Predicting Rooftop UAS Performance Using Computational Fluid Dynamics

Ken N. Andrew

Embry-Riddle Aeronautical University

UNSY 691 Graduate Capstone

Submitted to the Worldwide Campus

in Partial Fulfillment of the Requirements of the Degree of

Master of Science in Unmanned Systems

May 16, 2021

Abstract

Applications of unmanned aerial systems for package delivery will require an understanding of building-induced aerodynamics at, and above, rooftop level. Wind velocity and windshear above tall buildings can negatively impact or overwhelm an unmanned vehicle's wind resistance performance. The present study utilized computational fluid dynamics to investigate rooftop airflow patterns, identify alternative approach and departure paths, and compare results against performance data for three commercially available unmanned aerial delivery systems. The simulation used ANSYS Fluent to simulate a 30 mile per hour east wind applied to a model of a 16 floor, 185 foot high, non-symmetrical building in Portland, Oregon. Results showed that winds in the rooftop area were found to be 28 percent higher than the freestream wind velocity. Alternative approach and departure paths with decreased windshear and wind velocity were identified. Commercially available aerial delivery systems were found to have inadequate wind resistance performance for the conditions modeled. Future studies should aim to utilize computational fluid dynamics for buildings with varying shapes and heights, while increasing cell count for additional resolution.

Keywords: drone delivery, unmanned aerial system performance, wind resistance, building aerodynamics, computational fluid dynamics

The use of UAS for package delivery is being developed and trialed by a number of companies in the United States. These systems are currently being tested in relatively rural areas until they are proven to be safe, effective, and able to integrate into the national airspace system. The urban environment brings new challenges to UAS delivery, including operations over people, congested communication frequencies, and complex wind patterns.

When estimating wind in an urban area, the use of aviation weather sources, while sufficient for airport and winds aloft observations, cannot capture the complex, localized airflow patterns in an urban or rooftop area. Alternatively, the use of non-aviation ground weather stations located throughout an urban area cannot capture three-dimensional airflow patterns. Awareness of micro-scale wind patterns are beneficial for package delivery UAS as these systems will require a tether, parachute, or landing for payload delivery, all of which occur at lower altitudes where airflow complexity increases due to wind interaction with structures.

Statement of the Problem

The purpose of this study is to employ computational fluid dynamics (CFD) to predict rooftop-area airflow patterns for a 16 floor, 196 foot high, non-symmetrical building in Portland, Oregon.

This study is important as findings from CFD analyses could improve UAS safety and performance for rooftop delivery zones. As a UAS descends into a rooftop area, the geometry of the building could magnify the wind speed, creating wind shear between the frontal corner separated flow and rooftop recirculation flow. Modeling the altitude and magnitude of this transition determines if a rooftop payload delivery is feasible or not for a given wind speed. In addition, this modeling technique could identify alternative approach and departure paths that mitigate wind-based risk. The simulation output will be quantitatively and qualitatively analyzed against performance data for several delivery UAS models.

Research Questions

The research question for this study is: "Can computational fluid dynamics predict rooftop-area UAS performance for a building in the Portland area?" In researching this question, there are three supporting research questions that will be discussed:

1. Are airflow patterns and velocity at rooftop level significantly different from the freestream wind velocity?
2. Does the simulation identify alternative approach and departure paths with less wind shear (sudden velocity changes)?
3. Can today's delivery UAV withstand the wind conditions modeled in this study?

Delimitations

Due to the accelerated timeline of this 9-week course, the analysis will be limited to one predominant wind direction gathered from historical winter weather data. Wind speed will be selected as a moderately high value that is observed several times a year.

This study does not include temperature as a factor. Temperature can have an additional effect on wind and vertical flow patterns due to the urban heat island effect. Urban CFD simulations that include temperature are less common due to the increased complexity and processing time required for simulation.

For the sake of model simplification, the building under study will be modeled as an isolated building in a freestream. In a traditional urban environment, wind patterns are affected by other natural and manmade features such as terrain, buildings, structures, and trees. In CFD simulations, the inclusion of such features drastically increases cell count and simulation runtime.

Limitations and Assumptions

Due to the short length of this capstone course, this airflow analysis will be limited to one building. The building under study is The Civic in Portland, Oregon, located at 1926 W Burnside Street. This 16-floor building was constructed in 2007 and is approximately 196 feet in height. Although the results of this study will be determined by the height and geometric features of The Civic, the concepts and findings are applicable to other mid-rise buildings where UAS will be operating.

The software chosen to run this CFD simulation is ANSYS Student. The student version of this software is limited to a maximum of 512,000 cells for fluid physics simulations. This limitation will result in models and results that are lower in resolution compared to the examples in the literature review.

Literature Review

Computational fluid dynamics (CFD) is a processor-intensive simulation that predicts the flow of fluid by solving the Navier-Stokes equation. CFD is the digital equivalent of a physical wind tunnel. This type of analysis is used in various engineering disciplines such as aerospace, HVAC, fire protection, and electronics manufacturing. CFD has been used to model urban areas as early as 1995 (Mestayer, et al.). These early models were limited by very low resolution and focused on small segments of an urban area.

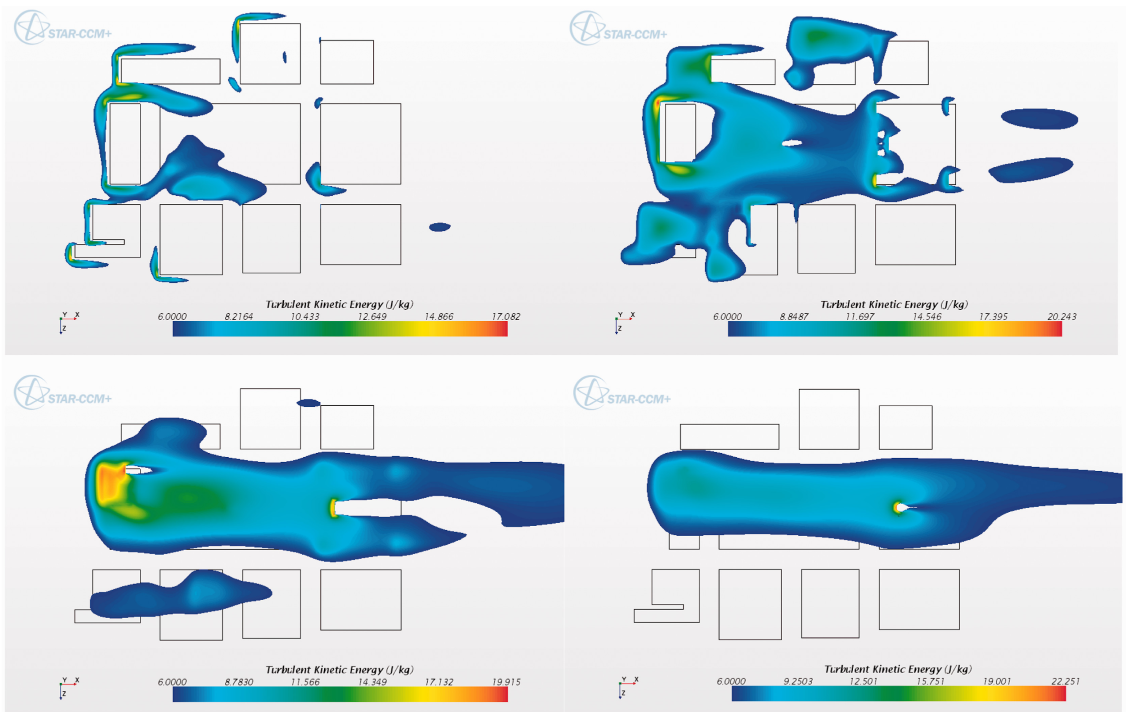
By 2010, Ashie and Kono (2010) performed a large-scale CFD analysis of the Tokyo Metropolitan Area to study the urban heat island effect while incorporating onshore cooling flow from Tokyo Bay. This study, while not UAS centric, provides one of the highest resolution CFD studies for an urban area to date. The inclusion of temperature highlighted how heat from buildings creates temperature streaks that can vary local air temperatures by 2 to 3 degrees Celsius. Ashie and Kono's research exhibits how groups of buildings influence each other.

Subsequent research has applied CFD to the UAS environment at a smaller scale. Ware and Roy (2016) studied wind field estimation for quadrotor UAS flight in the urban canopy layer. This study utilized CFD using a model of the Massachusetts Institute of Technology campus approximately 1 by 1.2 km in length. This CFD data was used to determine UAS energy consumption and speed at an altitude of 10 m. This study did not analyze wind patterns above 10 m where commercial UAS will spend the majority of their time in flight.

Murray and Anderson (2017) modeled a small area of Glasgow (250 m by 300 m) which was used for a CFD simulation. This simulation studied how a small UAS would be affected by building-induced aerodynamics at low altitudes (10 m, 20 m, 30 m, and 40 m). Murray and Anderson found that their specific model of UAS could operate anywhere in the model at a wind speed of 5 m/s (11.18 mph). However, when a wind speed of 10 m/s (22.37 mph) was simulated, there were areas in the model that exceeded the critical turbulent kinetic energy (TKE) for their specific unmanned aircraft. Of note is the 30 m view, where high TKE values can be seen just above the roof of the windward building.

Figure 1

TKE at Various Heights



Note. Areas where TKE surpasses 10 m/s at heights of 10 m, 20 m, 30 m, and 40 m (left to right, top to bottom) (Murray & Anderson, 2017).

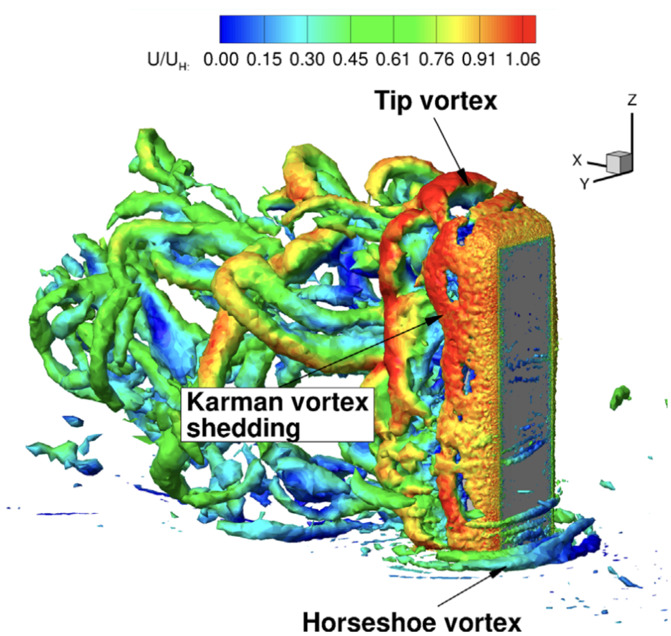
CFD has been used to study the effect of wind on high-rise buildings, however, studies typically target wind load, vibrations, and occupant comfort. Thordal et al. (2020a) used both CFD and wind tunnel modeling to study wind load on a generic 180 m (590 ft) high-rise building using five wind directions: 0°, 30°, 45°, 60°, and 90°. The wind flow around the building was found to be asymmetric, being influenced by an upstream church tower. Due to this finding, the mesh needed refinement to better capture the airflow reattachment points on the building's surfaces.

A follow-up study by Thordal et al. (2020b) updated the building's shape by rounding the four 90° edges. Similarly to the previous study, results were agreeable between a physical wind tunnel and CFD results. One of the wind directions (0°) caused the simulation to not properly resolve airflow near the modified corners. The identified mitigation was to use a smaller time step in the simulation.

CFD has also been used to study wind-induced vibrations on high-rise buildings. Zhang et al. (2015) used the same generic high-rise building as Thordal et al. to study the effect of wind using a Reynolds-Averaged Navier–Stokes turbulence model. This study consisted of 1.6 million cells (nodes) and took a total of 122 hours to process. Like the other studies, CFD was found to be accurate compared to physical modeling. In addition, CFD was found to be less expensive and easier to modify than physical modeling. Of note in the study was the wind velocities that exceed the freestream velocity originating at the building's leading edges.

Figure 2

Large-Scale Turbulence around Building



Note. Wind velocity exceeds that of the freestream velocity 8.945 m/s (20 mph) originating at the building's leading edges (Zhang et al., 2015).

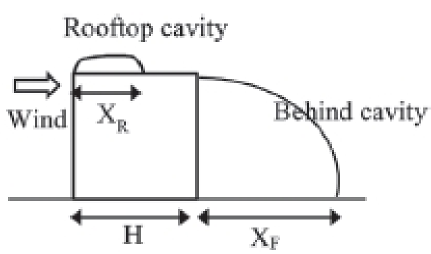
Several studies have applied CFD to rooftop areas, however, none pertain specifically to how rooftop wind patterns impact UAS. Tabrizi et al. (2014) performed an external CFD analysis of a western Australian warehouse to predict inflow for a roof-mounted wind turbine. This application, although not specifically for UAS and utilizing a low-height building, demonstrates that CFD modeling results for rooftops are reasonably accurate compared to measured data.

Jeong (2017) studied the effect of rooftop odor dispersion on downstream buildings. This research studied the downstream plume in three configurations: one, two, or three downstream buildings. Similar to previous studies, Jeong (2017) found that CFD data was agreeable with wind tunnel experiments. It was concluded that increasing the number of downstream buildings caused recirculation regions with high particulate count. The results of this study could be of interest if the delivery UAV were carbon-fuel based and emitted exhaust gases.

Jeong and Kim (2018) revisited this topic, this time focusing on how atmospheric stability influences rooftop emissions. It was found that CFD produced agreeable results when considering rooftop cavity reattachment length (X_R) compared to wind tunnel results. However, reattachment length of the building cavity was found to be overestimated in the CFD simulation (X_F).

Figure 3

Diagram of Reattachment Lengths



Note. CFD produces agreeable results for the rooftop cavity area but overestimates the cavity behind (leeward) of the building (Jeong & Kim, 2018).

Summary

This section has reviewed the relevant literature pertaining to CFD and UAS. Thus far, applicable CFD studies have focused on large-scale city modeling, UAS modeling at lower altitudes, or non-UAS high-rise applications. The use of CFD to model rooftop areas for UAS delivery applications has not yet been researched.

Method

This section describes research questions, design, data collection, and data analysis.

Research Questions

The main research question for this study is: "Can computational fluid dynamics predict rooftop-area UAS performance issues for a high rise building in the Portland area?" In researching this question, there are three supporting research questions that will be discussed:

1. Are airflow patterns and velocity at rooftop level significantly different from the freestream wind velocity?
2. Does the simulation identify alternative approach and departure paths with less wind shear (sudden velocity changes)?
3. Can today's delivery UAV withstand the wind conditions modeled in this study?

Geometry

A model of The Civic building in Portland was created in ANSYS SpaceClaim, a 3D modeling program. Building footprint information was gathered from a combination of Google Earth satellite imagery and a publicly available floor plan drafted by SERA Architects Inc. The building height was determined to be 59.7 m (196 ft), gathered from the City of Portland's ArcGIS Open Data site. Building geometry was simplified to remove unnecessary detail that could have complicated the meshing process (e.g. vehicles, trees, or rooftop HVAC rooftop units). A fluid enclosure was added around the building with sufficient windward, leeward, and vertical distance from the building.

Figure 4

Google Earth vs. Generated 3D Model

Note. Perspective view of The Civic in Google Earth (left). 3D model simplified for CFD, generated in ANSYS SpaceClaim software (right). Enclosure not shown.

Meshing

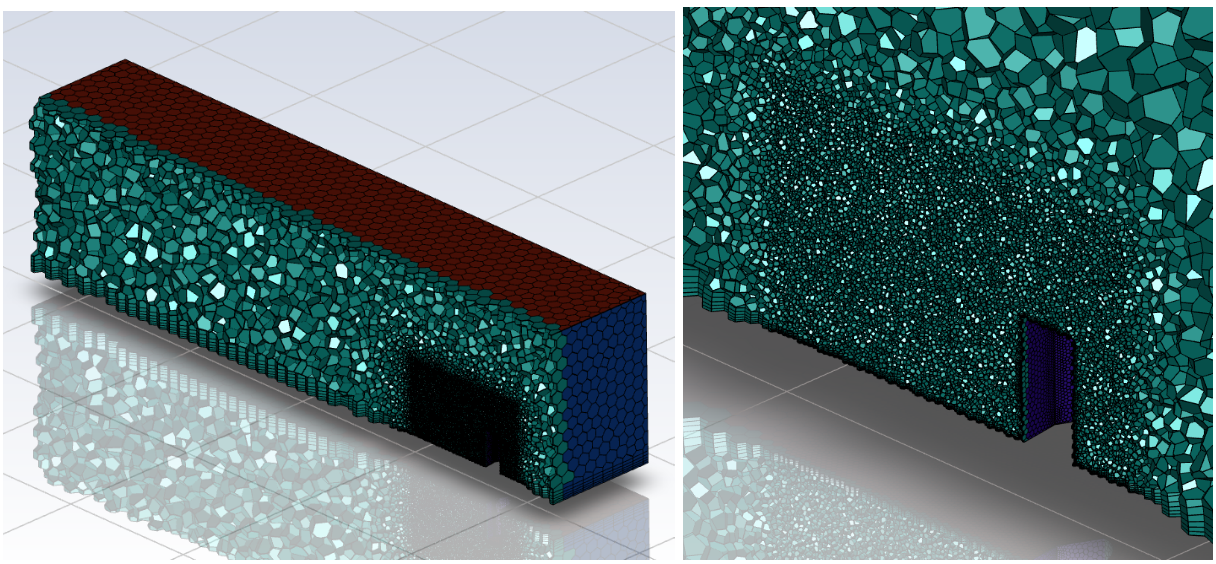
The meshing process divides the computational domain into smaller, discretized volumes known as cells. The density of cells can be adjusted on a global level, within specific volumes (body of influences), by plane, or edge.

When meshing The Civic, the maximum mesh length was set to 20 meters. Cell density was increased using a body of influence around the building. This body of influence encompassed areas where complex airflow was expected: from the ground to above the rooftop that serves as the UAS payload delivery zone, past the sides of the building for corner streams, and behind the building for vortices and recirculation zones. The target mesh size for the body of influence was set to 2.6 meters with a growth rate of 1.2. The number of boundary layers was increased from the default of 3 to 5 to accurately capture velocity gradients near surfaces.

Cell count directly affects computation time, therefore, a balance between acceptable simulation density and computation time must be found. The ANSYS Student software was limited to a maximum of 512,000 cells for fluid physics simulations. After several rounds of refinement, the total cell count for the simulation was 478,971 polyhedra cells.

Figure 5

Final Mesh



Note. Perspective views of final mesh in ANSYS Fluent. Overall mesh of entire enclosure (left). Refined mesh area within the body of influence (right). The area void of cells is the building's volume.

Inputs

Wind direction and speed were determined from historical weather data gathered at the Portland International Airport, 10 km (6.2 mi) NE of the city center. Wind from the east was selected as the most common wind direction during the winter season. A wind speed of 13.4 m/s (30 mph) was selected as a value that was high, but not untypical for the season.

When applying inputs, the two volumes were first assigned as solid or fluid. The building volume was assigned as a solid and the surrounding air enclosure was assigned as a fluid. Next, the capping surfaces of the enclosure were assigned with inlet, outlet, or symmetry boundary conditions. An inlet velocity of 13.4 m/s was applied to the eastern enclosure surface. The western enclosure surface at the leeward side of the model was designated as a 0 Pa pressure outlet. A symmetry boundary condition was applied to three surfaces (northern, southern, and upper), all parallel to the wind input direction.

Solving

ANSYS Fluent software was used to run the CFD simulation. CFD software calculates fluid flow by solving governing equations for each cell in the simulation over a number of iterations until the solution converges. A converged solution is reached when changes in variables (x, y, z-velocity, pressure, turbulence kinetic energy, turbulence eddy dissipation, etc.) are negligible between iterations.

The amount of time required for convergence is highly dependent on the total cell count. Due to the low number of cells in this simulation, convergence was achieved in 500 iterations with a runtime of 24 minutes and 57 seconds.

Post-Processing

Data from the simulation was gathered using various post-processing tools built into the ANSYS Fluent software. Data most relevant to this study were wind velocity and direction. This information was obtained using data points, lines, speed distribution layers, and particle trajectories.

Results

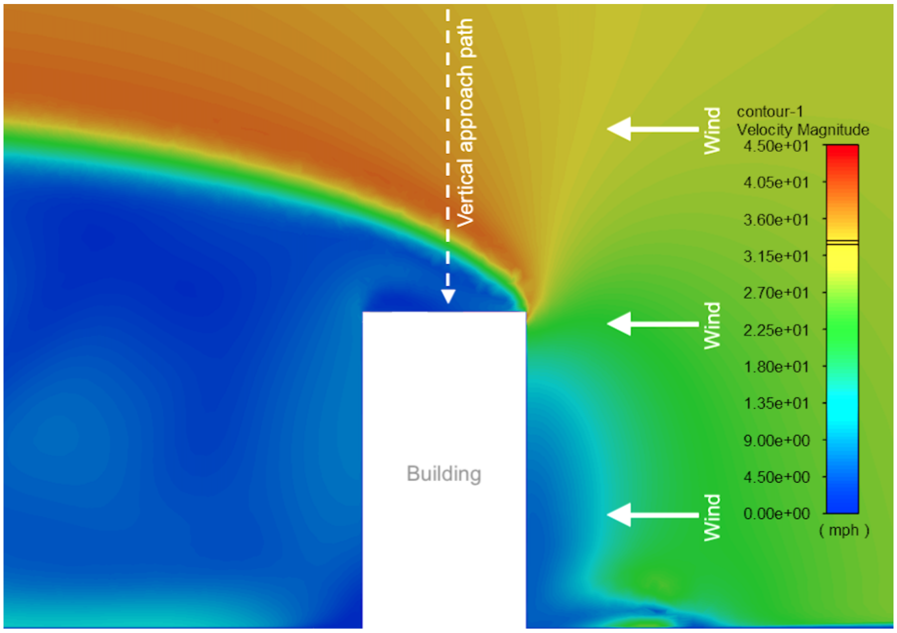
The CFD simulation results were consistent with relevant literature and studies. As expected, the wind speed upstream of the building was uniform at 30 mph. The wind approaching the building's east-facing facade was split downward and upward at the stagnation point, at a height of approximately 24 m (80 ft). The downward stream created a vortex at the base of the building. The upward stream climbed vertically past the rooftop, then accelerating and flowing over the building at speeds greater than 30 mph. The areas to the side of the building were subject to corner streams where wind velocity also exceeded 30 mph. Below the high speed flow over the building, the rooftop area produced a slow vortex of recirculating air. The area downstream of the building was subject to a large recirculation area filled with complex but slow moving wake vortices. The flow around the building was restored at approximately 10 building lengths downstream.

Traditional Flight Path

The area extending vertically above the rooftop delivery zone was analyzed. This area is favorable for traditional UAS approaches and departures. When descending from a height of 121.9 m (400 ft) above ground, or 62.2 m (204 ft) above the rooftop, the UAV would first be subject to winds of 32.8 mph. As the UAV continued to descend, wind speed would increase to a maximum of 38.3 mph at an altitude of 78 m (256 ft) above ground, or 18.2 m (60 ft) above the roof. When descending a further 10.5 m (34.5 ft), the vehicle would see a sudden decrease in wind speed, from the aforementioned 38.3 mph to a relatively calm 3.6 mph. When descending from this height to the rooftop, wind speed remained below 4.5 mph in a slow vortex of recirculating air (blue triangular region in Figure 6).

Figure 6

Speed Distribution Plane



Note. This figure displays a speed distribution plane that intersects the building parallel to the wind.

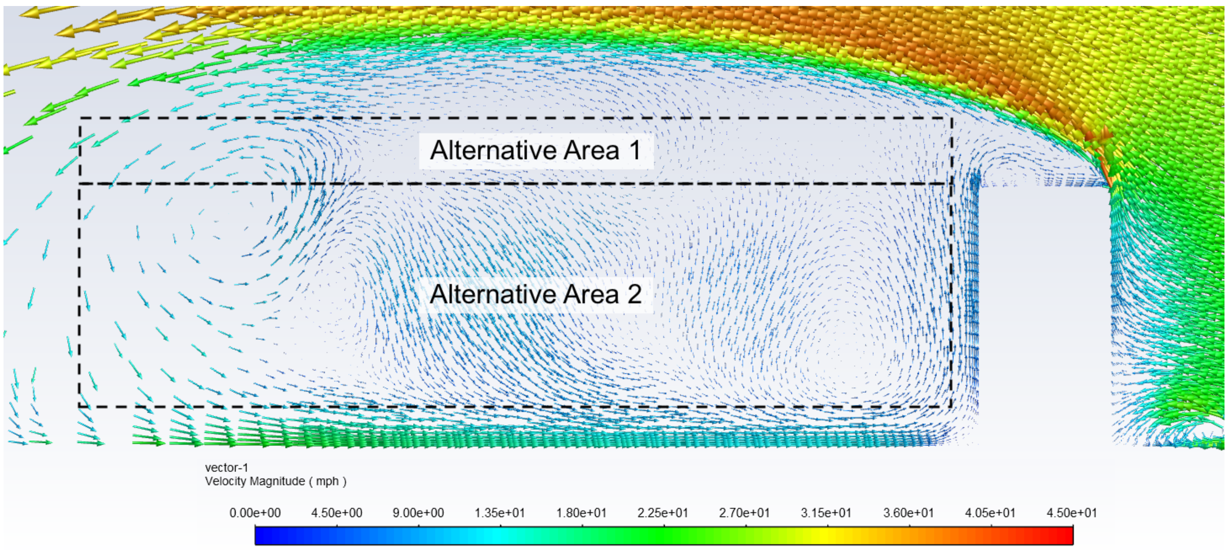
Alternative Flight Paths

Downwind and side regions were analyzed for alternative approach and departure paths to maximize efficiency. Side regions were found to have areas of windshear similar to the approach path above the rooftop. Side regions were not investigated further due to the hazard this presented.

The region downwind of the building was found to have desirable airflow patterns. This area, although subject to changes in wind direction, maintained wind velocities under 10 mph. This relatively stagnant area exists between the altitudes of 10 m (32.8 ft) and 77 m (252.6 ft) above ground level and extended 160 m (525 ft) behind the building. The area below rooftop level (Figure 7, Alternative Area 2) was subject to more downdrafts than the area above rooftop level (Figure 7, Alternative Area 1), making Alternative Area 1 preferred for alternative flight paths.

Figure 7

Velocity Vectors



Note. This figure displays a velocity vector plane that intersects the building parallel to the wind. Alternative Area 1 and 2 are shown as potential approach areas to minimize wind velocity changes.

Comparing Results to Commercially Available Delivery UAS

To evaluate safety, wind speed velocities gathered from this simulation were compared against wind resistance data for three delivery UAS currently on the market: Matternet M2, Rakuten ACSL, and Wingcopter 178.

The Matternet M2 is a delivery UAS being trialed by Swiss Post. This UAS has an empty weight of 9.5 kg (20.9 lb) and a payload capacity of 2.0 kg (4.4 lb) (Matternet, 2017). Wind resistance data is not available for the M2, however, the model has a maximum airspeed of 12.0

m/s (26.8 mph). Based on this information, the M2 would not be capable of safe flight with the given wind input of 13.4 m/s (30 mph).

Rakuten, a Japanese e-commerce company, has partnered with Autonomous Control Systems Laboratory Ltd. (ACSL) to trial package delivery using UAS. The ACSL delivery UAS has an empty weight of 7.1 kg (15.6 lb) and a payload capacity of 2.8 kg (6.1 lb) (ACSL, 2021). ACSL states that this model of UAS has a wind resistance of 10.0 m/s (22.3 mph) which is not sufficient for the conditions in this analysis.

The Wingcopter 178 is a delivery UAS that can transition between hover and fixed-wing modes. The Wingcopter 178 has an empty weight of 8.4 kg (18.5 lb) and a payload capacity of 6.0 kg (13.2 lb) (Wingcopter, 2021). This model has a specific wind resistance value of 15.0 m/s (33.5 mph) for takeoff and landing. Although this wind resistance value exceeds the freestream wind speed of 30 mph, the Wingcopter 178 could not operate within a safe flight envelope given the results of this study. Approaching or departing the rooftop of The Civic would subject the UAV to gusts modeled to be 38.3 mph.

Conclusion

The purpose of this study was to gain a better understanding of rooftop area wind patterns using computational fluid dynamics. The results of the study support that computational fluid dynamics can successfully predict package delivery UAS performance issues for a high rise building in the Portland, Oregon area.

Airflow patterns and velocity at rooftop level were found to be 28 percent higher than the freestream wind velocity. Given the height and shape of the building under study, it was found that the 30 mph freestream velocity resulted in a high velocity region above the roof with predicted wind speeds of 38.3 mph. Given a traditional vertical approach and departure path above the rooftop, a UAV would encounter windshear that could destabilize the vehicle. A taller building or higher wind speed would further increase the amount of windshear.

The simulation identified alternative approach and departure paths with decreased wind shear (sudden velocity changes). The alternative approach and departure path with the smallest velocity changes was downwind of the building, just above rooftop level. In real-world applications, this region behind the building may not be usable as it could be occupied by other buildings. Furthermore, a high-speed UAV flying horizontally at altitudes below rooftop level could pose a safety hazard. However, this research shows that more efficient flight paths exist leeward of structures that impede the freestream. These findings could be applied to clusters of buildings (like a downtown urban area) or geographic features (hills or mountains).

This study also demonstrated that today's delivery UAV cannot withstand the wind conditions modeled in this study. Wind resistance data was gathered for three delivery UAS currently on the market: Matternet M2, Rakuten ACSL, and Wingcopter 178. All three of these commercially available aerial delivery systems were found to have inadequate wind resistance performance for the conditions modeled. Only one of the three, the Wingcopter 178, had a wind resistance value higher than the freestream wind velocity. However, the simulation results showed that these values would be exceeded due to the windshear above the rooftop caused by the accelerated wind flow over the building.

One limitation of this study was the use of the ANSYS Student software package, which restricted model cell count to 512,000 cells for fluid physics simulations. The use of a non-student licence would allow for a simulation with much greater resolution. Additional resolution could be used to more accurately model the rooftop area with a landing pad, rooftop HVAC units, cladding, and other relevant building features. In addition, wind velocities and airflows could be better captured and studied along with interactions from surrounding buildings and structures.

Recommendations

As package delivery expands beyond limited trials and testing, it is imperative that the effect of building induced aerodynamics is considered for rooftop and urban environments. Package delivery UAV will be operating at heights where an accident could pose great danger to pedestrians at ground level. Based on the results of this study, two recommendations are provided:

1. Given the wind resistance data for package delivery UAS currently on the market, any operations involving high rise buildings should be suspended when local wind speed is measured to be 75 percent of the manufacturer's established wind resistance value. This will ensure that the UAV is not subject to wind speeds that exceed the safety envelope of the vehicle.
2. A downwind, horizontal approach path just above rooftop level should be utilized to minimize windshear and high wind velocity. This approach path uses the building as an impediment to the wind, using a flight path in the lower velocity, wake vortex region that exists downwind of the building. In real-world applications, this region behind the building may not be usable as it could be occupied by other buildings. Furthermore, spatial, safety, or regulatory conflicts may occur with these lower altitude operations.

Future Research

Much work remains to be done when applying computational fluid dynamics to rooftop package delivery UAS performance. Future directions for research should expand on this study by:

1. Utilizing buildings with various shapes and heights.
2. Modeling larger areas to study the wind interactions between buildings in an urban core.
3. Studying the effect of wind direction on non-symmetrical buildings.
4. Studying the effect of local terrain in cities with close proximity to mountains (e.g. Portland, Oregon; Denver, Colorado; Hong Kong, PRC; Seoul, Korea).
5. Utilizing non-student software for greater cell count and model resolution.

Despite the given limitations, this study has provided an initial step towards integrating the fields of package delivery UAS performance and computational fluid dynamics.

References

Ashie, Y., & Kono, T. (2011). Urban‐scale CFD analysis in support of a climate‐sensitive design for the Tokyo bay area. *International Journal of Climatology, 31*(2), 174-188. doi:10.1002/joc.2226

Autonomous Control Systems Laboratory Ltd. (ACSL) (2021). *Delivery Logistics / Mail*. Retrieved from ACSL website at https://www.acsl.co.jp/en/solutions/delivery-logistics-mail/

Jeong, S. J. (2017). A CFD study of near-field odor dispersion around a cubic building from rooftop emissions. *Asian Journal of Atmospheric Environment, 11*(3), 153-164. https://doi.org/10.5572/ajae.2017.11.3.153

Jeong, S. J., & Kim, A. R. (2018). CFD study on the influence of atmospheric stability on near-field pollutant dispersion from rooftop emissions. *Asian Journal of Atmospheric Environment, 12*(1), 47-58. https://doi.org/10.5572/ajae.2018.12.1.047

Matternet Inc. (2017). *Specifications (March 2017)*. Retrieved from Swiss Post website at https://www.post.ch/-/media/post/ueber-uns/medienmitteilungen/2017/drohnen/spezifikationen-matternet-m2.pdf

Mestayer, P. O., Sini, J. F., & Jobert, M. (1995). Simulation of the wall temperature influence on flows and dispersion within street canyons. *WIT Transactions on Ecology and the Environment, 9* doi:10.2495/AIR950141

Murray, C. W., & Anderson, D. (2017). A CFD-based procedure for airspace integration of small unmanned aircraft within congested areas. *International Journal of Micro Air Vehicles, 9*(4), 235-252. doi:10.1177/1756829316669957

Tabrizi, A. B., Whale, J., Lyons, T., & Urmee, T. (2014). Performance and safety of rooftop wind turbines: Use of CFD to gain insight into inflow conditions. *Renewable Energy, 67* 242-251. https://doi.org/10.1016/j.renene.2013.11.033

Thordal, M. S., Bennetsen, J. C., Capra, S., & Koss, H. H. H. (2020a). Towards a standard CFD setup for wind load assessment of high-rise buildings: Part 1 – benchmark of the CAARC building. *Journal of Wind Engineering and Industrial Aerodynamics, 205*, 104283. https://doi.org/10.1016/j.jweia.2020.104283

Thordal, M. S., Bennetsen, J. C., Capra, S., Kragh, A. K., & Koss, H. H. H. (2020b). Towards a standard CFD setup for wind load assessment of high-rise buildings: Part 2 – blind test of chamfered and rounded corner high-rise buildings. *Journal of Wind Engineering and Industrial Aerodynamics, 205*, 104282. https://doi.org/10.1016/j.jweia.2020.104282

Ware, J., & Roy, N. (2016). An analysis of wind field estimation and exploitation for quadrotor flight in the urban canopy layer. Paper presented at the 1507-1514. doi:10.1109/ICRA.2016.7487287

Wingcopter (Jan 2021). *Technical Details Wingcopter - 178 Heavy Lift A (Delivery Variant)*. Retrieved from Wingcopter website at https://wingcopter.com/wp-content/uploads/2021/02/Technical-Details-Wingcopter-178-Heavy-Lift-A-Delivery-Variant-1-1.pdf

Zhang, Y., Habashi, W. G., & Khurram, R. A. (2015). Predicting wind-induced vibrations of high-rise buildings using unsteady CFD and modal analysis. *Journal of Wind Engineering and Industrial Aerodynamics, 136*, 165-179. https://doi.org/10.1016/j.jweia.2014.11.008