

1. Introduction

Micrometeorology is a part of meteorology that deals with observations and processes at small scales of time and space, approximately smaller than 1 km and one day (Foken et al., 2008). This region is characterized by different small-scale meteorological processes such as turbulence, wind shear, wind gusts, surface friction, and wind load. These processes affect UAV operations in dense urban airspaces. Wind shear and turbulence caused by buildings in the vicinity of airports have been identified as causes of aircraft instability during landing and takeoff for some time (Krüs et al., 2003). Therefore, in order to realize safe flight operations, careful study of turbulence modeling, simulation, and validation, as well as taking these processes into account in the design of advanced air mobility systems, is essential. In this concept note, I describe my understanding of these processes one by one.

2. Atmospheric Turbulence

2.1 Governing Equations (RANS)

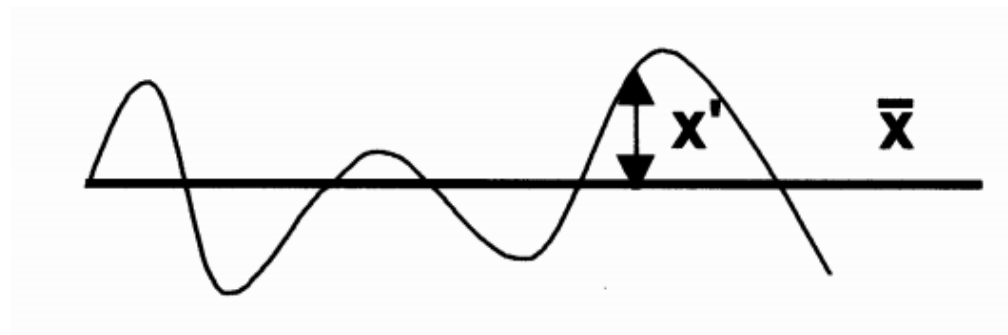
Atmospheric turbulence is initially represented by the Navier–Stokes equation, which is derived from Newton’s second law of motion. This law states that the acceleration of the atmosphere is equal to the summation of the forces causing motion in the atmosphere. These forces are the pressure gradient force, gravitational force, friction (viscous force), and Coriolis force

$$\begin{aligned}\frac{\partial u}{\partial t} &= -u \frac{\partial u}{\partial x} - v \frac{\partial u}{\partial y} - w \frac{\partial u}{\partial z} - \frac{1}{\rho} \frac{\partial p}{\partial x} + f v + \nu \nabla^2 u \\ \frac{\partial v}{\partial t} &= -u \frac{\partial v}{\partial x} - v \frac{\partial v}{\partial y} - w \frac{\partial v}{\partial z} - \frac{1}{\rho} \frac{\partial p}{\partial y} - f u + \nu \nabla^2 v \\ \frac{\partial w}{\partial t} &= -u \frac{\partial w}{\partial x} - v \frac{\partial w}{\partial y} - w \frac{\partial w}{\partial z} + g + \nu \nabla^2 w\end{aligned}$$

Even though the above equation represents motion in the atmosphere for zonal, meridional, and vertical components, it lacks turbulent representation; therefore, modification has to take place by introducing Reynolds's decomposition, as indicated by Foken et al. (2008).

$$\mathbf{x} = \overline{\mathbf{x}} + \mathbf{x}'.$$

The application of Reynolds's decomposition requires some averaging rules for the turbulent value x' (a represents a constant), which are termed Reynolds's postulates:



$$\begin{aligned} I \quad & \overline{x'} = 0 \\ II \quad & \overline{xy} = \overline{x} \overline{y} + \overline{x'y'} \\ III \quad & \overline{\overline{xy}} = \overline{x} \overline{y} \\ IV \quad & \overline{ax} = a \overline{x} \\ V \quad & \overline{x+y} = \overline{x} + \overline{y} \end{aligned}$$

After applying Reynolds's decomposition, Reynolds averaging rules (postulates), the Boussinesq approximation, and Einstein summation notation to the Navier–Stokes equations, we obtain the turbulent equations (RANS) (Foken et al., 2008).

$$\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial}{\partial x_j} (\bar{u}_j \bar{u}_i + \overline{u_j' u_i'}) = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + \nu \frac{\partial^2 \bar{u}_i}{\partial x_j^2} + g \delta_{i3} + \varepsilon_{ijk} \bar{\omega}_k$$

2.2 Computational Fluid Dynamics (CFD)

CFD numerically solves the governing Navier–Stokes equations to simulate fluid motion, and turbulence is either directly resolved or parameterized using approaches such as Reynolds-Averaged Navier–Stokes (RANS) or Large Eddy Simulation (LES), depending on the desired level of temporal and spatial resolution and computational cost (Al-Khalidy.. et al 2018)

2.3 Turbulence Modeling and Closure

The quality of CFD simulation depends on the selected turbulence model. In practical problems the turbulence model should be as simple as the relevant physics will permit For most real world building problems turbulence is, in principle, described by the Reynolds Averaged Navier-Stokes (RANS) equations [6].

According to Chaudhry [16] the three-dimensional Reynolds-Averaged Navier–Stokes (RANS) using the k–ε turbulence model is one of the most suitable numerical methods for built environment simulations. The model solves for two governing equations, the kinetic energy and turbulent dissipation.

2.3.1 Closure Techniques

The transition from the equation of motion for the mean stream to the turbulent flow gives a system of differential equations with more unknown parameters than equations. To solve the system of equations, assumptions have to be made to calculate the unknown parameters. These assumptions, for example the covariance terms, are called closure techniques (Foken et al., 2008).

After Reynolds averaging, the RANS equations contain more unknowns than equations. This is called the closure problem. The assumptions used to model these unknown turbulent terms are called closure techniques. In RWIND, closure is achieved using turbulence models such as k–ε and k–ω, which replace unknown turbulent terms with modeled expressions.

The Reynolds-averaged Navier–Stokes equations contain unknown Reynolds stress terms, leading to the closure problem. Closure techniques, such as the k - ϵ and k - ω models used in RWIND, parameterize these terms by relating them to mean flow variables through an eddy-viscosity assumption.

2.4 RWIND-Based Urban Flow Simulation

2.4.1 Initial and Boundary Conditions

Simulation Parameters ✕

General **Steady Flow** Turbulence Wind Profile Advanced Info

Simulation Type

☒ Steady flow Settings...

☐ Transient flow

☐ Save extended data to continue calculation

Flow Parameters

Inlet velocity: [m/s] Profile...

Kinematic viscosity: [m²/s]

Density: [kg/m³]

Finite Volume Mesh

Mesh density: [%]

Mesh cell estimation: 297 904 cells, min. cell size = 0.012 m

☒ Standard mesh refinements ☒ Refine downwind region

☒ Refine upwind region

☒ Boundary layers Model NL: Tunnel BL:

☒ Snap to model edges if possible

Default

OK Cancel Apply Help

All the required initial condition for this simulation is based on RWING3 guide line obtained from <https://www.dlubal.com/en/downloads-and-information/documents/online-manuals/rwind-3/005631>

The image shows a software dialog box titled "Simulation Parameters" with a close button (X) in the top right corner. The dialog has several tabs: "General", "Steady Flow", "Turbulence", "Wind Profile", "Advanced", and "Info". The "Steady Flow" tab is currently selected. Inside this tab, there are three main sections: "Initial Condition", "Steady Flow Calculation", and "Other Options".

- Initial Condition:** Contains a checked checkbox labeled "Use potential flow solver to calculate initial condition".
- Steady Flow Calculation:** Contains several input fields and a checkbox:
 - "Maximum number of iterations:" with a text box containing "1000".
 - "Minimum number of iterations:" with a text box containing "300".
 - "Convergence Criterion" is a label followed by a horizontal line.
 - "Residual type:" with a dropdown menu showing "Pressure".
 - "Residual target value:" with a text box containing "0.001".
 - A checkbox labeled "Use second-order numerical scheme" which is currently unchecked.
- Other Options:** Contains an unchecked checkbox labeled "Save the solver data in binary format".

At the bottom of the dialog, there is a "Default" button on the left and a row of four buttons on the right: "OK", "Cancel", "Apply", and "Help". The "OK" button is highlighted with a blue border.

Use potential flow solver to calculate initial condition Check this box (ON)
This is standard practice in RANS simulations (especially in wind engineering) because: *RWIND*

3 *Manual* mentions using potential flow for initial field helps solver start from a physically meaningful state. RWIND 3 Online Manual (Dluba) — see “Initial condition” section <https://www.dluba.com/en/downloads-and-information/documents/online-manuals/rwind-3/005640>

The image shows a software dialog box titled "Simulation Parameters" with a close button (X) in the top right corner. The dialog has five tabs: "General", "Steady Flow", "Turbulence", "Wind Profile", and "Advanced". The "Turbulence" tab is currently selected. Below the tabs, there are three main sections: "Steady Flow", "Transient Flow", and "Parameters".

Steady Flow section:

- ☒ Consider turbulence
- Turbulence model: RANS - K-omega SST (dropdown menu)

Transient Flow section:

- ☒ Consider turbulence
- Turbulence model: URANS - K-omega SST (dropdown menu)

Parameters section:

- ☒ Calculate parameters from the intensity of turbulence
- Turbulence intensity I: 20 [%] (input field) [Profile... button]
- Turbulent kinetic energy k: 11.76 [m²/s²] (input field)
- Turbulent dissipation rate ε: 2.51273 [m²/s³] (input field)
- Specific dissipation rate ω: 2.37409 [1/s] (input field)
- Turbulence length scale L: 2.637 [m] (input field) ☒ Automatic

At the bottom left of the Parameters section is a "Default" button. At the bottom of the dialog are four buttons: "OK", "Cancel", "Apply", and "Help".

General Steady Flow Turbulence **Wind Profile** Advanced Info

Wind Profile Type

Profile type: Wind Speed

Type of values: ☒ Relative ☐ Absolute (v in m/s)

Wind Profile Values

Number of points: 2

No.
1
2

Generate Wind Profile

Wind Profile Parameters

Profile height: 150.000 [m]

Number of points: 16

Profile type: Logarithmic

Minimum factor: 1

Maximum factor: 1

Wind speed at height: 10.000 [m]

Landscape type: Large towns with high buildings

OK Cancel Help

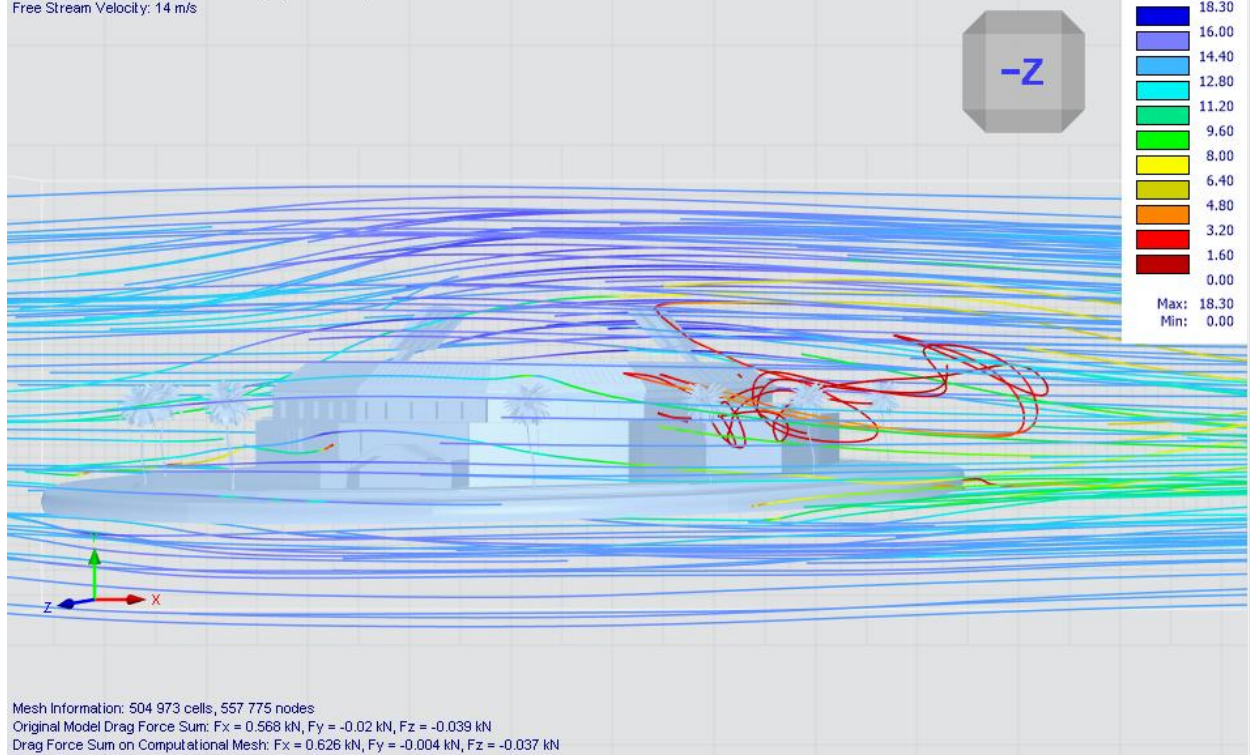
Factor [-]

0.6 0.9 1.2 1.5

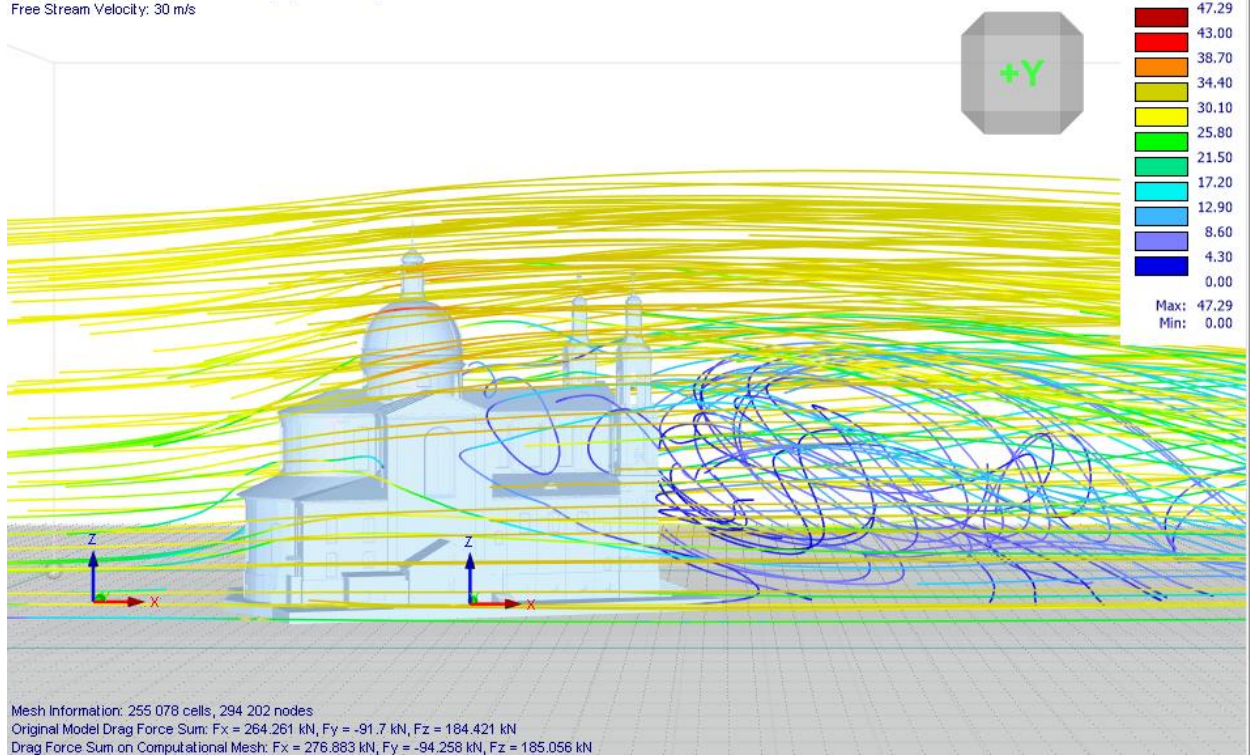
Default Constant Generate Refresh Graph

OK Cancel Apply Help

Wind Tunnel Dimensions: Dx = 45.063 m, Dy = 22.195 m, Dz = 18.874 m
Free Stream Velocity: 14 m/s



Wind Tunnel Dimensions: Dx = 429.576 m, Dy = 214.788 m, Dz = 123.816 m
Free Stream Velocity: 30 m/s



2.5 Conceptual framework

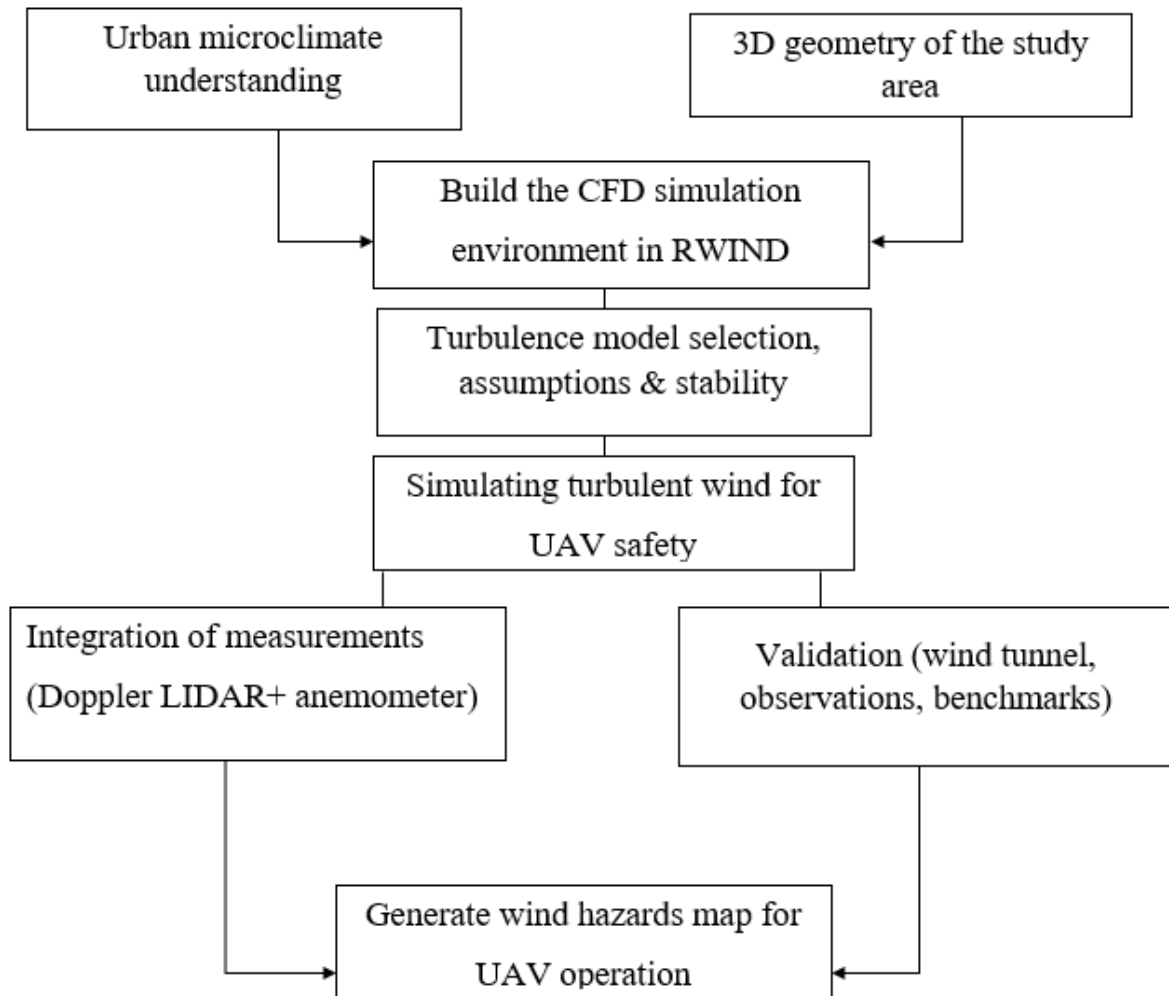


Figure 1 Conceptual framework linking urban micrometeorology, CFD modeling, and UAV wind safety assessment.

The CFD framework is based on the Reynolds-Averaged Navier–Stokes (RANS) equations with appropriate turbulence closure models (e.g., $k-\varepsilon$ and $k-\omega$.) Atmospheric stability assumptions and model uncertainties are considered when interpreting turbulence intensity and wind hazard maps for UAV operations.

References

Al-Khalidy, N. (2018). Building Generated Wind Shear and Turbulence Prediction Utilising Computational Fluid Dynamics. *WSEAS Transactions on Fluid Mechanics*, 13, 126-135

Foken, T., & Mauder, M. (2008). *Micrometeorology* (Vol. 2, p. 306). Berlin: Springer.

https://www.dlubal.com/en/downloads-and-information/documents/online-manuals/rwind-3/005640?utm_

Krüß, H. W., Haanstra, J. O., Van Der Ham, R., & Schreur, B. W. (2003). Numerical simulations of wind measurements at Amsterdam Airport Schiphol. *Journal of Wind Engineering and Industrial Aerodynamics*, 91(10), 1215-1223.

RWIND 3 Online Manual (Dlubal) — see “Initial condition” section

<https://www.dlubal.com/en/downloads-and-information/documents/online-manuals/rwind-3/005640>