°2853N; 83°1633E. 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

Figure 3: Site Selection Top View (Google Earth)

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 .

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

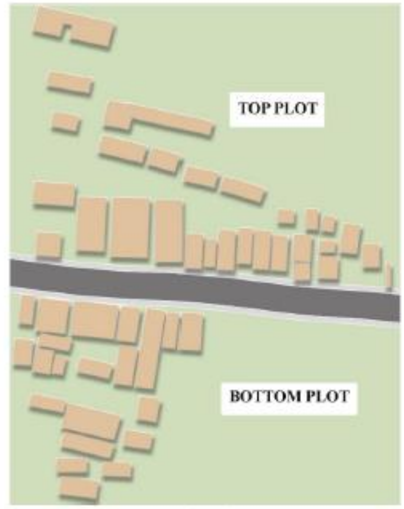


Figure 4: Site Selection

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.

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.

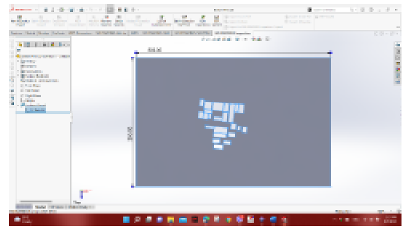


Figure 5: Solidworks Interface for modelling

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**

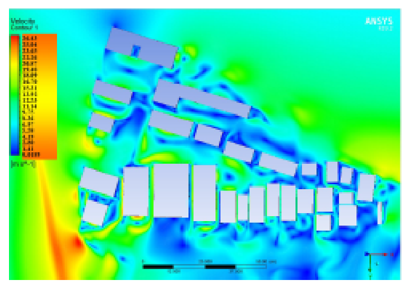


Figure 6: Velocity contour of Top Plot

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 speed of 16.5m/s between the buildings

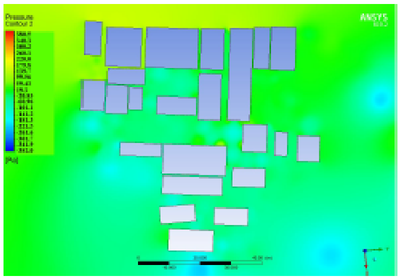


Figure 11: Pressure Contour of Bottom Plot

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

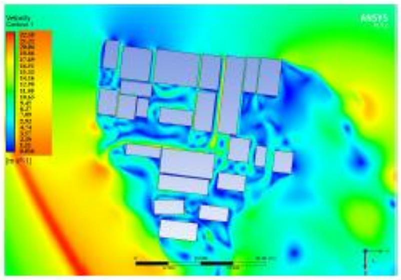


Figure 7: Velocity contour of Bottom Plot

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**

From figure 8 and 9, we can observe the wind

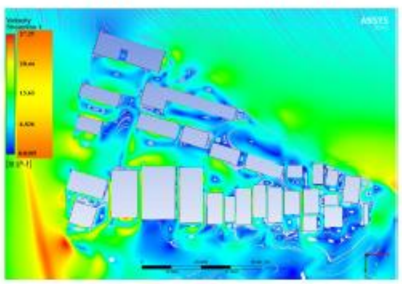


Figure 8: Velocity Streamline of Top Plot

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

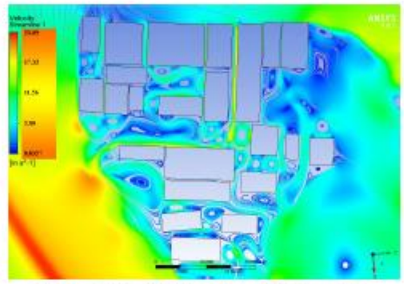


Figure 9: Velocity Streamline of Bottom Plot

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**

As we can see from figure 10 and 11, the wind

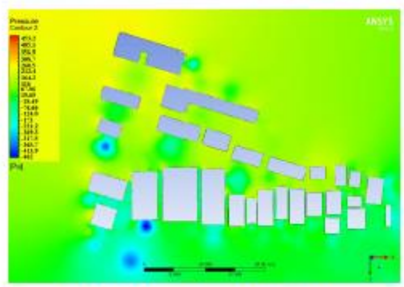


Figure 10: Pressure Contour of Top Plot

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 pressure ranges from 265 Pascal high to as low 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 pervious 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.

Acknowledgments

The authors would like to express their thanks to the Department of Architecture for permission to prepare and publish this paper.

References

[1] Plummer Morris, Simmonds. Quantification of the influences of wind and cloud on the nocturnal urban heat island of a large city. *Journal of Applied Meteorology*, 40, 2001.

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

[3] Wisse Willemsen. Design for wind comfort in the netherlands: procedures, criteria and open research issues. 2007.

[4] Tominaga Mochida Kubota, Miura. 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.

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

[6] Qian Sun Wang Gu, Fang. Spatial planning for urban ventilation corridors by urban climatology. *Ecosystem Health and Sustainability*, 2020.

[7] Cheng Xing Fang Zhang Ng Ren, Yang. Creating breathing cities by adopting urban ventilation assessment and wind corridor plan – the implementation in chinese cities. *Elsevier*, (170-188), 2018.

[8] Shishegar. Street design and urban microclimate: Analyzing the effects of street geometry and orientation on airflow and solar access in urban canyons. *Journal of Clean Energy Technology*, 2013.

[9] Johansson. Influence of urban geometry on outdoor thermal comfort in a hot dry climate: A study in fez, morocco. *Building and Environment*, 2006.

[10] Al-Rais Al-Sallal. Outdoor airflow analysis and potential for passive cooling in the traditional urban context of dubai. *Renewable Energy*, 2011.

[11] Hao-Xionglei Zhengwei Qingyan Sumei, Wuxan. Cfd simulations of wind distribution in an urban community with a full-scale geometrical model. *Elsevier*, pages 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].

Appendix B: IOE GC Acceptance Letter

त्रिभुवन विश्वविद्यालय
Tribhuvan University
इन्जिनियरिङ अध्ययन संस्थान
Institute of Engineering

डीनको कार्यालय
OFFICE OF THE DEAN

GPO box- 1915, Pulchowk, Lalitpur
Tel: 977-5-521531, Fax: 977-5-525830
dean@ioe.edu.np, www.ioe.edu.np
गोश्वारा पो.ब. नं- १९१५, पुल्चोक, ललितपुर
फोन- ५५२१५३१, फ्याक्स- ५५२५८३०

Date: September 14, 2022

To Whom It May Concern

This is to confirm that the paper titled "*Analysis of wind flow in Lumbini Sanskritik Municipality through ANSYS Fluent*" submitted by **Biplav Pokhrel** with Conference ID **12201** has been accepted for presentation at the 12th IOE Graduate Conference being held in October 19 – 22, 2022 at Thapathali Campus, Kathmandu.

Khem Gyanwali, PhD
Convener,
12th IOE Graduate Conference

Appendix C: Plagiarism Report

Biplav Pokhrel_076MSEEB006.pdf

ORIGINALITY REPORT

16%

SIMILARITY INDEX

PRIMARY SOURCES

	Source	Type	Words	%
1	www.slideshare.net	Internet	511 words	1%
2	ccsenet.org	Internet	377 words	1%
3	uscupstate.libguides.com	Internet	259 words	1%
4	elibrary.tucl.edu.np	Internet	252 words	1%
5	www.mdpi.com	Internet	245 words	1%
6	library.pdpu.ac.in:8080	Internet	235 words	1%
7	web5.arch.cuhk.edu.hk	Internet	215 words	1%
8	themechanicalengineering.com	Internet	209 words	1%