International Journal of Applied Engineering Research ISSN 0973-4562 Volume 19, Number 1 (2024) pp. 1-6

New Structures and Designs to Mitigate Building Wind-Related Disasters Based on Bernoulli's Equation

Yuri Bae[1*], **Euihyeon Hwang**[1*], **Jihye Park**[1*]

[1]*Department of Energy Engineering, Korea Institute of Energy Technology (KENTECH),*
Naju-si, Jeonnam 58330, Republic of Korea.

Abstract

When the 9th typhoon 'Maisak' landed in 2020, severe damage occurred to the skyscraper in Busan, South Korea [1]. As a result of analyzing the incident, the wind of the typhoon was strengthened due to the building wind that created a strong gust around the building, resulting in damage.

Recently, changes have also occurred in typhoons due to the influence of global warming [2]. In the case of typhoons, the number of occurrences decreased, and the intensity increased [3]. As the typhoon becomes stronger, the strength of the building wind also increases, and it can cause more damage to either buildings or pedestrians near buildings.

This paper aims to control wind flow by installing additional structures between buildings or on the ground and to see how they affect the building wind. Also, check what changes can be made while controlling the shape and location of the building and evaluate it by referring to the government guidelines [4].

Keywords: Building Wind Mitigation, NACA Airfoil Model, CFD Simulations, Navier Stokes Equation

INTRODUCTION

The increased construction of high-rise buildings and the rising frequency of extreme weather events have led to the emergence of building wind as a new form of urban pollution, causing concerns over the social damage it brings.
The current considerations for improving building-induced wind are summarized in

. The whole current solution addresses the separated wind flow and downdraft wind, not a valley wind that directly affects the pedestrian. Also, most of the solutions need to be considered before the construction.

Table 1. Current Solutions of Building Wind [4]

Method	Principle	Goal	Pre-construction design / Additional structures
Changing the shapes of edges	Edges divided into stairs reduce the interference phenomenon of boundary layer separation by the interaction of flows	Reduce separated wind flow	Pre-construction design
Surface roughness configuration	Create smaller eddies to weakness surrounding winds	Reduce separated wind flow	Pre-construction design
Design with large low-rise areas	The fodium design create the differential wind speeds due to the difference in planar surface area between the low-rise and high-rise sections.	Prevent downdrafts from reaching pedestrian areas.	Pre-construction design
Setback design	Create the differential wind speeds due to the difference in planar surface area between the low-rise and high-rise sections	Reduce separated wind flow and downdraft wind.	Pre-construction design
Partial building floor hollowing	Make a hole for wind flow	Reduce separated wind flow and downdraft wind.	Pre-construction design
Wind break fence.	Block the wind flow with obstacles	Reduce direct separated wind flow and downdraft wind.	Additional structures

The current solutions are not appropriate for addressing contemporary problems within a completely constructed urban city. Therefore, we aim to address the issue of wind-induced discomfort at a human pedestrian level within existing buildings. To mitigate the problem of building-induced wind speeds, we propose the installation of additional structures. The objective is to alleviate these concerns by implementing additional architectural elements.

METHODOLOGY

Bernoulli's Equation [5]

We assume the following to derive Bernoulli's equation for the ideal fluid.

Steady flow: The velocity of fluid does not change with time at any fixed point.

Incompressible flow: The density of fluid does not change, which means it has a constant and uniform value of density.

Non-viscous flow: There is no viscosity. Thus, an object can move at a constant speed through the fluid.

Irrotational flow: It is assumed that the flow is irrotational.

$$p_1 + \frac{1}{2}\rho v_1^2 + \rho g y_1 = p_2 + \frac{1}{2}\rho v_2^2 + \rho g y_2 \qquad (1)$$

y_1 represents the elevation, v_1 denotes the speed, and p_1 represents the pressure of the fluid entering at the left. y_2 denotes the elevation, v_2 is the speed, and p_2 represents the pressure of the fluid emerging at the right. Equation (1) can be written as follows:

$$p + \frac{1}{2}\rho v^2 + \rho g y = C \qquad (2)$$

Where C is a constant. Equations (1) and (2) express Bernoulli's equation. According to Bernoulli's Equation, the speed of wind increases when it nears a narrow area because the higher pressure behind it accelerates it. On the other hand, the speed of wind decreases when it nears a wide area because the higher pressure ahead of it decelerates it.

The objective of Solving the Problems within Building Wind

We identify the reason for building wind mathematically. The speed of the wind increases rapidly between two buildings due to the Venturi effect. The Venturi effect is that the speed of the fluid is high when it passes a narrow area, and the speed of the fluid is low when it passes a wide area. Bernoulli's Equation explains this effect mathematically. Therefore, the cause of building wind is explained by Bernoulli's Equation.

We suggest an appropriate additional structure by using differential equations and CFD to address the wind that directly affects pedestrians, unlike current solutions.

Constraints

(i) The structure should be designed to maintain stability against wind loads.

(ii) The cost may vary depending on the material of the structure, and therefore, it should be considered as a factor.

(iii) As the proposed additional structure is intended to block the building wind between existing buildings, it should be of an appropriate size for the space between the existing buildings.

Bernoulli's Equation

In Figure 1, the speed (v_1), atmospheric density (ρ_1), atmospheric pressure (p_1), and relative height (y_1) from the horizontal reference surface are set for the initial state. After that, a point between buildings is taken as point 2, and the wind speed, atmospheric density, air pressure, and relative height are determined as the second constant. Due to the venturi effect, the space between buildings is narrower than the outer space, so the pressure between buildings is lower than the atmospheric pressure, resulting in a faster wind speed even at the same relative height ($y_1 = y_2$).

If additional structures are installed (Point 3) to increase the relative height by changing the direction of wind blowing ($y_1 = y_2 < y_3$), or to reduce wind speed ($v_2 > v_3$), pressure changes between buildings can be reduced, which can affect wind speed changes in the overall space between buildings. Accordingly, the amount of increase in wind speed due to the building wind is reduced by using the influence of the additional structure.

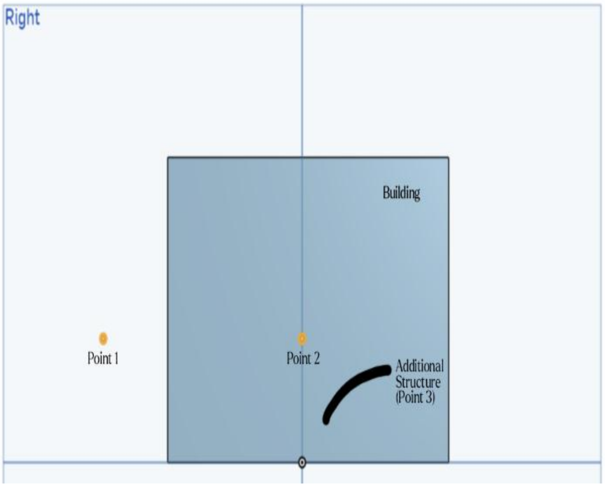


Figure 1. Initializing Point of Building Wind Explained by Bernoulli's Equation

Navier-Stokes Equation [6]

In this study, to identify changes in wind speed and pressure due to building wind and the degree of effects by additional structures, Computational Fluid Dynamics (CFD) is used to figure out wind speed and pressure around 10m above the ground, which affects pedestrians on the road. Navier-Stokes equation is a basic governing equation that expresses fluid motion mathematically by numerical analysis using a CFD model [7]. Navier-Stokes equation is expressed as follows:

$$\rho \frac{\partial}{\partial x}\vec{v} = -\vec{\nabla}P + \rho\vec{g} + \mu\nabla^2\vec{v} \qquad (3)$$

which $\vec{v}$ is the velocity of the fluid, $\vec{g}$ is the gravitational acceleration, ρ is the density, P is the pressure, and μ is the viscosity coefficient.

Installing additional structures is expected to affect the degree of compression, pressure change, and speed of the gas around the structure.

Numerical Analysis Using Computational Fluid. Dynamics

Onshape [8] and SimScale [9] are used to figure out the interaction between the building wind and additional structures. Onshape is a cloud-based computer-aided design (CAD) software that enables users to create, edit, and collaborate on 3D models and designs. Through this program, 3D models of buildings and additional structures are made for CFD analysis. SimScale is a cloud-based simulation platform specifically designed for conducting computational fluid dynamics (CFD) analysis. Loading CAD files of building and wind-block structures to SimScale, calculating the fluid, and analyzing the effects of additional structures on the speed and pressure of the wind.

Proposing Additional Structures

We compared our proposed model, inspired by the net wall for dispersing fluid and the NACA airfoil, with the conventional general wall structure previously used as an additional structure. Our goal was to minimize the impact on pedestrians from strong winds blowing between buildings by reducing the velocity using three different structures, as detailed in Table .

Preparation to Simulate Structure

Prior to running the simulations, we created a 3D model scaled to match the real-world dimensions for accurate representation. Figure 2 suggests the scale that we addressed.

Also, to figure out the effects of additional structures on building wind, we cited the AIJ case F project [10] demonstrating the relative wind velocity around the urban area. Following documentation by Simscale [11] and public simulation generated by Dlynch [12], we selected and referred to analysis type, solver type, fluid information, and boundary setting from references.

All models are generated by Onshape [8]. The modeling file was imported to SimScale CFD.

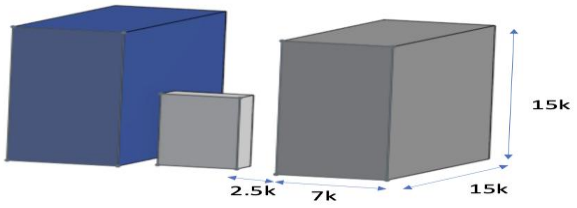


Figure 2. The Determined Scale of the Building and Additional Structures

3D Modeling

General Wall

In the report of [4], it offers a general wall structure as the only solution for mitigating building wind by additional structure. Since the general wall structure blocks the pedestrian pathways, pedestrian walkability is low. The scale description is in Table .

Utilize the Net Model

The net model disperses the energy of the wind and controls the fluid flow, reducing the wind speed without directly affecting pedestrians. However, the pedestrian walkability is low, like a general wall. The scale description is in Table

Utilize NACA Airfoil Model

The wing model disperses and makes a change of fluid that leads to availability for controlling the wind velocity. To perform the simulation, we made building structures as 0.15m×0.07m×0.15m (H×W×L) and distance between as 0.025m for only building, single wall, perforated wall, and single wing. Building structures are every single body generated by the Extrude function. In the case of multiple wing simulation, the building's length extends to 0.55m. When designing wings, we followed the NACA airfoil with 4 digits [13]. According to Table , the first digit describes the maximum camber value by percentage, the second digit expresses the position where the maximum camber is placed by tenths, and the remaining two digits denote the maximum thickness by percentage. Those properties are calculated by multiplying the length of the chord of the wing. This report uses NACA 2412 airfoil for wing structures, and the airfoil has a maximum camber of 2%, where 40% departs from the leading edge with a maximum thickness of 12% of the chord.

Table 2. Comparison of the additional structures

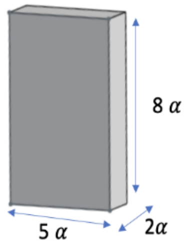
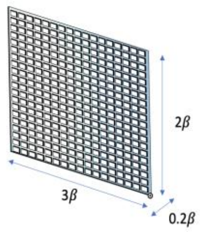
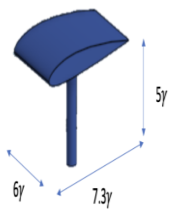
Scale	Structure Description	Pedestrian Walkability
	General wall	Low: necessary to go around
	Utilize net model	Low: necessary to go around
	Utilize NACA airfoil model	High: available to go through

Table 3. Description for NACA Airfoil with Four-digit Series

(Properties are expressed in a fraction of length of the chord)

Digit	X	X	X	X
Definition	Maximum camber	Distance of maximum camber from the airfoil leading edge	Maximum thickness of the airfoil	
Fraction	In percentage (X %)	As tenths (X0 %)	As percentage (XX %)	

Process of Simulate the Structures using Computational Fluid Dynamic

Simulation Materials

In this study, the analysis type is that the steady-state incompressible flow for applied wind is slower than 0.3 Mach (almost equal to 100m/s), considered the wind speed of a middle-size typhoon [14]. The simulation uses the K-omega SST (Shear Stress Transport) turbulence model to analyze the flow region for each situation (without structures, single wall, single perforated wall, single-wing, multiple wings), which covers the building and structure generated by the external flow region tool. The external flow region is developed to cover the outside of the two buildings with little space for wind escape after crashing the vertical wall of the building. To ensure the convergence of the simulation, we gave enough front, back, and upper spaces in the flow region. After setting the flow region, delete CAD 3D modeling inside of the flow region.

Boundary Conditions

Likely to the AIJ case simulation [12], for the inlet boundary, we set a fixed value and arranged the wind direction. The outlet boundary is sat to pressure outlet by 0 Pa, opposite face to inlet boundary. The bottom faces and other side faces are wall and slip condition, except for the bottom face to no-slip to perform as ground. Building walls and structures are also set to no-slip wall condition.

Mesh

Using a standard algorithm, we generated enough quality of mesh that satisfies the criteria (Overall quality between 0.035 to 1.0). However, there was a weak convergence value, and this problem means that this simulation needs to explain the effect on the real world correctly. To solve this problem, we generated mesh by Hex-dominant algorithm, and mesh quality criteria were the same as the standard algorithm. This solution helps to improve convergence value.

Simulation Run

Select the previous steps for running the simulation and add a new simulation run. Checking Convergence plots, evaluate the convergence value of simulation from domain, inlet, outlet, and residual. If the large scale of oscillation has occurred in plots, it could mean some problem occurred, and therefore, simulation requires aids such as refinement on the mesh. After finishing the simulation, post-processing of results could generate visual figures to explain the results carefully.

RESULT

Interpretation of Applied Navier-Stokes Equation

Equation (3) is a nonlinear partial differential equation that describes the motion of fluids. Navier- Stokes equation's solution is the velocity of the fluid. Computational fluid dynamics (CFD) analysis is done by numerical method to reach the approximate solution of Equation (3). We identify the velocity change by addressing each additional structure between buildings.

Evaluation of Additional Structures

We conducted CFD simulations using three different structures. Firstly, the general wall structure, as observed in Figure 3, strengthened the incoming wind speed. Secondly, Figure 4 demonstrates that utilizing the net model structure effectively disperses the incoming wind into weaker gusts along the pedestrian pathway, significantly reducing wind speed. Thirdly, when placed singularly, the utilization of the NACA airfoil model structure, which is in Figure 5, showed a minimal effect on reducing wind speed as the wind added momentum by passing around the central pillar.

Upon integrating the CFD results of the three structures and comparing them solely based on wind speed scale, the structure utilizing the net model, which maintained wind speeds of 13m/s to 35m/s in the rear space and pedestrian walkway, exhibited the most effective reduction in wind speed.

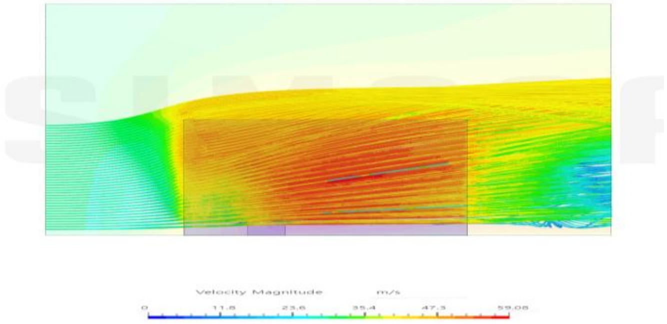


Figure 3. CFD Result of General Wall Structure

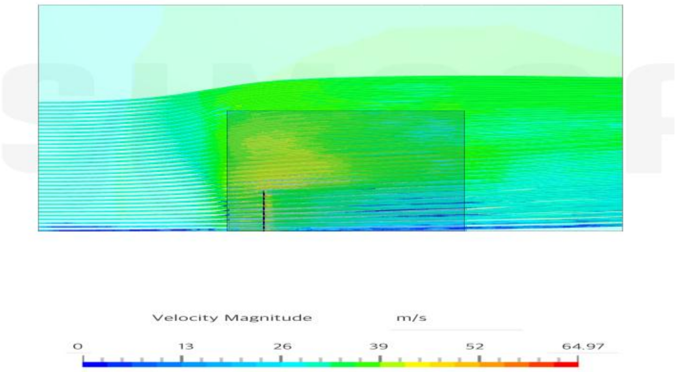


Figure 4. CFD Result of Utilizing Net Model Structure

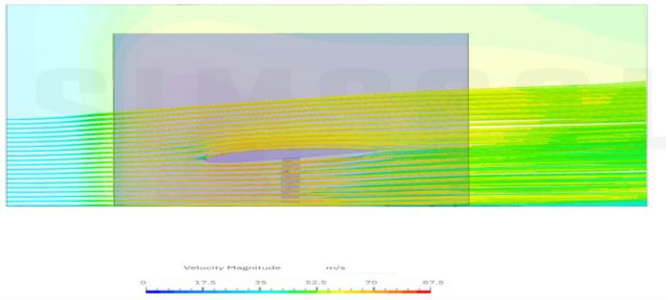


Figure 5. CFD Result of Utilizing NACA Airfoil Model Structure

Final Proposal to Mitigate Building Wind-Related Disasters

As evident from the results, the structure with lower pedestrian walkability demonstrated the most effective outcomes. However, our research objective was to propose a structure that improves pedestrian walkability while reducing wind speeds caused by building-induced winds. Therefore, we analyzed the reasons behind the limited effectiveness of the Utilizing NACA airfoil model structure. It was determined that the wind intensification due to the wind passing around the central pillar was the contributing factor.

To address this issue, we suggest continuous attenuation that can address the passing around wind from the central pillar. Finally, we proposed a solution using multiple Utilizing NACA airfoil model structures installed, similar to the configuration shown in Figure 6, which would not obstruct pedestrian movement, such as streetlights. Subsequently, we obtained CFD results, as shown in Figure 7.

Ultimately, we achieved wind speed reductions ranging from 15m/s to 30m/s within the pedestrian pathway where the structures were installed. This proposal successfully mitigated the wind speeds caused by building-induced winds while minimizing interference with pedestrian movement.

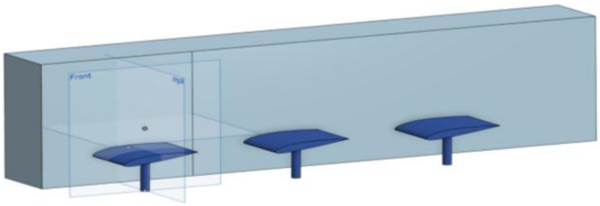


Figure 6. 3D Modeling of the Final Proposal

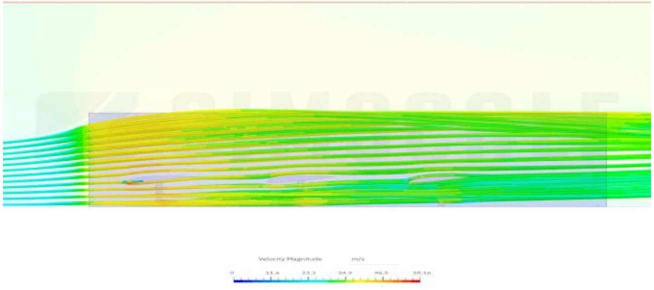


Figure 7. CFD Result of the Final Proposal

DISCUSSION

Building wind-related disasters is a severe problem for pedestrians. The strong wind that is generated by crossing through the buildings leads to disaster. Thus, this report proposes three structures to mitigate the building wind-related disasters between two buildings. One is a general wall structure, another is a utilizing net model structure, and the other is a utilizing NACA airfoil model structure. With 3D modeling of these three models, results about the effect on mitigating building wind-related disasters in each model are obtained in CFD simulations.

These three models are additional structures that change the wind flow with obstacles, unlike other methods like changing the shapes of edges, surface roughness configuration, and so on. Additional structures have the advantage that they are expected to affect the degree of compression, pressure change, and speed of the gas around the structure without changing the original structure of buildings.

Based on the results of simulating three proposal models, multiple utilizing NACA airfoil model structures which supplement other models by installing additional structures are proposed. The general wall structure is expected not to mitigate building wind-related disasters well, and it has low pedestrian walkability. Utilizing the net model structure is expected to reduce building wind-related disasters well compared to the final proposal, but it still has low pedestrian walkability.

This study has a limitation in multiple utilizing NACA airfoil model structures. The size of these structures varies from surrounding buildings, so these structures are designed diversely depending on the circumstances. The versatility will be enhanced if we consider the various sizes of these structures in further research. Unlike another current model, multiple utilizing NACA airfoil model structures succeed in well-mitigating building wind-related disasters and have high pedestrian walkability. Also, these structures have high efficiency of installing structures, because they can be installed on the streetlights. Since our proposal is available for installation in streetlights, the proposal has a sense of aesthetics to install in the middle of the pathway. Current methods to mitigate building wind-related disasters are unsuitable for completely conducted buildings. Therefore, the key solution to address the problem we faced was installing a new additional structure. This study is meant to suggest a key solution that is genuinely effective for lowering the wind velocity at the pedestrian level and efficient for installation that did not previously come out.

REFERENCES

[1] J. Kim, B. Kang, Y. Kwon, S. Lee, and S. Kwon, “A review on the building wind impact through on-site monitoring in Haeundae Marine City: 2021 12th Typhoon Omais Case Study,” Journal of Ocean Engineering and Technology, vol. 35, no. 6, pp. 414–425, 2021.

[2] D. Seol, “Global warming and trends of Typhoon Variation,” Journal of Korean navigation and port

research, vol. 34, no. 6, pp. 453–458, Aug. 2010.

[3] K. OOUCHI, J. YOSHIMURA, H. YOSHIMURA, R. MIZUTA, S. KUSUNOKI, and A. NODA, “Tropical cyclone climatology in a global-warming climate as simulated in a 20 km-mesh global atmospheric model: Frequency and wind intensity analyses,” Journal of the Meteorological Society of Japan. Ser. II, vol. 84, no. 2, pp. 259–276, 2006.

[4] “Wind Environment Analysis and Building Wind Reduction Guidelines”, 30-Nov-2021.[Online]. Available: http://www.molit.go.kr/USR/policyData/m_34681/dtl.jsp?search=&srch_dept_nm=&srch_dept_id=&srch_usr_nm=&srch_usr_titl=Y&srch_usr_ctnt=&search_regdate_s=&search_regdate_e=&psize=10&s_category=&p_category=&lcmspage=1&id=4593. [Accessed: 07-May-2023].

[5] D. Halliday, J. Walker, and R. Resnick, Principles of Physics. Singapore: Wiley, 2014

[6] Y. A. Çengel and J. M. Cimbala, “Chapter 9 Differential Analysis of Fluid Flow,” in Fluid Mechanics: Fundamentals and Applications, Singapore: McGraw-Hill Education, 2020, pp. 443–518

[7] Y. A. Çengel and J. M. Cimbala, “Chapter 15 Introduction to Computational Fluid Dynamics,” in Fluid Mechanics: Fundamentals and Applications, Singapore: McGraw-Hill Education, 2020, pp. 885–945

[8] Onshape. (2015). "Onshape." [Online]. Available: https://www.onshape.com/. Accessed: May 21, 2023.

[9] Simulation software: Engineering in the cloud,” SimScale, https://www.simscale.com/ (accessed Jun. 7, 2023)

[10] “Toshi No Kazekankyo Yosoku No Tame No CFD Gaidobukku.” Tokyo: Architectural Institute of Japan, 2020.

[11] “Pedestrian wind comfort: Aij case F: Validation case,” SimScale, https://www.simscale.com/docs/validation-cases/pedestrian-wind-comfort-aij-case-f/ (accessed Jun. 7, 2023).

[12] Dlynch, “‘AIJ case F’ simulation project by dlynch,” SimScale CAE Forum, https://www.simscale.com/forum/t/aij-case-f-simulation-project-by-dlynch/82882 (accessed Jun. 7, 2023).

[13] D. S. Körpe and I. H. Güzelbey, “NACA four-digit airfoil series optimization: A comparison between genetic algorithm and Sequential Quadratic Programming,” Journal of Mechanical Science and Technology, vol. 37, no. 5, pp. 2375–2382, May 2023. doi:10.1007/s12206-023-0414-9

[14] “Classification of Typhoons,” Korea Meteorological Administration, https://www.weather.go.kr/w/typhoon/basic/info3.do (accessed Jun. 7, 2023).