# Design Optimization for Resilient Building Design in Coastal Environments

Soo Jeong Jo[1], Dongkeun Lee[2], Farzaneh Oghazian[1]
*School of Architecture, Louisiana State University[1]*
*Department of Civil and Environmental Engineering, Southern University and A&M College[2]*

**Abstract**

THE PRESENT STUDY AIMS TO INVESTIGATE THE IMPACT OF THE BUILDING FORM ON THE SURROUNDING WIND FLOW AND SUPPORT RESILIENT FORM FINDING FOR DESIGNING BUILDINGS IN HURRICANE-PRONE AREAS. TO ACHIEVE THIS, AN IMMERSIVE CASE STUDY WAS CONDUCTED FOR A 1,350-SF SINGLE FAMILY HOUSE IN A COASTAL AREA IN LOUISIANA. THE FORM OF THE ROOF WAS THE TESTED DESIGN PARAMETER SINCE THE ROOF UPLIFT IS ONE OF THE MAJOR HURRICANE DAMAGES IN THE SELECTED AREA. COMPUTATIONAL FLUID DYNAMICS (CFD) SIMULATIONS WERE PERFORMED TO OPTIMIZE THE ROOF FORM TO MINIMIZE THE POSSIBILITY OF WIND UPLIFT. THROUGH THIS RESEARCH, THE AUTHORS TRIED TO RESPOND TO THE FOLLOWING QUESTION: HOW CAN THE ROOF FORM OF A BUILDING BETTER RESPOND TO EXTREME WIND CONDITIONS? THE PRESENT STUDY MAY ASSIST ARCHITECTURAL DESIGNERS IN DEVELOPING A MORE RESILIENT BUILDING FORM IN THE EARLY STAGES OF DESIGN AND ADVANCE THE UNDERSTANDING OF ENVIRONMENTAL ISSUES. THIS WORK WILL ULTIMATELY PROMOTE MORE SUSTAINABLE AND ENVIRONMENT-CONSCIOUS BUILDING DESIGN.

Resilient building design has become an urgent need and critical issue in the building industry as the acceleration of climate change has increased natural disasters worldwide. The impact on coastal areas may be more frequent hurricane and storm surge events with a higher wind and water velocity, such as Hurricane Katrina, which destroyed about 300,000 houses for one million people [1]. In light of this background, the present study aims to investigate the impact of building form on the fluid flow around buildings and establish a resilient form-finding framework for building design in coastal environments. Through this study, the authors question how the formative process of architectural design can better respond to extreme conditions, such as high-velocity wind events. To answer this question, an immersive case study was conducted for an imaginary 1,350-sf single-family house located in Grand Isle, Louisiana.

**Background**

Due to its coastal location on the Gulf of Mexico, Grand Isle has been hit by many hurricanes historically, such as Hurricane Katrina with a wind speed of 282 km/h in 2005, and Hurricane Gustave with a wind speed of 169 km/h in 2008 [2]. Compared to buildings in land areas, ones in coastal areas must be designed and constructed more rigorously as high-velocity winds may cause buildings to sway, which may lead to structural collapse. Severe wind gusts and storm surges substantially influence the design of buildings, especially in hurricane-prone zones. Responding to this, the town of Grand Isle adopted new building codes, such as break-away or flow-through walls requirement and ground-level electric wires prohibition [2], from the International Code Council [3] and the Federal Emergency Management Agency's Code of Federal Regulations [4] to minimize further damage in future hurricane events.

As with these efforts, architects and builders in hurricane-prone zones need reliable structural designs to mitigate the effects of strong winds and surges. For this purpose, research on structural resiliency against high-velocity winds and storm surges has been extensively conducted. For instance, Haddara and Soares studied the calculation of wind loads on offshore structures and ships using various methods and proposed an equation to estimate wind loads based on a neural network technique [5]. Also, Simiu and Yeo explored modern structural design considering wind effects [6], and Hatzikyriakou and Lin simulated storm surge waves to estimate structural vulnerability [7]. They developed fragility curves by statistically relating the simulated hazard variables to surveyed building damages in order to quantify structural vulnerability. Androulidakis et al. focused on extreme events in the Mediterranean Sea, especially the IANOS Medicane, which was one of the most devastating storms in the Mediterranean Sea [8]. They comprehensively investigated the characteristics of the storm surge and the coastal inundation, as well as the marine weather conditions based on hydrodynamic ocean simulations in combination with field and satellite data.

Recent advancements in computational tools and digital fabrication have opened new possibilities for designing and constructing more resilient and inspiring building typologies. Specifically, small-scale houses and facilities, generally handled prescriptively due to limited budget resources, may benefit from these technological advancements, which entail democratizing customized design. The J-House in New Orleans by Ammar Eloueini [3] serves as one such example. Additionally, developments in materials such as laminated timber have created unconventional forms. Research on robotically fabricated timber structures demonstrates how these advancements contribute to irregular roof geometries [4]. Reinforced concrete has historically been a key material for constructing thin-shell structures and double-curved surfaces. With the advent of 3D-printed concrete and mass production techniques, these forms are poised to re-emerge in architecture. *Essential Home*, a vault-shaped roof and floor system proposed by the Norman Foster Foundation and Holcim, is an example of a temporary settlement during the crisis [5]. Studies have been conducted to identify traditional dome-like shapes that can be mass-produced using 3D printing techniques for small residential buildings without the need for reinforcement [6]. HiRes Concrete Slab, made by researchers at ETH Zurich, is another example of thinking about the roof system beyond the limitations of conventional design [7]. While research in this area is ongoing, further investigation into the applicability of these systems under varying environmental conditions, such as wind forces, is essential to utilize the aforementioned technology for addressing structural resiliency issues in coastal areas.

**Methodology**

To explore the impact of roof design on structural stability in hurricane events, a simulation study was conducted for an imaginary house in an open field in Louisiana. The dimension of the 1,350-sf house was 13.7 m (45 ft) x 9.1 m (30ft), and the minimum height was 3.9 m (12'-10"). Based on historical information on the maximum wind speed during hurricane events, the authors assumed scenarios with various roof design possibilities and performed CFD simulations, employing a commercial code, Autodesk CFD. The baseline was a flat roof, which branched into the following five roof design options: 1) conventional gabled roof, 2) extended gabled roof, 3) shed roof, 4) concave roof, and 5) convex roof. The tested building geometry and roof design parameters, expressed as cross sections, are shown in Figure 1. The left side in the section was assumed to be the prevailing wind direction.

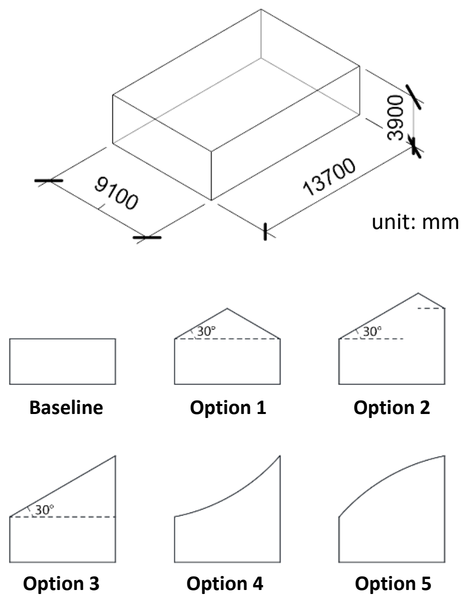


*Fig. 1. Tested building dimension (top) and the cross sections of the tested roof design parameters (bottom).*

*Computational Domain and Boundary Condition*

The references to set up the computational domain for the simulation were as follows: the best practice guideline by Franke et al. [8], the Architectural Institute of Japan (AIJ) guideline by Tominaga et al. [9], the decision-making framework for outdoor airflow simulations utilizing a case study of the Eindhoven University campus by Blocken et al. [10], the framework to apply local climate data to airflow simulations in CFD by Wu and Kriksic [11], the analysis of the commonly used parameters for urban-scale simulations by Blocken and Stathopoulos [12]. The guidelines suggest adding extra space in the computational domain for outdoor simulations, and the recommended distance for downstream direction, 15H (where H = building height), is longer than other areas. The side areas are recommended to be a minimum of 5H. Therefore, the total width of the computational domain is to be 10H + the width of the building. The windward area is recommended to be a minimum of 3H; the height of the domain is recommended to be a minimum of 6H, including the building height [13]. The computational domain (wind tunnel) based on this information is shown in Figure 2.

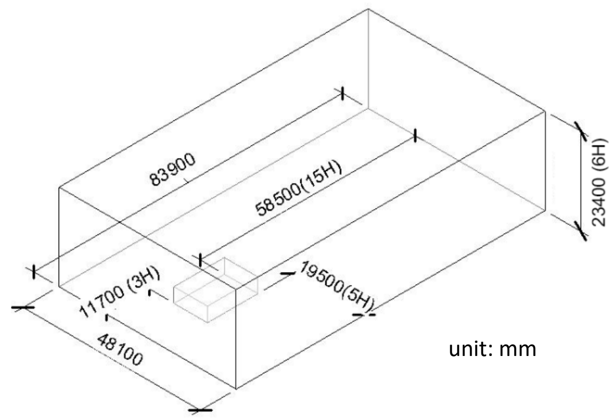


*Fig. 2. The computational domain (wind tunnel) dimension for the simulation.*

For the boundary condition, the incoming wind's velocity was assumed to be 67m/s (150 mph) based on the measured maximum velocity of Hurricane IDA, occurred in 2021 [14]. It should be noted that a logarithmic law or power law that can be applied to a typical wind velocity profile, was not employed for this study due to the super-gradient wind region of hurricane [15, 16]. The temperature was set at 30°C, and humidity was set at 75% based on the climate data of the site in hurricane season.

*Solver settings*

The steady-state 3D Reynolds-Averaged Navier-Stokes (RANS) approach, which is known for efficiency and acceptable accuracy, was applied to the simulation [8, 17]. For determining the Reynolds stresses, the K-epsilon turbulence model was used. This model is a widely used two-equation eddy-viscosity model, optimized for free flow, based on the two equations on the turbulent kinetic energy, K, and the turbulent dissipation rate, epsilon [18]. For the calculations of the equations, the Segregated Solver approach was used. This approach solves each governing momentum equation separately, thus, only one degree of freedom is calculated each time. In addition, it skips certain equations depending on the condition and reduces time for solving similar equations. Therefore, the Segregated Solver approach may reduce the required computer memory [19]. For coupling of pressure and velocity, this approach employs the Semi-Implicit Method for Pressure Linked Equations (SIMPLE) algorithm that has been widely used by researchers since the first development by Patankar and Spalding (1972). For the viscous terms and convection terms of the governing equations, second-order discretization schemes were employed. The second-order schemes are generally considered more accurate compared to first-order schemes and recommended for triangular and tetrahedral grids. Convergence was obtained with the automatic convergence function when the instantaneous convergence curve slope reached 0.001, the time-averaged convergence curve slope and concavity reached 0.032, and the field fluctuation reached 0.0001.

*Meshing and Sensitivity Analysis*

Sizing a computational grid is a critical process in CFD simulations as the size of the grid cell highly impacts the memory requirement and the accuracy of the simulation. The cells should be sufficiently fine to capture the essential phenomena and avoid discretization errors while the number of cells should be limited to ensure acceptable memory requirement and processing time *[8]*. Studying grid sensitivity allows users to find the optimum size of the grid satisfying the two aforementioned criteria. Therefore, the grid sensitivity analysis was conducted for the present study. In the grid sensitivity study for ACFD, the finite element method (FEM) for tetrahedral cells was used for the computational grid generation to ensure a continuous solution and simplify the governing equations [19]. The grid size for the simulation was determined based on the sensitivity study using four mesh resolutions. For the sensitivity study, a wind tunnel test by Karava et al. [20] was replicated as shown in Figure 3.

The iterations started with automatic sizing, and the mesh size was refined by 0.7 until the difference from the previous result reached around 5%. The initial number of cells after the automatic sizing was 46,000, and the cell number was increased as follows: 84,000 cells, 130,000 cells, and 260,000 cells. For comparing the results, the air velocity distribution along the section was observed. Figure 4 illustrates the air velocity distributions for each

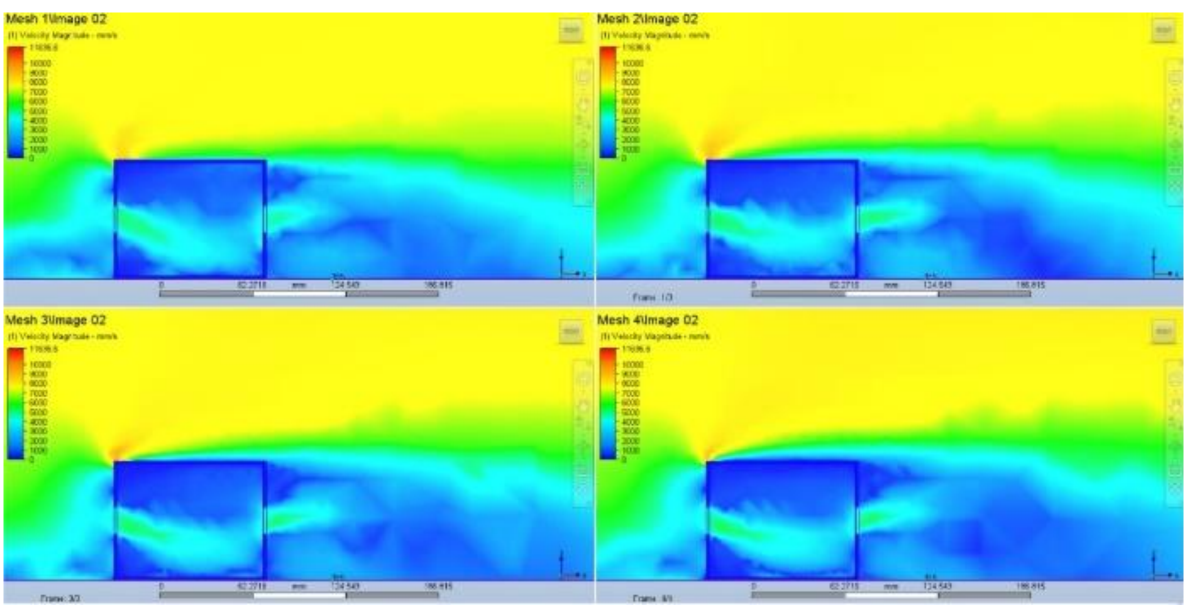


*Fig. 3. Grid sensitivity study - velocity contour plots.*

mesh refinement level. The differences of the simulation results between 130k cells and 260 k cells reached less than 5%, therefore, 130,000 cells were considered a reasonable mesh size. The acceptable range, which was less than 5%, was determined based on the literature [21, 22] and the ACFD educational resources [19].

**Results and Discussions**

Figure 5 shows the simulation results - the profiles of the air pressure on the roof surface along the center axis of the buildings. Although the simulation tool generates 3-dimensional results, these section views were selected to avoid confusion in reading quantitative data. The prevailing wind direction of the site during the hurricane season, from August to November, is northeast, and the left side of the computational domain was assumed to be the prevailing wind direction. The blue areas indicate the zones affected by the negative pressure, and darker blue areas indicate the zones with a stronger negative pressure that implies a higher risk of wind uplift. The result showed that conventional gable roof designs (Option 1 and Option 2) had a higher risk of wind uplift on the top of the leeward side of the roof. Although the intensity of the negative pressure on the flat (baseline) roof was not as strong as the gabled roof options, it had a wider roof area affected by the negative pressure from the incoming wind. Compared to the previous options, the shed roof design (Option 3) minimized negative pressure and the affected area. Applying a curve to this and making a concave roof design (Option 4) could further reduce the negative pressure and the affected roof area. However, a convex roof (Option 5) was less effective than Option 3 and Option 4 in reducing the negative pressure and the affected roof area, although it was still more effective than Option 1 and 2. Overall, the highest negative pressures of 47.5 psf and 47.4 psf were found in Option 1 and Option 2, respectively. Option 4 had the lowest negative pressure of 11.8 psf. The maximum negative pressure (-11.8 psf) in Option 4 was 75% smaller than that (- 47.5 psf) in Option 1. The practical implication of these results may be to recommend a shed roof design, possibly with a concave curve, to minimize the risk of the wind uplift of the roof if the prevailing wind direction can be identified.

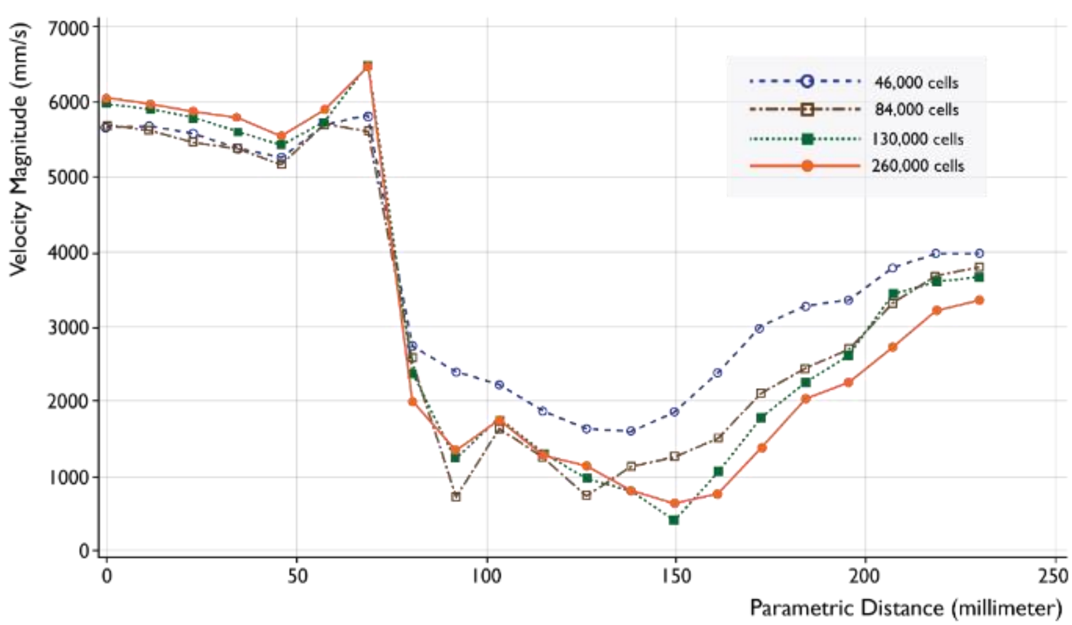


*Fig. 4. Grid sensitivity study - velocity distributions along the center of the building section.*

## Conclusion

The present study explored how various roof form options could affect air pressure distribution on the roof surface, focusing on hurricane events in Louisiana. In addition to a baseline with a flat roof, five design options were developed for the study: conventional gable roof, extended gable roof, shed roof, concave roof, and convex roof. CFD simulations were performed to examine the risk of wind uplift in the selected design scenarios in a hurricane event. The simulation result indicated that the shed roof design, which can be combined with a concave curve, may effectively reduce the risk of wind uplift if the wind direction is well known. The limitation of the study would be the fixed wind direction with an assumed building orientation and the absence of physical validation. Experimental research testing multiple building orientations and wind directions may overcome this limitation. The present study emphasizes the beginning of a more comprehensive workflow to understand the behavior of an expanded set of roof form systems, which will be allowed at an affordable cost due to the advancement of automated construction technology. This study may be expanded to other parameters, such as symmetrical and asymmetrical slopes on different sides of the building and the curvature at the corners of the walls. Additional tools for CFD simulations may also be explored, such as Eddy3D, a plugin for Grasshopper/Rhino. The established simulation-assisted design framework, focusing on the resiliency of the structure, may be utilized for developing new roof design prototypes for hurricane-prone areas and for pedagogical studies addressing resiliency issues. The present study may assist architectural designers in

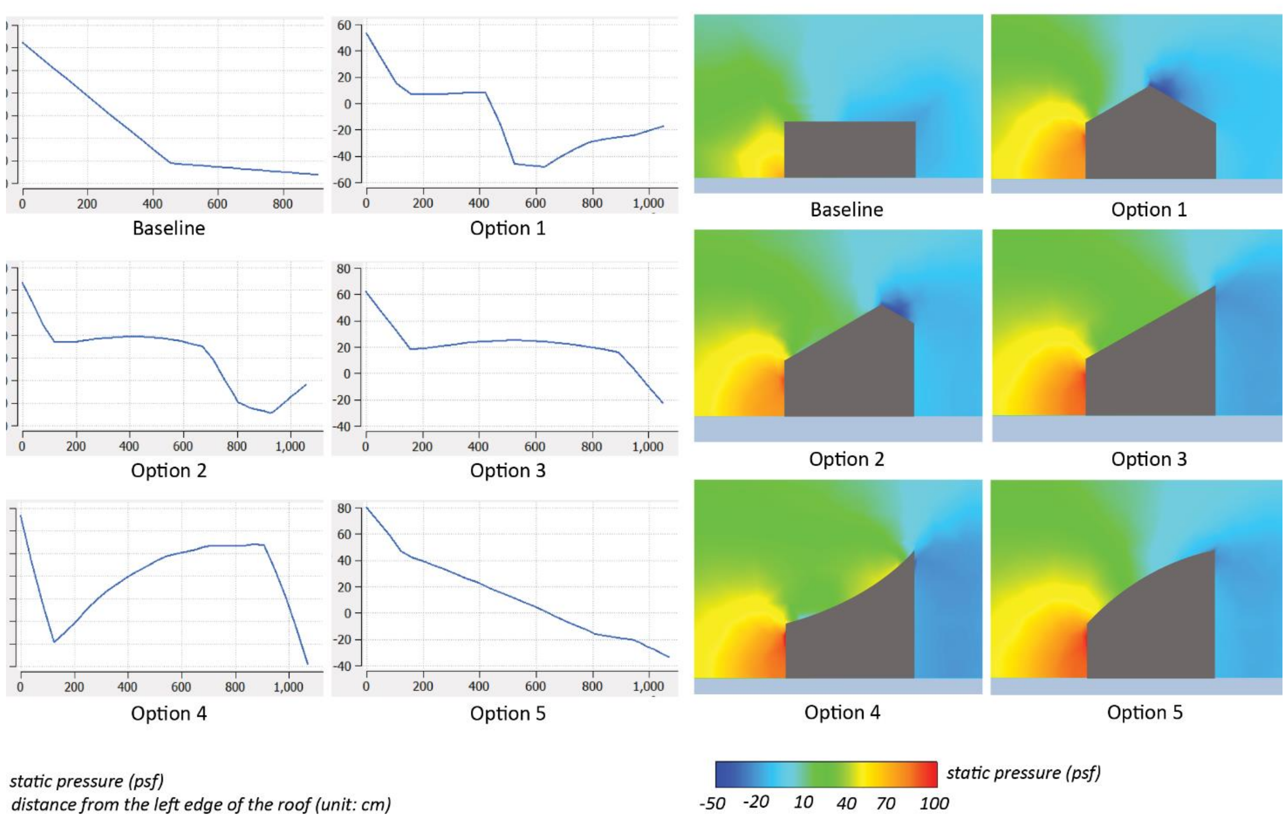


*Fig. 5. Air pressure distribution on the roof surface along the center axis of the tested buildings (simulated in Autodesk CFD)*

creating a more resilient building form in the early stages of design and advance the understanding of the environmental issues and evolving needs for sustainable design. This work will ultimately promote more environment-responsive building design.

**Notes:**

1. FEMA, *Natural Disaster Housing Strategy*. 2009.
2. BevenlI, J.L., A. Hagen, and R. Berg, *NATIONAL HURRICANE CENTER TROPICAL CYCLONE REPORT - HURRICANE IDA (AL092021)*. 2022.
3. Henry, C. *In Progress: The J-House / AEDS*. 2011 [cited 2025; Available from: https://www.archdaily.com/138398/in-progress-the-j%25e2%2580%2590house-aeds.
4. Adel, A., *Co-Robotic Assembly of Nonstandard Timber Structures*, in *ACADIA*. 2022.
5. ArchDaily. *Norman Foster Foundation and Holcim Present New Concept for an "Essential Home"*. 2023 [cited 2025; Available from: https://www.archdaily.com/1000718/norman-foster-foundation-and-holcim-present-new-concept-for-an-essential-home?ad_medium=gallery.
6. Duarte, G., et al., *Learning from historical structures under compression for concrete 3D printing construction.* Journal of Building Engineering, 2021. **43**: p. 103009.
7. REPORTER, S. *ETH Zurich Combines 3d Printing and Casting To Create Energy Efficient Concrete Slab Ceiling In This Office | Switzerland*. 2022 [cited 2025; Available from: https://surfacesreporter.com/articles/134274/eth-zurich-combines-3d-printing-and-casting-to-create-energy-efficient-concrete-slab-ceiling-in-this-office-switzerland.
8. Franke, J., et al., *The COST 732: Best practice guideline for the CFD simulation of flows in the urban environment.* University of Hamburg, Meteorological Institute, Hamburg, Germany, 2007.
9. Tominaga, Y., et al., *AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings.* Journal of wind engineering and industrial aerodynamics, 2008. **96**(10): p. 1749-1761.
10. Blocken, B., W. Janssen, and T. van Hooff, *CFD simulation for pedestrian wind comfort and wind safety in urban areas: General decision framework and case study for the Eindhoven University campus.* Environmental Modelling & Software, 2012. **30**: p. 15-34.
11. Wu, H. and F. Kriksic, *Designing for pedestrian comfort in response to local climate.* Journal of Wind Engineering and Industrial Aerodynamics, 2012. **104**: p. 397-407.
12. Blocken, B. and T. Stathopoulos, *CFD simulation of pedestrian-level wind conditions around buildings: Past achievements and prospects.* Journal of Wind Engineering and Industrial Aerodynamics, 2013. **121**: p. 138-145.
13. Jo, S.J. and J. Jones, *A Review of the Fundamental Concepts of CFD for Novice Users in Building Research and Design*, in *The 18th IBPSA Conference Building Simulation 2023*. 2023, IBPSA: Shanghi, China.
14. NOS, *Hurricane Ida - NOS Fiscal Year 2021 Year in Review*. 2021, National Ocean Services.
15. Snaiki, R. and T. Wu, *An analytical model for rapid estimation of hurricane supergradient winds.* Journal of Wind Engineering and Industrial Aerodynamics, 2020. **201**: p. 104175.
16. Vickery, P.J., et al., *A hurricane boundary layer and wind field model for use in engineering applications.* Journal of Applied Meteorology and Climatology, 2009. **48**(2): p. 381-405.
17. Ramponi, R. and B. Blocken, *CFD simulation of cross-ventilation flow for different isolated building configurations: validation with wind tunnel measurements and analysis of physical and numerical diffusion effects.* Journal of Wind Engineering and Industrial Aerodynamics, 2012. **104**: p. 408-418.
18. Blazek, J., *Computational fluid dynamics: principles and applications*. 2015: Butterworth-Heinemann.
19. Autodesk. *Autodesk Knowledge Network*. 2019 [cited 2020 09/15]; Available from: https://knowledge.autodesk.com/support/cfd/learn?sort=score.
20. Karava, P., T. Stathopoulos, and A. Athienitis, *Airflow assessment in cross-ventilated buildings with*

*operable façade elements.* Building and Environment, 2011. **46**(1): p. 266-279.

21. Wang, J., et al., *Numerical investigation of single-sided natural ventilation driven by buoyancy and wind through variable window configurations.* Energy and Buildings, 2018. **168**: p. 147-164.
22. Riverola Calavia, J., *Discretization error estimation and mesh optimization for plasma edge transport simulations of nuclear fusion tokamaks*. 2016, Industriales.