

Article

CFD Modeling of Near-Field Tornado Flow Using RANS Turbulence Models

Tomasz Lamparski *  and Maciej Dutkiewicz 

Faculty of Civil and Environmental Engineering and Architecture, Bydgoszcz University of Science and Technology, Al. Prof. S. Kaliskiego 7, 85-796 Bydgoszcz, Poland; maciej.dutkiewicz@pbs.edu.pl

* Correspondence: tomasz.lamparski@pbs.edu.pl

Abstract

The primary objective of this study is to model the velocity distribution in the near-field of a tornado-like flow using computational fluid dynamics (CFD) simulations conducted in ANSYS Fluent 2025 R1. A complementary goal is to develop a computational framework capable of analyzing tornado-induced velocity fields while maintaining a simplified model structure that minimizes computational cost and processing time. The paper begins by introducing the phenomenon of extreme wind events, their classification, and representative cases, with particular emphasis on tornado characteristics and the inherent challenges in accurately capturing their dynamics through numerical modeling. Several Reynolds-Averaged Navier–Stokes (RANS) turbulence models were applied and compared with an analytical Rankine-type vortex profile. The methodological section presents the model development process and the comparative evaluation of simulation results. Various modeling approaches were analyzed to minimize numerical errors, with particular attention paid to mesh refinement and grid sensitivity analysis. The results include a comprehensive investigation of velocity distributions across different model configurations, initial wind speeds, computational domain geometries, and radial distances from the tornado core. Based on these analyses, a representative wind velocity profile was formulated. The findings demonstrate that both the selected RANS turbulence model and the computational domain geometry significantly influence the predicted velocity field in the tornado near-field. The study emphasizes the comparative performance of the Spalart–Allmaras, Realizable $k-\epsilon$, and SST $k-\omega$ models, which showed the closest agreement with the analytical vortex profile.

Keywords: tornado-like flow; velocity distribution; strong wind; computational fluid dynamics (CFD); RANS models; Rankine vortex; model optimization



Academic Editor: N.C. Markatos

Received: 11 November 2025

Revised: 12 May 2026

Accepted: 3 July 2026

Published: 1 August 2026

Copyright: © 2026 by the authors.

Licensee MDPI, Basel, Switzerland.

This article is an open access article distributed under the terms and

conditions of the [Creative Commons Attribution \(CC BY\)](https://creativecommons.org/licenses/by/4.0/) license.

1. Introduction

This study focuses on modeling the effects of extreme winds and analyzing the velocity distribution in tornado-like flows [1,2]. As discussed in detail in subsequent sections, parametric analyses of computational domain dimensions were performed to evaluate the influence of domain geometry on the simulation outcomes. These phenomena have been widely investigated by numerous researchers using field measurements, laboratory experiments, and numerical simulations. The following section provides a concise literature review concerning the analysis of tornado-induced flow fields [3].

Field measurements (in situ observations) provide fundamental data on tornado–structure interactions. For instance, Wurman and Kosiba (2013) [4] conducted the first

three-dimensional mapping of the tornado boundary layer and its evolution using mobile Doppler radars. Similarly, Bluestein et al. (2016) [5] employed Doppler radar techniques to investigate pressure gradients during anticyclonic tornadoes. More recent field investigations by Wang et al. [6] examined wind characteristics acting on low-rise buildings, while Feng et al. [7] analyzed wind pressure effects on open roof structures during Typhoons HaiKui and SuLi. A broader review of observed and simulated tornadic wind fields was presented by Honerkamp et al. [8].

Wind tunnel experiments represent the next stage in validating numerical approaches by replicating tornado-like flows under controlled conditions. Haan Jr., Sarkar, and Gallus (2008) [9] designed, built, and tested a laboratory-scale tornado simulator capable of generating vortices with tornado-like characteristics. The simulator employed a “rotating forced downdraft” technique, loosely corresponding to the rear-flank downdraft (RFD), frequently emphasized as critical for tornadogenesis. Haan et al. [10] investigated tornado-induced wind loads acting on low-rise buildings and compared their characteristics with conventional atmospheric wind loading. Similarly, Chen et al. (2023) [11] conducted laboratory investigations of tornado-like loading on low-rise building models, analyzing the influence of vortex structure on aerodynamic response. Additional experimental studies by Sabareesh et al. [12] investigated the influence of building location and ground roughness on pressure distributions within tornado-like vortices. Furthermore, Zhong et al. [13] analyzed the effects of surrounding terrain conditions on wind tunnel simulations of low-rise buildings, highlighting the importance of environmental configuration in aerodynamic testing.

Recent studies [14,15] demonstrate the increasing use of CFD approaches in tornado-like flow simulations. In recent years, numerical analyses and computational simulations have gained prominence due to their relatively lower cost compared to laboratory experiments. They have become a key tool in reproducing complex tornado-like flow structures and analyzing their fundamental characteristics. S Cao, Wang, and J Cao (2018) [16] developed simulations of large-scale tornado-induced flow fields acting on low-rise buildings with flat and gable roofs. Their study, based on the ISU tornado simulator, was compared with laboratory and field data. Verma and Selvam (2021) [17] further advanced this approach by combining all three methodologies—numerical simulations, wind tunnel data, and field measurements—comparing CFD-predicted tornado wind fields against experimental and observational benchmarks, with a particular focus on tangential velocity profiles at specific heights above ground level.

Despite computational limitations, numerical methods provide valuable insights into tornado dynamics and play a vital role in advancing the design of resilient structures capable of withstanding extreme atmospheric conditions.

It should be noted that the present study focuses on an idealized tornado-like flow field and does not include direct simulation of building geometries. References to structural effects are provided primarily for engineering context and motivation.

1.1. Types of Strong Winds

Extreme atmospheric events associated with strong winds [18–21] are characterized by violent air mass movements exceeding 50 m/s. Depending on their intensity and form, they may manifest as gales, hurricanes, or tornadoes. These phenomena differ in strength and geographic occurrence, with their nature largely governed by meteorological conditions such as atmospheric pressure gradients, temperature variations, and topographic features.

The primary driving mechanism behind such events lies in pressure differentials between regions, caused by the uneven heating of the Earth’s surface by solar radiation, which generates atmospheric circulation. Coastal and oceanic regions, for example, frequently experience abrupt changes in temperature and pressure, creating favorable conditions for

the development of severe winds. While certain areas—particularly coastal zones—are more prone to these phenomena due to their geographic and topographic setting, no region is entirely immune to extreme wind events.

Strong winds can be classified into several categories depending on their origin, intensity, and geographic occurrence. This study focuses primarily on tornadoes due to their extreme rotational velocities and destructive impact, but other strong wind events are also relevant:

- **Gales**—Strong winds typically arising from pressure differences between high- and low-pressure systems. They can occur worldwide, particularly in temperate and polar zones. Gales are characterized by wind speeds exceeding 20 m/s, which may lead to infrastructure damage, roof detachment, and tree breakage.
- **Storms and Hurricanes**—Phenomena occurring mainly in tropical and subtropical regions. Hurricanes represent the most intense form of tropical cyclones, developing over warm ocean waters and associated with rapid pressure drops and extremely high wind speeds, sometimes exceeding 70 m/s. Their impact often includes building destruction, flooding, and large-scale infrastructure collapse in both coastal and inland areas.
- **Tornadoes**—Extremely violent vortical flows characterized by intense rotational motion and wind speeds sometimes exceeding 100 m/s. Tornadoes occur most frequently in North America but have also been documented in Europe and Asia. They can completely destroy buildings, uproot trees, and transport heavy objects over long distances.

1.2. Hazards Associated with Strong Wind Events

Many researchers [22–24] have analyzed the potential damage caused by extreme wind events and their influence on structural safety. In particular, Huang et al. (2016) [25] presented a comprehensive overview of tornado hazards in structural engineering, including damage mechanisms associated with extreme wind loading. Understanding the hazards posed by strong winds is essential for assessing structural vulnerability and developing appropriate engineering solutions. The most significant risks include:

- **Structural Damage**—High-velocity winds can severely damage buildings, including roof detachment, wall collapse, and window breakage, posing risks to occupants and nearby individuals.
- **Power Supply Disruptions**—Strong winds may damage power lines and poles, leading to large-scale electricity outages, particularly critical in densely populated areas.
- **Threats to Human Life**—Tornadoes and hurricanes can cause fatalities due to collapsing buildings and airborne debris traveling at high speeds.
- **Engineering Challenges**—Strong winds require engineers and designers to account for wind loads in construction, particularly in regions frequently exposed to extreme atmospheric events.

Given these risks, accurate modeling of strong wind phenomena is essential for predicting their effects on structures and for developing effective mitigation strategies.

2. Modeling of Strong Wind Phenomena

Accurate wind modeling requires a proper understanding of both the wind action acting on a structure [26–28] and the structural response to such loading [29]. Replicating strong wind phenomena under laboratory and computational conditions is essential for understanding their fundamental flow characteristics and supporting the development of engineering-oriented analysis approaches. For this purpose, wind models are employed—advanced tools based on airflow theory and fluid mechanics that enable the analysis of airflow behavior and velocity field characteristics of various shapes and scales.

Numerous studies provide practical recommendations for CFD modeling, including guidelines presented in [30–32], which support the development of models that accurately reproduce real flow conditions. These models aim to capture real-world wind effects on buildings and other structures as accurately as possible, taking into account key factors such as wind speed and direction, flow field variations, turbulence intensity, and terrain characteristics (e.g., surrounding buildings, vegetation, or surface formations). A comprehensive overview of the development of Computational Wind Engineering (CWE) over the last 50 years was presented by Blocken [33].

In this study, parametric analyses of computational domain geometry were performed and compared with a theoretical Rankine-type vortex profile to evaluate domain-size effects and provide verification against an analytical reference. The modeling approach focuses on reproducing a controlled tornado-like flow field using CFD simulations conducted in ANSYS Fluent 2025 R1. The developed models account for wind speed, flow structure, turbulence intensity, and terrain effects.

2.1. Wind Turbulence Phenomena

Turbulence refers to irregular and chaotic air motion occurring across a wide range of spatial and temporal scales. In the context of wind engineering, it significantly affects the velocity field distribution and flow structure and may induce resonance phenomena, particularly dangerous for tall and slender structures [34–36]. Due to its inherently unstable and multiscale nature, turbulence is difficult to model directly. While advanced approaches such as Large Eddy Simulation (LES) or Direct Numerical Simulation (DNS) can capture fine-scale details, their computational cost limits their use mainly to reference studies. For practical engineering applications, turbulence is approximated through reduced-order models embedded in CFD frameworks.

2.2. Reynolds Number and Flow Classification

The Reynolds number (Re) is a dimensionless parameter used to describe the nature of fluid flow, determining the ratio of inertial forces to viscous forces within a given flow. The Reynolds number allows prediction of whether a flow remains laminar (organized) or transitions into a turbulent state characterized by chaotic vortical structures. In the case of airflow around buildings, the Reynolds number depends on wind speed, obstacle size, and air properties [37,38].

Mathematically, the Reynolds number is defined as:

$$Re = \frac{\rho u L}{\mu} \quad (1)$$

where:

- ρ is the fluid (air) density, u is the flow velocity, L is the characteristic length of the obstacle (e.g., building height), μ is the dynamic viscosity of the fluid.

Depending on the Reynolds number, flows can be classified as shown in Table 1.

In building aerodynamics, turbulent regimes ($Re > 10^5$) dominate and dictate the need for turbulence modeling. The Reynolds number is a crucial parameter in wind studies as it allows engineers to predict how different flow regimes will interact with buildings. For example, turbulent flow increases dynamic loads, which may affect the stability and durability of structures [39].

Table 1. Flow regimes based on Reynolds number.

Flow Type	Re	Description
Laminar flow	$\text{Re} < 2 \times 10^3$	Fluid particles move in smooth, parallel layers without vortex formation.
Transitional flow	$2 \times 10^3 < \text{Re} < 10^5$	Intermediate regime where instabilities and irregularities begin to appear.
Turbulent flow	$\text{Re} > 10^5$	Chaotic, irregular motion with numerous vortices strongly affecting loads.

2.3. Navier–Stokes Equations and Their Application in Turbulence Modeling

The Navier–Stokes equations are fundamental partial differential equations describing fluid motion, including airflows in the atmosphere [37,38]. In turbulent flow modeling, Direct Numerical Simulation (DNS) solves these equations without modeling assumptions, while LES resolves large eddies and models smaller scales; both are used here for reference. Formulated by Claude-Louis Navier and George Stokes in the 19th century, these equations are used to model fluid dynamics, encompassing flows of various velocities, densities, and viscosities.

These equations express the principles of mass, momentum, and energy conservation, describing the relationships between pressure, velocity, and internal and external forces acting on fluid particles. Mathematically, the Navier–Stokes equations take the form:

$$\rho \frac{\partial u}{\partial t} + u \dots \nabla u = -\nabla p + \mu \nabla^2 u + f \quad (2)$$

where:

- ρ is the fluid density, u is the fluid velocity vector, t is time, p is pressure, μ is the fluid viscosity, f represents external force vectors.

In the context of turbulence, the Navier–Stokes equations form the basis for describing chaotic flow phenomena. Turbulence is difficult to predict, as it involves continuous, disordered changes in airflow velocity and direction, covering a broad range of scales—from large vortices affecting entire structures to small oscillations. Solving the Navier–Stokes equations for turbulent flows is exceptionally complex and requires significant computational resources. Therefore, turbulence modeling approaches are commonly applied to approximate the effects of complex turbulent flow structures in practical engineering simulations.

The Navier–Stokes equations serve as a fundamental tool in wind flow calculations, enabling the analysis of airflow behavior and velocity field characteristics and providing a more precise understanding of turbulence-related effects.

3. Selection of the Turbulence Model

Replicating complex wind flow phenomena within numerical simulations requires the use of simplified yet physically justified turbulence models and numerical approaches [30–39]. Computational Fluid Dynamics (CFD) provides an effective framework, offering greater flexibility and control compared with laboratory experiments, but requiring careful selection of wind representations and turbulence models. The following subsections present the modeling assumptions applied in this study.

3.1. Wind Models and Their Assumptions

Wind models are developed to approximate real atmospheric conditions and to capture the distribution of velocity fields and flow characteristics. Their assumptions typically include averaged velocity fields, as well as the possibility of parameter adjustments during simulation [30–33]. The most common approaches are:

- $k-\varepsilon$ models—robust and computationally efficient, often applied to simple building forms.
- Large Eddy Simulation (LES)—resolves large turbulent structures, providing higher accuracy for complex flows.
- Direct Numerical Simulation (DNS)—the most accurate but computationally prohibitive, applied mainly for small-scale reference studies.
- Hybrid RANS–LES—combines the efficiency of RANS near walls with LES accuracy in separated flow regions.

In this work, RANS-based approaches were prioritized as the core engineering tool, supplemented by reduced-order vortex models (e.g., Rankine) as an analytical reference for verification purposes.

3.2. Selection of the Turbulence Model

Different turbulence models available in ANSYS Fluent 2025 R1 were considered to reproduce tornado-like flow structures [40,41]. Although no single model is universally superior, each has specific strengths depending on the flow characteristics and computational constraints. Below is a brief overview of the main models:

- Spalart–Allmaras—a one-equation model designed for aerodynamic applications, efficient for boundary layers with adverse pressure gradients.
- $k-\varepsilon$ family (including the Realizable $k-\varepsilon$ variant)—robust, widely used in engineering; effective for general engineering flows but less accurate in regions with strong flow separation.
- $k-\omega$ family—better near-wall resolution and improved prediction of flow separation; the SST $k-\omega$ variant is the most reliable and commonly applied.
- Reynolds Stress Model (RSM)—accounts for turbulence anisotropy, offering higher accuracy but at greater computational cost.
- Large Eddy Simulation (LES)—resolves large turbulent structures, highly accurate but computationally demanding, mainly for research purposes.
- Hybrid RANS–LES models—combine RANS efficiency near walls with LES accuracy in separated regions, useful for complex flows at high Reynolds numbers.

Among the analyzed approaches, the $k-\varepsilon$ and SST $k-\omega$ models demonstrated the most favorable balance between computational efficiency and agreement with the analytical vortex profile, making them the most suitable engineering-oriented solutions in the present study. The selection of RANS-based models was motivated by the need to achieve a balance between computational efficiency and the ability to reproduce key features of tornado-like flow structures. These models are widely used in engineering applications and provide stable and numerically consistent results for simplified flow configurations.

All turbulence models were applied using standard ANSYS Fluent implementations with default model constants. Table 2 summarizes the principal model constants and calculation parameters applied in the analyzed turbulence models, including the standard coefficients used in the $k-\varepsilon$ family.

Table 2. List of models and their most important parameters.

Model	Parameters
Spalart–Allmaras	$Cb1 = 0.1355$; $Cb2 = 0.622$; $Cv1 = 7.1$; $Cw2 = 0.3$; $Cw3 = 2.0$
k–epsilon	$Cmu = 0.09$; $C1\text{-Epsilon} = 1.44$; $C2\text{-Epsilon} = 1.92$; $TKE = 1.0$; $TDR = 1.3$
k–omega	$\alpha^*_{inf} = 1.0$; $\alpha_{inf} = 0.52$; $\beta^*_{inf} = 0.09$; $a1 = 0.31$; $TKE\ (inner) = 1.176$; $TKE\ (outer) = 1.0$
k–kl–omega	$Cmu = 0.09$; $C\text{-lambda} = 2.495$; $CR = 0.12$; $ANAT = 200$; $Cw1 = 0.44$; $Cw3 = 0.3$
SST	$\alpha^*_{inf} = 1.0$; $\alpha_{inf} = 0.52$; $\beta^*_{inf} = 0.09$; $a1 = 0.31$; $Ca1 = 2.0$; $Ca2 = 0.06$; $Ce1 = 1.0$; $Ce2 = 50.0$
Reynolds Stress	$Cmu = 0.09$; $C1\text{-Epsilon} = 1.44$; $C2\text{-Epsilon} = 1.92$; $C1\text{-PS} = 1.8$; $C2\text{-PS} = 0.6$; $C1'\text{-PS} = 0.5$; $C2'\text{-PS} = 0.3$

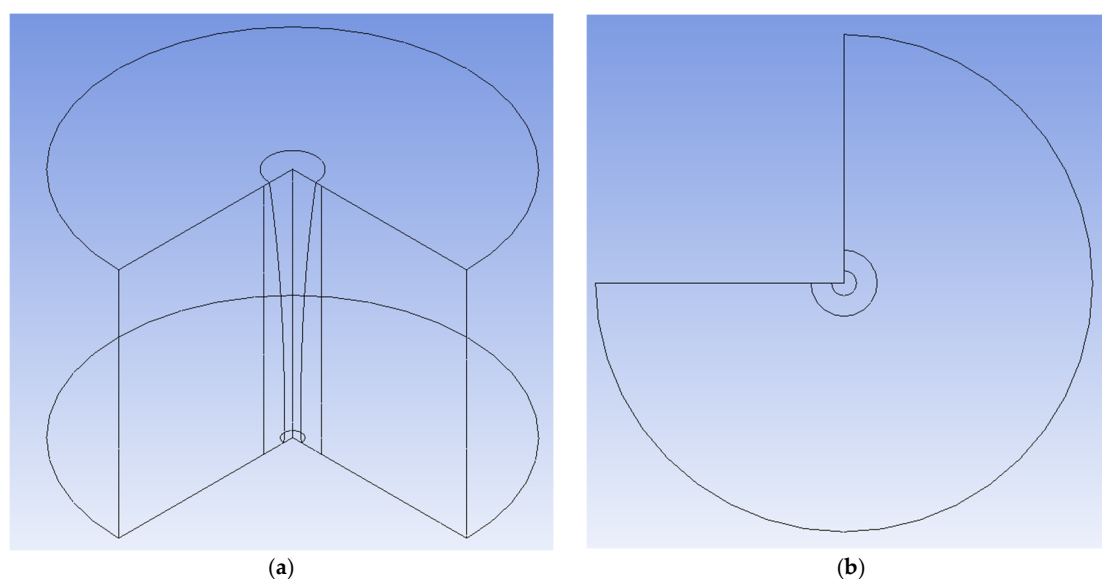
4. Computational Process

The following section outlines the methodology applied in the development of the computational model used in this study. While earlier parts of the paper addressed the theoretical background and classification of wind phenomena, here the focus is shifted toward the practical implementation of these concepts in CFD simulations. The process involved defining the computational domain, testing its dimensions, selecting representative wind velocities, and optimizing the numerical mesh. The aim was to balance accuracy with computational efficiency, ensuring reliable results without excessive computational cost [42–44].

The simulations were performed under steady-state conditions using a pressure-based solver with second-order discretization schemes. The flow was assumed incompressible and isothermal; therefore, the energy equation was not solved.

4.1. Model Presentation and Development Process

The computational domain was constructed in the form of a three-quarter cylinder with a radius of 60 m (diameter 120 m) and a height of 80 m. One-fourth of the cylinder base was removed, creating an opening angle of 270° [Figure 1]. This configuration created two perpendicular vertical planes with dimensions 120×80 m, aligned with the X and Y axes of the cylindrical cross-section. These planes were designated as the inlet and outlet surfaces, enabling precise control of velocity conditions and their variation as a function of the distance from the cylinder center [Figure 2].

**Figure 1.** 3D model of the medium consisting of a 3/4 cylinder: (a) 3D view, (b) top view.

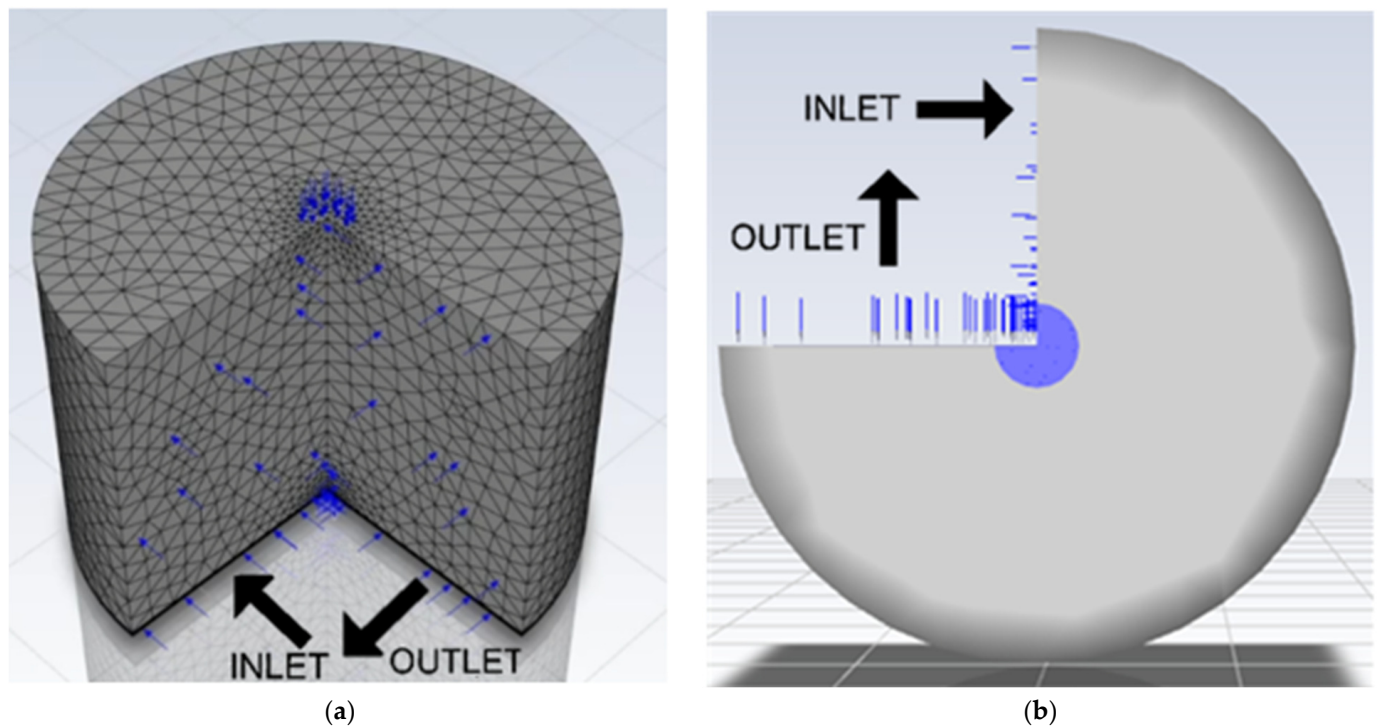


Figure 2. Inlet and outlet planes: (a) 3D view, (b) top view.

At the center of the domain, a tornado funnel was modeled with a tapering geometry: a lower base diameter of 2 m and an upper base diameter of 8 m. As with the cylinder, one-fourth of the funnel was removed to expose its vertical cross-section. Figure 1 illustrates the volumetric model of the domain, composed of a 3/4-cylinder with a visible tornado funnel at its center.

Two fundamental particle motion directions were introduced to represent tornado dynamics: horizontal circular motion around the vertical axis and vertical uplift within the funnel. The imposed velocity field consisted of a tangential component defined at the inlet boundaries and a vertical velocity component within the funnel region, enabling the formation of a vortex-like flow structure.

The imposed boundary conditions accounted for both components, generating resultant vectors that represented the resultant motion of air particles [Figure 2]. This configuration ensured realistic flow representation in the vicinity of the tornado core. Figure 2 illustrates the designated inlet and outlet planes.

The inlet boundary conditions were defined using a prescribed tangential velocity profile based on an analytical vortex formulation. Turbulence parameters were specified using a turbulence intensity of 5% and a viscosity ratio of 10. The ground surface was modeled as a no-slip wall, while the remaining boundaries were defined using pressure-outlet or symmetry-type conditions to minimize artificial disturbances within the computational domain.

The software used enables real-time monitoring of input data through visualization of particle trajectory paths in a transparent manner. Figure 3 presents the trajectory, with line colors indicating velocity intensity (yellow—slower, orange—faster).

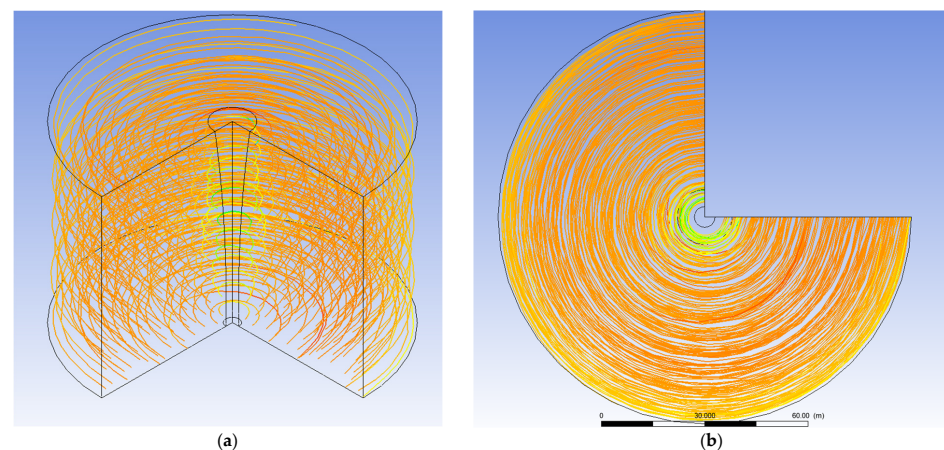


Figure 3. Trajectory of particle displacements: (a) 3D view, (b) top view.

4.2. Selection of Computational Domain Dimensions

To evaluate the sensitivity of the results to domain size, a series of parametric simulations was carried out using cylinders with varying radii ($R = 60, 120, 240$, and 480 m) and heights ($H = 80, 120, 160$, and 200 m). The results demonstrated that enlarging the computational domain beyond $R = 60$ m and $H = 80$ m produced only minor differences in velocity distributions—typically within 5%.

Since larger domains significantly increased computational costs without offering substantial improvements in accuracy, the domain size of $R = 60$ m and $H = 80$ m was adopted for all subsequent analyses. This choice ensured a balance between numerical fidelity and efficiency, enabling reliable representation of tornado-like flow structures while minimizing computational time.

Representative results illustrating the limited influence of domain enlargement on velocity predictions are presented in Table 3.

Table 3. Comparison of velocity values [m/s] and relative variability parameters obtained for different computational domain dimensions at selected radial distances from the vortex center.

Case	D [m]	Distance [m]	Maximum Speed Depending on Altitude $v(h)$ [m]/Wind Speed Relative Variability Parameter [%]											
			H = 1 m	H = 5 m	H = 10 m	H = 15 m	H = 20 m	H = 30 m	H = 40 m	H = 50 m	H = 60 m	H = 70 m	H = 75 m	H = 79 m
A1.1	120	24.0	13.86	14.09	13.91	13.95	13.93	13.81	13.28	13.52	13.56	13.67	13.71	13.37
		Wind speed relative variability parameter	1%	5%	5%	4%	3%	2%	6%	3%	5%	4%	3%	4%
A1.2	240	24.0	13.30	14.53	14.62	14.50	14.42	14.18	14.08	14.04	14.16	14.01	14.04	14.03
		Wind speed relative variability parameter	5%	2%	0%	0%	0%	−1%	0%	0%	1%	1%	1%	0%
A1.3	480	24.0	14.76	15.71	15.31	15.14	14.91	14.27	15.04	14.40	15.00	14.90	14.86	14.59
		Wind speed relative variability parameter	−6%	−6%	−5%	−4%	−3%	−1%	−6%	−3%	−5%	−5%	−5%	−4%
		Average	13.97	14.78	14.61	14.53	14.42	14.09	14.13	13.99	14.24	14.19	14.20	13.99
A1.2	120	16.0	18.10	18.23	18.10	17.68	17.31	16.50	15.47	16.09	16.62	17.04	17.34	17.38
		Wind speed relative variability parameter	1%	4%	3%	2%	3%	−2%	2%	−3%	2%	3%	4%	3%
A1.2	240	16.0	18.47	19.22	18.90	18.13	17.92	16.37	16.03	15.63	16.56	17.61	17.85	17.79
		Wind speed relative variability parameter	−1%	−2%	−1%	−1%	−1%	−1%	−1%	0%	2%	0%	2%	1%
A1.3	480	16.0	18.44	19.26	19.04	18.28	18.22	15.86	16.01	15.00	17.48	18.16	19.20	18.82
		Wind speed relative variability parameter	−1%	−2%	−2%	−1%	−2%	2%	−1%	4%	−4%	−3%	−6%	−5%
		Average	18.34	18.90	18.68	18.03	17.81	16.25	15.84	15.57	16.89	17.61	18.13	18.00

The analysis confirms that the selected domain size ensures stable velocity results without significant influence of the domain boundaries.

4.3. Wind Velocity Considered in the Analysis

The analysis focused on velocity distribution patterns, which required testing different inflow velocities to determine their influence on the results [32–34]. Three cases were considered based on standard tornado velocity classifications derived from applicable design guidelines and established velocity scales, such as the Enhanced Fujita Scale [45]:

- (a) 22 m/s—normative wind speed according to the applicable standard for wind load design in Poland [46],
- (b) 50 m/s—velocity at the boundary between EF1 and EF2 classifications,
- (c) 90 m/s—velocity corresponding to extreme EF5 tornado conditions.

Table 4 categorizes wind groups based on velocity and the associated potential damage.

Table 4. Wind Categories Defined by the Enhanced Fujita (EF) Scale [35].

Category	Wind Speed [m/s]	Damage Description
EF0	29 to 38	Roof damage to houses, branches torn off trees
EF1	38.5 to 49.5	Roofs torn off, overturned and destroyed caravans
EF2	50 to 60.5	Roofs torn off from solid structures, large trees uprooted, light cars lifted off the ground
EF3	61 to 74	Whole floors of solid houses destroyed, damage to large buildings, trains derailed, heavier cars lifted off the ground
EF4	74.5 to 89.5	Solid houses razed to the ground, cars thrown into the air
EF5	Above 90	Houses with a very strong skeleton razed to the ground down to the foundations, cars carried up to 100 m, skyscrapers with deformed structures

For the maximum velocity of 90 m/s, the corresponding Mach number remains below 0.3, which justifies the assumption of incompressible flow in the present study.

4.4. Mesh Optimization

The accuracy and stability of numerical simulations in CFD are strongly dependent on the quality of the applied mesh. A computational mesh divides the analyzed domain into finite elements, enabling the solution of governing fluid dynamics equations. The mesh must be sufficiently refined to capture flow gradients, turbulence structures, and interactions with solid boundaries, while maintaining acceptable computational efficiency.

Several types of meshes are available in CFD practice, including structured meshes (regular grids), unstructured meshes (triangular or tetrahedral elements), and hybrid meshes combining both approaches. For the purposes of this study, an unstructured triangular mesh was selected, as shown in Figure 4. This approach provided flexibility in accurately reproducing the complex geometry of the domain and the tornado funnel.

The global element size was set to approximately 9 m, while additional refinement (inflation layers) was introduced at the ground boundary to properly capture near-wall velocity gradients and shear stresses. The inflation method was applied with a smooth transition, five inflation layers, and a transition ratio of 0.27, ensuring continuity between the boundary region and the core flow field.

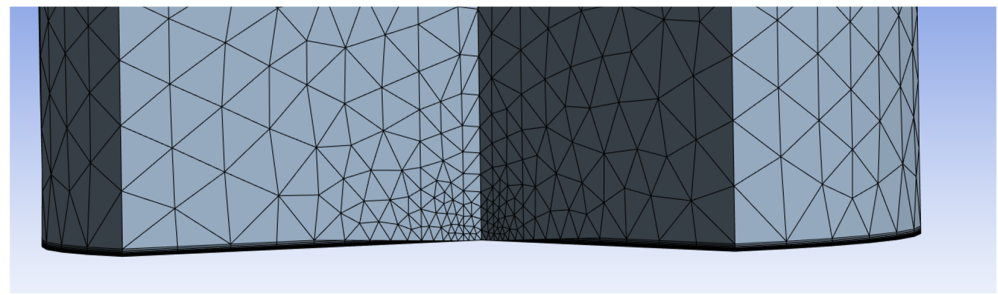


Figure 4. The model adopts a triangular mesh with density in the central part and inflation at the ground level.

To determine the optimal mesh configuration, a grid sensitivity analysis was performed using multiple mesh densities. The comparison of velocity profiles obtained for successive refinements indicates that further mesh densification does not significantly affect the results, confirming the adequacy of the adopted mesh configuration. Table 5 summarizes the principal mesh configurations considered during the grid sensitivity analysis, including the approximate number of elements, characteristic element size, and selected velocity values obtained at representative locations within the computational domain. The comparison confirms that further mesh refinement produced only minor variations in the predicted velocity field while significantly increasing the computational requirements. Consequently, the adopted mesh was considered sufficient for the purposes of the present engineering-oriented analysis.

Table 5. Summary of mesh configurations and selected velocity results obtained during the grid sensitivity analysis.

Case	Max Element Size [m]	Distance [m]	Maximum Speed Depending on Altitude $v(h)$ [m]/Wind Speed Relative Variability Parameter [%]											
			H = 1 m	H = 5 m	H = 10 m	H = 15 m	H = 20 m	H = 30 m	H = 40 m	H = 50 m	H = 60 m	H = 70 m	H = 75 m	H = 79 m
A2.1	15	24.0	14.76	15.71	15.31	15.14	14.91	14.27	15.04	14.40	15.00	14.90	14.86	14.59
	Wind speed relative variability parameter		−3%	−6%	−5%	−5%	−4%	−2%	−6%	−3%	−5%	−4%	−4%	−4%
A2.2	12	24.0	13.99	14.57	14.41	14.06	13.90	13.68	14.30	13.82	14.29	14.10	14.03	13.85
	Wind speed relative variability parameter		2%	1%	1%	3%	3%	3%	−1%	1%	0%	1%	2%	1%
A2.3	10	24.0	14.25	14.45	14.3	14.35	14.33	14.15	13.75	13.95	14	14.1	14.2	13.85
	Wind speed relative variability parameter		0%	2%	2%	1%	0%	−1%	3%	0%	2%	1%	1%	1%
A2.4	9.4	24.0	14.2	14.4	14.25	14.32	14.3	14.1	13.7	13.9	13.95	14.05	14.15	13.8
	Wind speed relative variability parameter		1%	3%	2%	1%	0%	0%	4%	1%	3%	2%	1%	2%
	Average		14.30	14.78	14.57	14.47	14.36	14.05	14.20	14.02	14.31	14.29	14.31	14.02
A2.1	15	16.0	19.72	19.59	19.30	18.91	18.45	16.11	17.03	15.82	17.01	16.78	17.28	16.95
	Wind speed relative variability parameter		−6%	−3%	−3%	−3%	−3%	3%	−5%	2%	0%	2%	2%	3%
A2.2	12	16.0	18.00	18.80	18.43	18.10	17.65	16.56	16.13	15.89	17.19	16.99	17.59	17.23
	Wind speed relative variability parameter		4%	1%	2%	1%	2%	1%	1%	2%	−1%	1%	0%	1%
A2.3	10	16.0	18.55	18.7	18.6	18.15	17.85	17	15.95	16.6	17.1	17.55	17.85	17.9
	Wind speed relative variability parameter		1%	1%	1%	1%	0%	−2%	2%	−2%	0%	−2%	−1%	−2%
A2.4	9.4	16.0	18.5	18.65	18.55	18.1	17.8	16.95	15.9	16.55	17.05	17.5	17.8	17.85
	Wind speed relative variability parameter		1%	2%	1%	1%	1%	−2%	2%	−2%	0%	−2%	−1%	−2%
	Average		18.69	18.94	18.72	18.31	17.94	16.65	16.25	16.21	17.09	17.21	17.63	17.48

4.5. Adopted Velocity Distribution Model

Verification of the developed CFD model is based on comparison with a theoretical reference to ensure reliability of numerical predictions in extreme tornado-like flows [47]. In this study, the velocity field generated in the simulations was confronted with the Rankine vortex profile, widely used to approximate tangential velocity distributions in rotating flows [48]. The adopted analytical form was:

$$V = V_{max} * \left(\frac{r}{R_{max}} \right)^{\varphi}, \text{ for } r \leq 10 \text{ m} \quad (3)$$

$$V = V_{max} * \left(\frac{R_{max}}{r} \right)^{\varphi}, \text{ for } r > 10 \text{ m} \quad (4)$$

where $R_{max} = 10 \text{ m}$, $\varphi = 0.6$.

Figure 5 presents the theoretical velocity distribution described by the Rankine vortex model. The profile combines two flow regions: the inner solid-body rotation zone, where velocity increases linearly with distance from the vortex center, and the outer region, where velocity decreases inversely with radius. The maximum tangential velocity occurs at the transition point between these regions (R_{max}), marking the boundary between the core and the free vortex zone. This idealized model provides a simplified yet fundamental representation of tornado-like flow behavior, serving as a theoretical reference for subsequent CFD analyses.

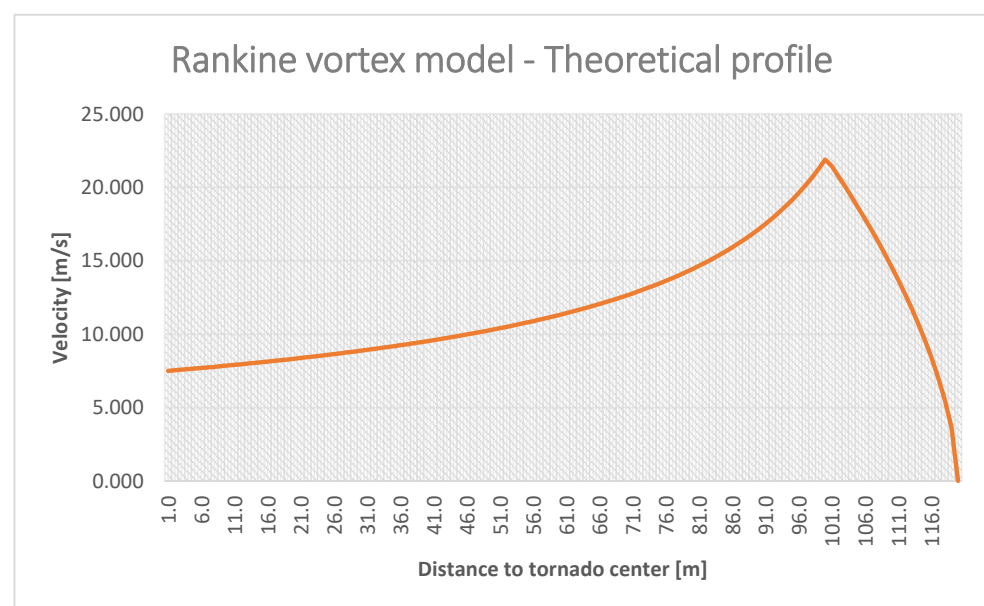


Figure 5. Theoretical Rankine vortex velocity profile.

This verification step supports the credibility of the subsequent analyses focused on velocity field distributions. By ensuring that the velocity field is consistent with theoretical predictions, the developed CFD framework provides a reliable basis for the interpretation of the results presented in Section 5.

5. Results

Having verified the CFD model against an analytical Rankine-type vortex profile, the next step is to analyze the obtained simulation outputs in detail [47,48]. The obtained velocity field provides a reliable basis for further analysis of tornado-like flow distributions, which are essential for evaluating the behavior of tornado-like flow structures. To ensure

that the presented outcomes are both consistent and comparable, a systematic approach to interpreting the results is required.

5.1. Method of Interpreting Results

It is not always possible to unequivocally determine whether a given model is fully accurate or whether it is subject to limitations. One of the most common issues involves disturbances resulting from limitations inherent to simplified numerical models [47,48]. In practice, it may turn out that the studied medium is insufficiently large or that the ratio of vertical to horizontal velocity is inadequate. To minimize these disturbances and to ascertain with greater certainty that the created model corresponds to real conditions, different variants should be examined and the obtained results compared, paying particular attention not only to the values but primarily to their distribution. Considering the above statement, it is necessary to establish a clear method for reading the results that allows for comparative analysis.

In the described model, a vertical plane was introduced, positioned perpendicularly to the inlet plane, and thus located at one-third of the distance traveled by the moving air particles within the model. Subsequently, on this prepared plane, horizontal lines were introduced at different heights along the X-axis [Figure 6].

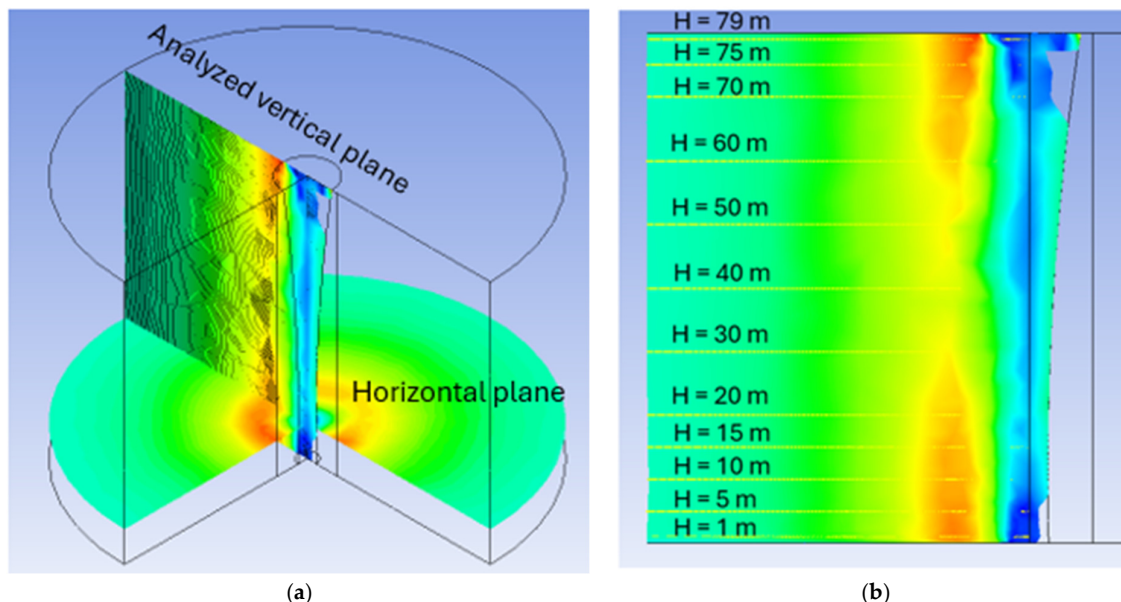


Figure 6. Location and method of data extraction: (a) 3D view, (b) analyzed plane view.

5.2. Summary of Results—Main Assumptions

The primary objective of this study is to compare available RANS models based on the obtained wind velocity results at selected points (at different heights) on the analyzed vertical plane while maintaining identical external data, such as the studied medium, assumed mesh, and ground resistance parameters.

Theoretical Rankine model predictions were used as a baseline reference when assigning case nomenclature and analyzing deviations. Various computational models are available in Ansys Fluent software. To systematize and simplify further nomenclature, each model under specific conditions was assigned a case name. The cases are as follows: A3: Spalart–Allmaras, A4: k–epsilon, A5: k–omega, A6: k–kl–omega, A7: SST, A8: Reynolds Stress.

In the next phase, the created models were modified by changing the inlet and outlet velocities. The initial velocity of 22 m/s serves as a reference in accordance with the Polish wind standard [46]. However, numerical models allow for extending research beyond

standard regulations by increasing wind velocity. In the second phase of the analysis, the assumed velocities were set at 50 m/s and 90 m/s, corresponding respectively to a change in the EF group from 1 to 2, representing common strong winds, and exceeding the EF5 group, which corresponds to extreme recorded phenomena.

5.3. Individual Results

The following section presents detailed research results obtained during numerical simulations, along with a basic statistical analysis of the obtained results by means of indicating the relative measurement error.

Wind speed relative variability parameter [%] at height h , $\epsilon(h)$, is defined as:

$$\epsilon = \frac{\Delta v}{v} * 100\% = \frac{|v - v_0|}{v} * 100\% \quad (5)$$

where v_0 is the maximum wind speed at height h and distance R from the vortex center, and v is the average of maximum speeds over all heights at the same radial position R .

This parameter does not represent a classical error with respect to experimental or theoretical data, but rather an indicator of velocity variation across different heights at a given radial position.

Tables 6–11 display wind velocity values at different locations (changing distance from the vortex center R and height above ground H) on the analyzed vertical plane. Each result includes a relative measurement error expressed as a percentage. This value refers to the average velocity at the same distance from the vortex center R but at different heights. The parameter reflects differences in the obtained velocity depending on the height above the ground.

Table 6. Wind speed values on the analyzed vertical plane and relative measurement error compared to other heights. Case A3 (Spalart–Allmaras model, inlet velocity = 22 m/s), heights from $H = 1$ m to $H = 79$ m.

Distance R	Maximum Speed Depending on Altitude $v(h)$ [m]/Wind Speed Relative Variability Parameter [%]											
	H = 1 m		H = 5 m		H = 10 m		H = 15 m		H = 20 m		H = 30 m	
[m]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]
60	7.73	0%	7.77	−1%	7.80	−1%	7.76	0%	7.73	0%	7.73	0%
55	8.03	0%	8.14	−1%	8.19	−2%	8.12	−1%	8.09	0%	8.09	0%
50	8.34	2%	8.60	−1%	8.64	−2%	8.54	−1%	8.56	−1%	8.59	−1%
45	9.18	−1%	9.23	−1%	9.19	−1%	9.18	−1%	9.17	−1%	9.14	0%
40	9.97	−1%	9.96	−1%	9.97	−1%	9.98	−1%	9.98	−1%	9.93	−1%
35	10.75	0%	10.80	−1%	10.79	−1%	10.84	−1%	10.82	−1%	10.88	−1%
30	11.42	3%	11.96	−2%	11.91	−2%	11.88	−1%	11.77	0%	11.87	−1%
25	12.89	1%	13.19	−1%	13.35	−2%	13.33	−2%	13.19	−1%	13.07	0%
20	15.56	−4%	15.36	−2%	15.56	−4%	15.34	−2%	15.08	−1%	14.51	3%
15	18.60	−8%	18.37	−6%	18.11	−5%	17.91	−4%	17.40	−1%	16.13	7%
10	18.78	−9%	18.49	−8%	17.95	−5%	17.47	−2%	16.74	2%	14.30	17%
Distance R	Maximum Speed Depending on Altitude $v(h)$ [m]/Wind Speed Relative Variability Parameter [%]											
	H = 40 m		H = 50 m		H = 60 m		H = 70 m		H = 75 m		H = 79 m	
[m]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]
60	7.69	1%	7.72	0%	7.72	0%	7.62	1%	7.62	1%	7.81	−1%
55	7.99	1%	8.04	0%	8.03	0%	7.97	1%	7.96	1%	8.03	0%

Table 6. Cont.

50	8.49	0%	8.40	1%	8.44	1%	8.49	0%	8.44	1%	8.45	1%
45	9.04	1%	9.04	1%	8.96	2%	9.14	0%	9.04	1%	9.05	1%
40	9.82	0%	9.77	1%	9.68	2%	9.67	2%	9.73	1%	9.76	1%
35	10.69	0%	10.83	−1%	10.64	1%	10.49	2%	10.65	1%	10.63	1%
30	11.54	2%	11.91	−2%	11.65	1%	11.57	1%	11.74	0%	11.52	2%
25	12.99	1%	13.12	0%	13.06	0%	12.92	1%	13.07	0%	12.78	2%
20	14.36	4%	14.58	3%	14.93	0%	14.80	1%	14.89	1%	14.91	1%
15	15.49	10%	16.08	7%	16.94	2%	17.29	0%	17.51	−1%	17.61	−2%
10	14.13	18%	14.04	18%	15.84	8%	18.06	−5%	19.67	−15%	20.57	−20%

Table 7. Wind speed values on the analyzed vertical plane and relative measurement error compared to other heights. Case A4 (k-epsilon model, initial velocity 22 m/s), heights from H = 1 m to H = 79 m.

Distance R	Maximum Speed Depending on Altitude v(h) [m]/Wind Speed Relative Variability Parameter [%]											
	H = 1 m		H = 5 m		H = 10 m		H = 15 m		H = 20 m		H = 30 m	
[m]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]
60	9.88	−23%	9.39	−17%	8.41	−5%	7.81	3%	7.48	7%	7.49	7%
55	9.32	−15%	8.87	−9%	8.33	−2%	7.82	4%	7.77	5%	7.84	4%
50	8.95	−6%	8.80	−4%	8.38	0%	8.11	4%	8.17	3%	8.36	1%
45	9.68	−7%	9.24	−2%	8.91	1%	8.78	3%	8.84	2%	8.95	1%
40	10.66	−8%	10.41	−5%	10.11	−2%	9.73	1%	9.78	1%	9.78	1%
35	12.48	−14%	11.68	−7%	11.01	−1%	10.75	1%	10.71	2%	10.76	1%
30	13.62	−14%	12.80	−7%	12.08	−1%	11.90	0%	11.74	2%	11.78	2%
25	14.18	−7%	13.54	−3%	13.49	−2%	13.40	−2%	13.21	0%	13.01	1%
20	15.72	−5%	15.50	−3%	15.66	−4%	15.41	−3%	15.11	−1%	14.47	3%
15	18.73	−8%	18.49	−7%	18.22	−5%	17.99	−4%	17.41	−1%	16.12	7%
10	19.00	−10%	18.70	−9%	18.11	−5%	17.59	−2%	16.81	2%	14.34	17%

Distance R	Maximum Speed Depending on Altitude v(h) [m]/Wind Speed Relative Variability Parameter [%]											
	H = 40 m		H = 50 m		H = 60 m		H = 70 m		H = 75 m		H = 79 m	
[m]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]
60	7.58	6%	7.67	5%	7.69	4%	7.60	5%	7.61	5%	7.78	3%
55	7.87	3%	7.97	2%	7.98	2%	7.94	2%	7.93	3%	8.00	2%
50	8.36	1%	8.32	1%	8.38	0%	8.45	0%	8.40	0%	8.41	0%
45	8.91	1%	8.96	1%	8.91	1%	9.09	−1%	9.00	0%	9.02	0%
40	9.70	2%	9.69	2%	9.62	3%	9.63	3%	9.69	2%	9.73	2%
35	10.59	3%	10.76	1%	10.58	3%	10.45	4%	10.62	3%	10.59	3%
30	11.47	4%	11.84	1%	11.58	3%	11.53	4%	11.68	2%	11.47	4%
25	12.93	2%	13.04	1%	13.00	1%	12.87	2%	12.99	2%	12.68	4%
20	14.31	5%	14.52	3%	14.86	1%	14.71	2%	14.82	1%	14.83	1%
15	15.45	11%	16.02	7%	16.80	3%	17.20	0%	17.44	−1%	17.55	−2%
10	14.12	18%	14.08	18%	15.80	8%	17.99	−5%	19.54	−14%	20.36	−18%

Table 8. Wind speed values on the analyzed vertical plane and relative measurement error compared to other heights. Case A5 (k-omega model, initial velocity 22 m/s), heights from H = 1 m to H = 79 m.

Distance R	Maximum Speed Depending on Altitude v(h) [m]/Wind Speed Relative Variability Parameter [%]											
	H = 1 m		H = 5 m		H = 10 m		H = 15 m		H = 20 m		H = 30 m	
[m]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]
60	7.73	0%	7.77	−1%	7.81	−1%	7.77	−1%	7.73	0%	7.75	0%
55	7.92	1%	8.10	−1%	8.18	−2%	8.11	−1%	8.09	−1%	8.09	−1%
50	8.12	4%	8.54	−1%	8.62	−2%	8.53	−1%	8.55	−1%	8.58	−1%
45	8.95	1%	9.16	−1%	9.18	−1%	9.17	−1%	9.16	−1%	9.13	−1%
40	9.75	1%	9.91	−1%	9.97	−2%	9.97	−2%	9.98	−2%	9.92	−1%
35	10.65	1%	10.80	−1%	10.79	−1%	10.84	−1%	10.81	−1%	10.87	−1%
30	11.43	2%	11.93	−2%	11.90	−2%	11.87	−1%	11.76	0%	11.85	−1%
25	12.96	1%	13.17	−1%	13.34	−2%	13.32	−2%	13.17	−1%	13.04	0%
20	15.62	−4%	15.34	−3%	15.56	−4%	15.33	−2%	15.06	−1%	14.48	3%
15	18.61	−8%	18.37	−7%	18.09	−5%	17.89	−4%	17.35	−1%	16.10	7%
10	18.76	−9%	18.48	−8%	17.91	−5%	17.44	−2%	16.68	3%	14.31	16%

Distance R	Maximum Speed Depending on Altitude v(h) [m]/Wind Speed Relative Variability Parameter [%]											
	H = 40 m		H = 50 m		H = 60 m		H = 70 m		H = 75 m		H = 79 m	
[m]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]
60	7.69	0%	7.72	0%	7.72	0%	7.61	1%	7.61	1%	7.79	−1%
55	7.98	1%	8.03	0%	8.02	0%	7.96	1%	7.94	1%	8.02	0%
50	8.47	0%	8.39	1%	8.42	0%	8.47	0%	8.42	0%	8.43	0%
45	9.03	1%	9.03	1%	8.94	1%	9.12	0%	9.03	1%	9.04	0%
40	9.80	0%	9.75	1%	9.66	2%	9.65	2%	9.72	1%	9.75	1%
35	10.67	0%	10.82	−1%	10.61	1%	10.48	2%	10.64	1%	10.62	1%
30	11.53	2%	11.88	−1%	11.61	1%	11.56	1%	11.70	0%	11.50	2%
25	12.97	1%	13.08	0%	13.03	0%	12.89	1%	13.01	0%	12.70	3%
20	14.33	4%	14.54	3%	14.87	1%	14.72	2%	14.83	1%	14.83	1%
15	15.44	10%	16.03	7%	16.84	2%	17.19	0%	17.42	−1%	17.54	−2%
10	14.07	18%	14.08	18%	15.89	7%	18.06	−5%	19.57	−14%	20.34	−19%

Table 9. Wind speed values on the analyzed vertical plane and relative measurement error compared to other heights. Case A6 (k-kl-omega model, initial velocity 22 m/s), heights from H = 1 m to H = 79 m.

Distance R	Maximum Speed Depending on Altitude v(h) [m]/Wind Speed Relative Variability Parameter [%]											
	H = 1 m		H = 5 m		H = 10 m		H = 15 m		H = 20 m		H = 30 m	
[m]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]
60	7.62	1%	7.74	0%	7.87	−2%	7.80	−1%	7.75	0%	7.74	0%
55	7.66	4%	8.05	0%	8.23	−3%	8.13	−1%	8.11	−1%	8.10	−1%
50	7.69	9%	8.46	0%	8.66	−3%	8.55	−1%	8.56	−2%	8.59	−2%
45	8.54	6%	9.09	−1%	9.20	−2%	9.18	−2%	9.17	−1%	9.14	−1%
40	9.34	4%	9.79	0%	9.99	−2%	9.99	−2%	9.99	−2%	9.93	−2%

Table 9. Cont.

35	10.26	4%	10.72	0%	10.81	−1%	10.85	−2%	10.83	−1%	10.87	−2%
30	10.85	7%	11.92	−2%	11.93	−2%	11.90	−2%	11.77	−1%	11.85	−2%
25	12.22	6%	13.24	−2%	13.38	−3%	13.35	−3%	13.19	−1%	13.05	0%
20	14.90	0%	15.42	−3%	15.60	−5%	15.36	−3%	15.08	−1%	14.48	3%
15	18.13	−5%	18.44	−7%	18.10	−5%	17.90	−4%	17.38	−1%	16.09	6%
10	18.51	−8%	18.35	−7%	17.84	−4%	17.35	−1%	16.63	3%	14.28	17%
Distance R	Maximum Speed Depending on Altitude v(h) [m]/Wind Speed Relative Variability Parameter [%]											
	H = 40 m		H = 50 m		H = 60 m		H = 70 m		H = 75 m		H = 79 m	
[m]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]
60	7.69	0%	7.71	0%	7.71	0%	7.60	2%	7.60	1%	7.78	−1%
55	7.99	0%	8.03	0%	8.01	0%	7.95	1%	7.93	1%	8.00	0%
50	8.48	−1%	8.39	0%	8.42	0%	8.47	−1%	8.41	0%	8.42	0%
45	9.03	0%	9.02	0%	8.94	1%	9.12	−1%	9.02	0%	9.03	0%
40	9.81	0%	9.75	0%	9.66	1%	9.65	1%	9.71	1%	9.74	0%
35	10.67	0%	10.81	−1%	10.60	1%	10.47	2%	10.64	0%	10.61	1%
30	11.53	1%	11.89	−2%	11.60	1%	11.55	1%	11.69	0%	11.49	1%
25	12.97	0%	13.09	−1%	13.03	0%	12.88	1%	12.99	0%	12.68	2%
20	14.33	4%	14.53	2%	14.86	0%	14.69	1%	14.80	1%	14.79	1%
15	15.46	10%	16.03	7%	16.89	2%	17.15	0%	17.38	−1%	17.48	−2%
10	14.08	18%	14.09	18%	16.02	6%	18.15	−6%	19.62	−15%	20.32	−19%

Table 10. Wind speed values on the analyzed vertical plane and relative measurement error compared to other heights. Case A7 (SST model, initial velocity 22 m/s), heights from H = 1 m to H = 79 m.

Distance R	Maximum Speed Depending on Altitude v(h) [m]/Wind Speed Relative Variability Parameter [%]											
	H = 1 m		H = 5 m		H = 10 m		H = 15 m		H = 20 m		H = 30 m	
[m]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]
60	7.76	0%	7.79	−1%	7.81	−1%	7.76	0%	7.73	0%	7.74	0%
55	7.95	1%	8.11	−1%	8.18	−2%	8.11	−1%	8.09	−1%	8.09	−1%
50	8.16	4%	8.55	−1%	8.63	−2%	8.53	−1%	8.55	−1%	8.58	−1%
45	9.01	1%	9.18	−1%	9.18	−1%	9.18	−1%	9.16	−1%	9.13	−1%
40	9.82	0%	9.92	−1%	9.97	−1%	9.98	−2%	9.98	−2%	9.92	−1%
35	10.74	0%	10.81	−1%	10.79	−1%	10.84	−1%	10.81	−1%	10.87	−1%
30	11.49	2%	11.96	−2%	11.90	−2%	11.88	−1%	11.76	0%	11.85	−1%
25	12.97	1%	13.18	−1%	13.35	−2%	13.33	−2%	13.18	−1%	13.05	0%
20	15.64	−4%	15.35	−3%	15.58	−4%	15.34	−2%	15.07	−1%	14.48	3%
15	18.60	−8%	18.38	−7%	18.10	−5%	17.89	−4%	17.36	−1%	16.10	7%
10	18.75	−9%	18.47	−8%	17.91	−5%	17.42	−2%	16.66	3%	14.30	16%
Distance R	Maximum Speed Depending on Altitude v(h) [m]/Wind Speed Relative Variability Parameter [%]											
	H = 40 m		H = 50 m		H = 60 m		H = 70 m		H = 75 m		H = 79 m	
[m]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]
60	7.69	0%	7.72	0%	7.72	0%	7.61	2%	7.61	1%	7.79	−1%
55	7.98	1%	8.03	0%	8.02	0%	7.96	1%	7.94	1%	8.01	0%

Table 10. Cont.

50	8.48	0%	8.39	1%	8.42	1%	8.47	0%	8.42	1%	8.43	0%
45	9.03	1%	9.03	1%	8.94	2%	9.12	0%	9.03	1%	9.04	1%
40	9.80	0%	9.75	1%	9.66	2%	9.65	2%	9.72	1%	9.75	1%
35	10.67	0%	10.82	−1%	10.61	1%	10.48	2%	10.64	1%	10.62	1%
30	11.53	2%	11.88	−1%	11.61	1%	11.56	1%	11.70	0%	11.50	2%
25	12.97	1%	13.07	0%	13.03	0%	12.89	1%	13.01	0%	12.70	3%
20	14.33	4%	14.55	3%	14.87	1%	14.71	2%	14.83	1%	14.83	1%
15	15.45	10%	16.02	7%	16.83	2%	17.18	0%	17.41	−1%	17.53	−2%
10	14.08	18%	14.09	18%	15.95	7%	18.06	−5%	19.56	−14%	20.33	−19%

Table 11. Wind speed values on the analyzed vertical plane and relative measurement error compared to other heights. Case A8 (Reynolds Stress model, initial velocity 22 m/s), heights from H = 1 m to H = 79 m.

Distance R	Maximum Speed Depending on Altitude v(h) [m]/Wind Speed Relative Variability Parameter [%]											
	H = 1 m		H = 5 m		H = 10 m		H = 15 m		H = 20 m		H = 30 m	
[m]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]
60	7.79	0%	7.82	0%	7.83	−1%	7.79	0%	7.77	0%	7.80	0%
55	7.91	2%	8.09	0%	8.19	−2%	8.12	−1%	8.11	−1%	8.12	−1%
50	8.02	5%	8.53	−1%	8.64	−2%	8.54	−1%	8.57	−1%	8.59	−2%
45	8.91	2%	9.19	−1%	9.21	−1%	9.19	−1%	9.18	−1%	9.14	−1%
40	9.76	1%	9.92	−1%	9.99	−2%	9.99	−2%	9.99	−2%	9.93	−1%
35	10.63	1%	10.80	−1%	10.81	−1%	10.85	−1%	10.83	−1%	10.88	−1%
30	11.25	4%	11.95	−2%	11.91	−2%	11.89	−1%	11.78	0%	11.87	−1%
25	12.76	2%	13.18	−1%	13.35	−2%	13.33	−2%	13.19	−1%	13.08	0%
20	15.54	−4%	15.34	−2%	15.56	−4%	15.33	−2%	15.08	−1%	14.52	3%
15	18.58	−8%	18.37	−6%	18.11	−5%	17.91	−4%	17.38	−1%	16.14	7%
10	18.87	−10%	18.58	−8%	18.01	−5%	17.52	−2%	16.76	3%	14.28	17%

Distance R	Maximum Speed Depending on Altitude v(h) [m]/Wind Speed Relative Variability Parameter [%]											
	H = 40 m		H = 50 m		H = 60 m		H = 70 m		H = 75 m		H = 79 m	
[m]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]	[m/s]	[%]
60	7.74	1%	7.76	0%	7.76	0%	7.74	1%	7.74	1%	7.89	−1%
55	8.01	1%	8.05	0%	8.03	0%	8.01	1%	7.98	1%	8.02	0%
50	8.49	0%	8.40	1%	8.42	0%	8.48	0%	8.41	1%	8.39	1%
45	9.04	0%	9.04	0%	8.95	1%	9.14	−1%	9.01	1%	8.99	1%
40	9.82	0%	9.77	1%	9.67	2%	9.67	2%	9.74	1%	9.74	1%
35	10.69	0%	10.84	−1%	10.63	1%	10.51	2%	10.66	1%	10.62	1%
30	11.55	1%	11.91	−2%	11.64	1%	11.59	1%	11.75	0%	11.54	2%
25	13.00	1%	13.12	0%	13.08	0%	12.93	1%	13.09	0%	12.83	2%
20	14.37	4%	14.59	3%	14.95	0%	14.80	1%	14.89	1%	14.90	1%
15	15.51	10%	16.08	7%	16.97	2%	17.30	0%	17.52	−1%	17.52	−1%
10	14.16	18%	14.01	19%	15.80	8%	18.18	−6%	19.68	−14%	20.54	−19%

For each case, two parts of the tables were created to improve readability. The first part presents results from $H = 1$ m to $H = 30$ m, while the second part covers $H = 40$ m to $H = 79$ m. It is important to remember that these tables collectively form a complete dataset.

At the end of each discussed case, the tabulated values were also presented using graphs illustrating the relationship between velocity and the distance from the vortex center. On the X-axis, values range from 60 m to 0 m. This deliberate inversion ensures that the vortex center is at 0 m, positioned on the right side. This provides better comparability of the graphs with the analyzed plane view (Figure 6). Consequently, graphical representations of values were created using Figures 7–12.

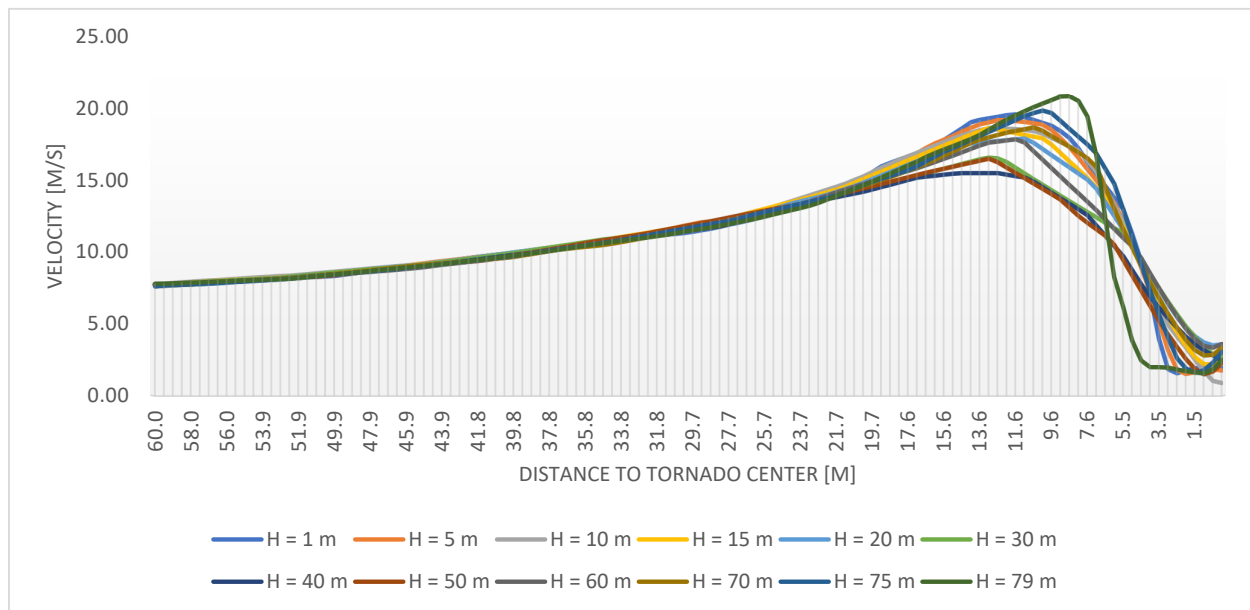


Figure 7. Graphical representation of Case A3 (Spalart–Allmaras model, inlet velocity = 22 m/s): velocity distribution as a function of distance from the vortex center at different heights.

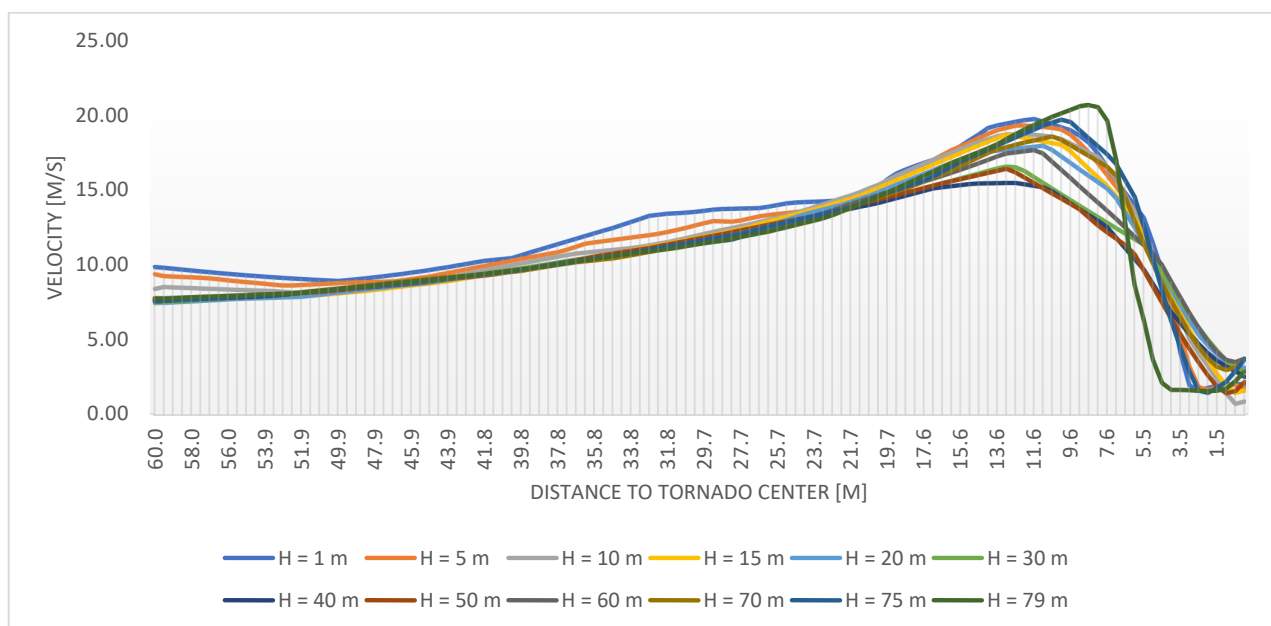


Figure 8. Graphical representation of case A4 (k-epsilon model, initial velocity 22 m/s), a graph of the dependence of velocity on distance from the vortex center for different heights.

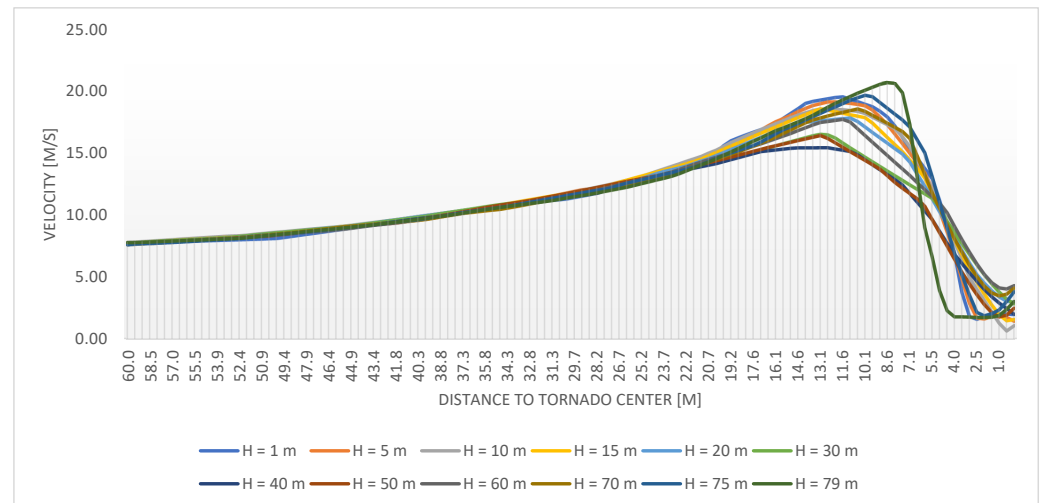


Figure 9. Graphical representation of case A5 (k-omega model, initial velocity 22 m/s), a plot of velocity versus distance from the vortex center for different heights.

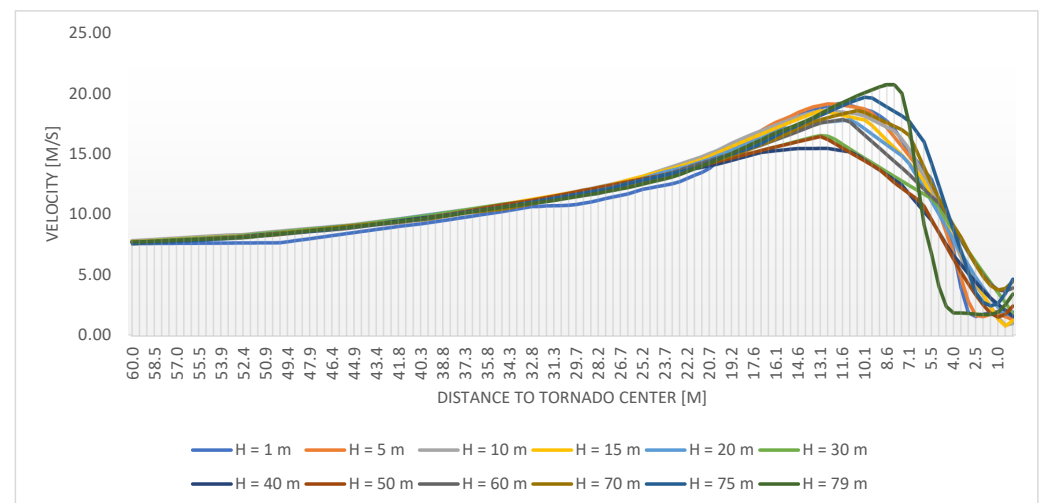


Figure 10. Graphical representation of case A6 (k-kl-omega model, initial velocity 22 m/s), a plot of velocity versus distance from the vortex center for different heights.

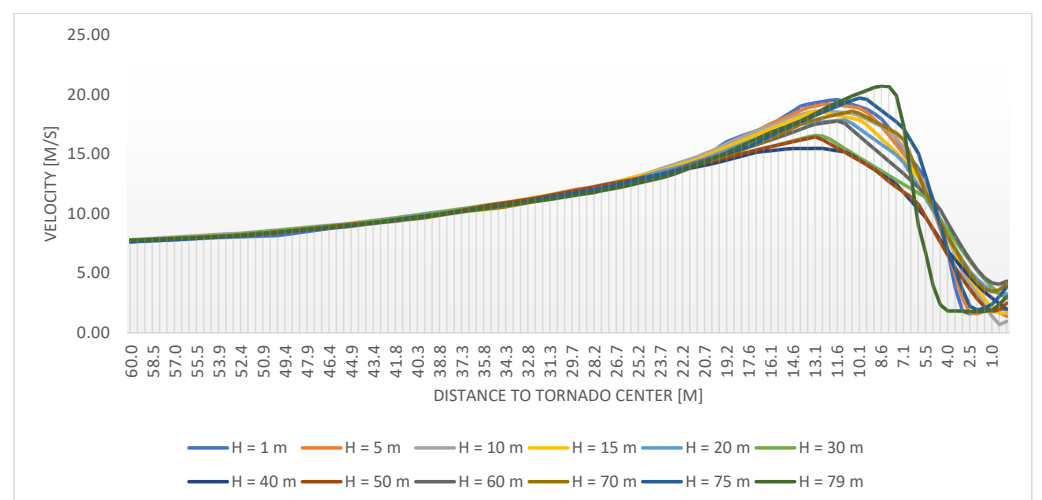


Figure 11. Graphical representation of case A7 (SST model, initial velocity 22 m/s), a plot of velocity versus distance from the vortex center for different heights.

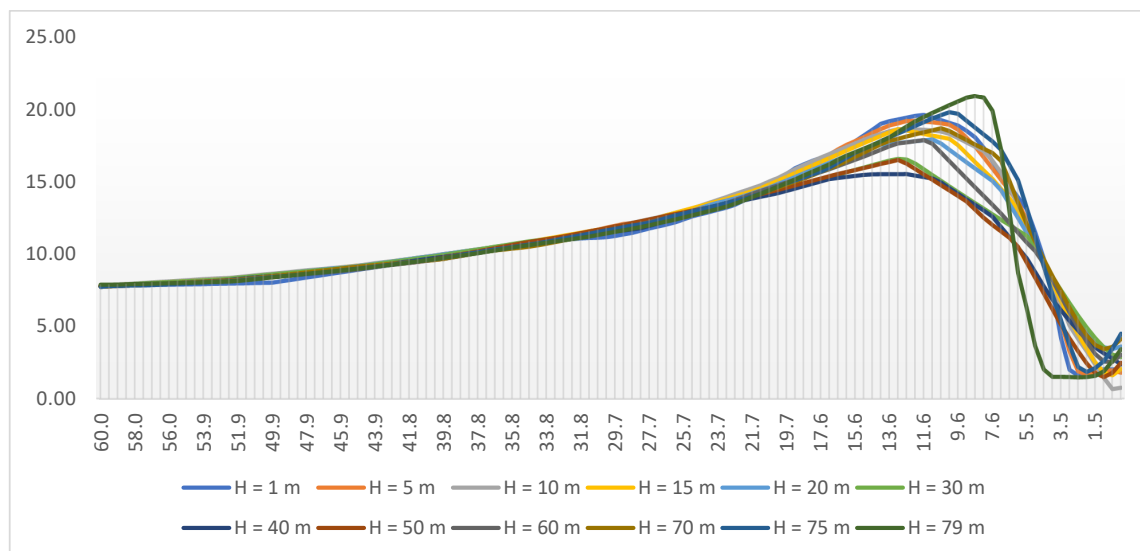


Figure 12. Graphical representation of case A8 (Reynolds Stress model, initial velocity 22 m/s), a plot of velocity versus distance from the vortex center for different heights.

5.3.1. Case A3: Spalart–Allmaras Model, Initial Velocity 22 m/s, Medium Size 60 m

Velocity distribution was obtained from the Spalart–Allmaras model for an inlet velocity of 22 m/s at different heights above the ground. The curves illustrate the variation in tangential velocity with distance from the tornado center. The general shape of the profiles follows the theoretical Rankine vortex pattern, with the maximum velocity occurring near the vortex core radius. Minor discrepancies between heights indicate vertical variability of the flow field and gradual weakening of the vortex intensity with elevation.

5.3.2. Case A4: k–Epsilon Model, Initial Velocity 22 m/s, Medium Size 60 m

The obtained velocity distributions show satisfactory agreement with the theoretical Rankine-type vortex profile. For most analyzed regions, the relative variability remained within approximately 5–8% up to 18% in close range to tornado center, indicating stable and numerically consistent flow behavior. Larger discrepancies were observed mainly in the immediate vicinity of the vortex core and near the upper boundary of the computational domain.

The velocity profiles obtained using the k– ϵ model generally follow the theoretical Rankine vortex distribution. Minor deviations occur mainly near the vortex core, while the overall agreement between different heights remains within acceptable engineering limits.

5.3.3. Case A5: k–Omega Model, Initial Velocity 22 m/s, Medium Size 60 m

The obtained velocity distributions show satisfactory agreement with the theoretical Rankine-type vortex profile. For most analyzed regions, the relative variability remained within approximately 1–4% up to 19% in close range to tornado center, indicating stable and numerically consistent flow behavior. Larger discrepancies were observed mainly in the immediate vicinity of the vortex core and near the upper boundary of the computational domain.

The k– ω model reproduced the expected vortex structure with stable velocity distributions across different heights. The results indicate good consistency with the analytical Rankine-type profile and limited sensitivity to vertical position.

5.3.4. Case A6: k–kl–Omega Model, Initial Velocity 22 m/s, Medium Size 60 m

The k–kl– ω model demonstrated slightly greater variability near the vortex center compared to other RANS approaches. Nevertheless, the overall velocity distribution remained consistent with the expected Rankine-type vortex behavior.

5.3.5. Case A7: SST Model, Initial Velocity 22 m/s, Medium Size 60 m

The SST model provided one of the closest agreements with the analytical vortex profile. The velocity distributions remained stable across different heights, with deviations generally limited to regions near the vortex core.

5.3.6. Case A8: Reynolds Stress Model, Initial Velocity 22 m/s, Medium Size 60 m

The Reynolds Stress Model reproduced the general vortex structure correctly; however, increased local fluctuations were observed near the vortex center and at higher elevations due to turbulence anisotropy effects.

Tables 6–11 summarize six RANS cases (A3–A8) evaluated at different heights ($H = 1\text{--}79\text{ m}$). For the majority of the analyzed domain, the obtained velocity distributions remained consistent with the theoretical Rankine-type profile, with relative deviations typically ranging between 4% and 8%. Larger local discrepancies—reaching up to approximately 19%—were observed primarily near the vortex core ($R \leq 10\text{ m}$) and close to the upper boundary of the computational domain ($H \geq 75\text{ m}$), where geometric effects associated with the funnel configuration influenced the flow structure. Regardless of the selected model, the largest disturbances occurred within 20 m from the vortex center. Beyond 20 m, significant deviations were observed only in model A4, with slight deviations in model A6, while the remaining models showed full agreement at this distance. Among different heights, the most noticeable deviations occurred at $H = 79\text{ m}$.

5.4. Comparison of Results

The following section compares the results of models A3, A4, A5, A6, A7, and A8 for selected heights $H = 1\text{ m}$, $H = 30\text{ m}$, $H = 79\text{ m}$, at a constant wind velocity of 22 m/s. Consequently, graphical representations in Figures 13–15 were created to compare models A3–A8 with one another.

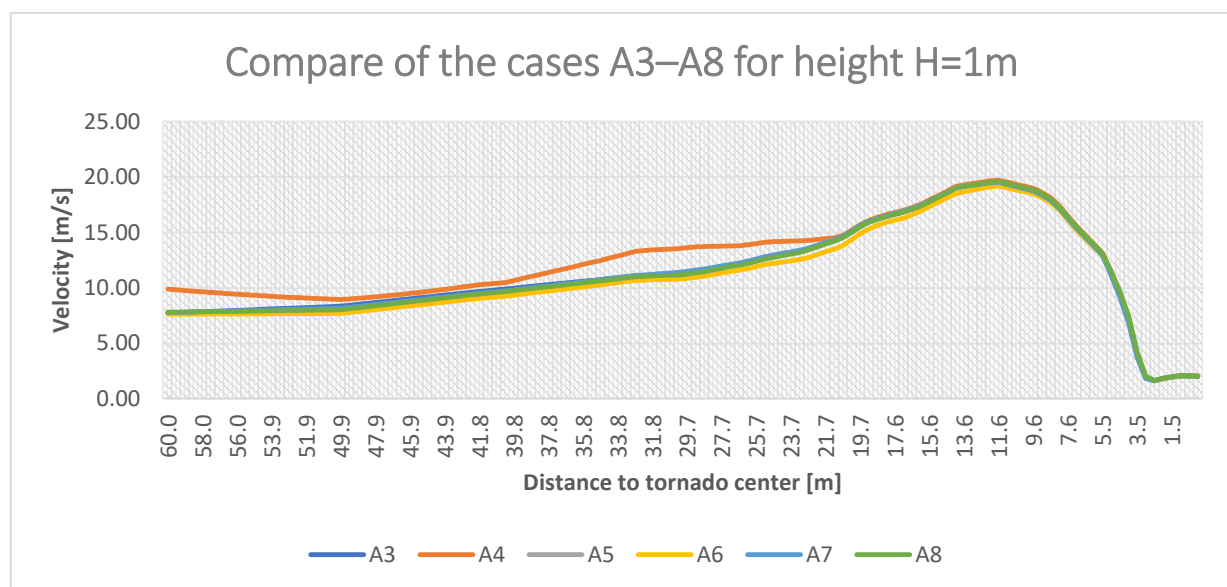


Figure 13. Graphical comparison of cases A3–A8, graph of the dependence of velocity on the distance from the vortex center at a constant height $H = 1\text{ m}$.

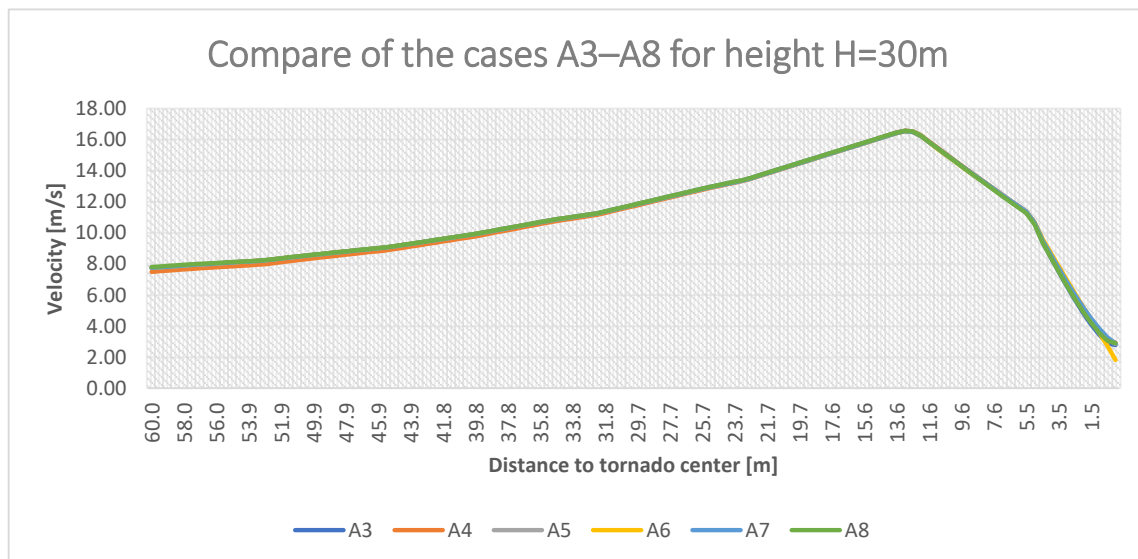


Figure 14. Graphical comparison of cases A3–A8, graph of the dependence of velocity on the distance from the vortex center at a constant height $H = 30$ m.

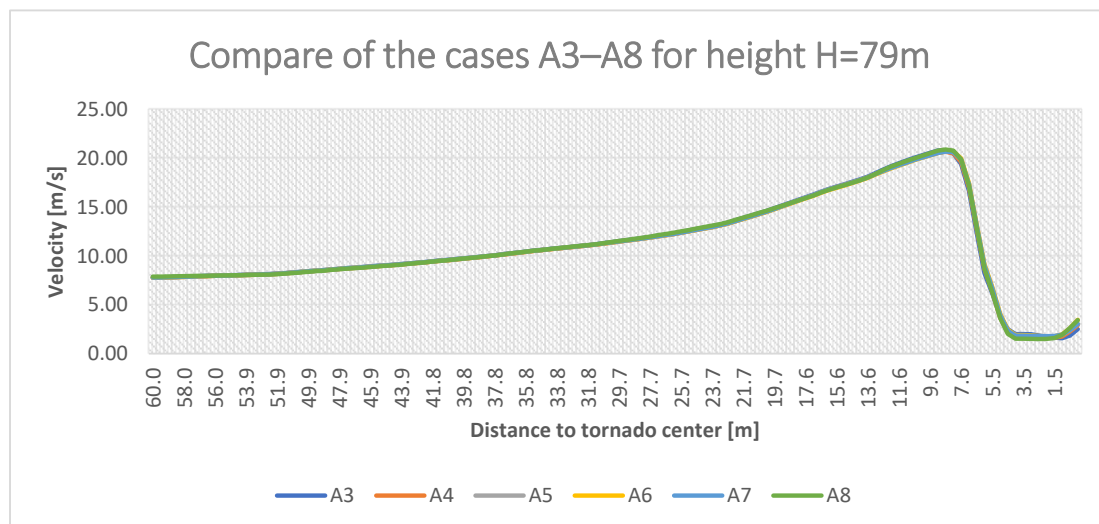


Figure 15. Graphical comparison of cases A3–A8, graph of the dependence of velocity on the distance from the vortex center at a constant height $H = 79$ m.

At $H = 1$ m, the differences between the turbulence models are most visible due to near-wall effects and large velocity gradients around 30 m and at the end of the domain.

At intermediate height ($H = 30$ m), the velocity distributions become more uniform, and the agreement between individual RANS models improves significantly.

At higher elevations ($H = 79$ m), the analyzed models maintain similar overall vortex structures, although local deviations become more noticeable near the vortex center.

Figures 13–15 compare RANS velocity profiles for models A3–A8, confirming that the Realizable $k-\varepsilon$ and SST $k-\omega$ turbulence models yielded the closest agreement with the analytical Rankine-type profile.

The most significant differences and deviations from the mean are clearly visible in the lower part of the medium, near the ground, at $H = 1$ m. Less distinct but still noticeable differences appear at the vortex center. Regardless of the reference height, different models provide slightly varying velocity values. Among the analyzed models, A4 and A6 exhibited the largest deviations from the mean velocity distributions, particularly in regions close to the vortex center.

The comparison indicates that the Realizable $k-\epsilon$ and SST $k-\omega$ models provide the most consistent representation of the vortex structure among the tested RANS approaches.

5.5. Wind Profile

The relationship between wind velocity and height above ground level can be represented using a wind profile. In civil engineering, wind profiles are essential for analyzing the variation in velocity with height.

Wind profiles compare observed CFD velocities with Rankine vortex theory, plotting height versus speed to reveal frictional and vortex effects. In a classical approach, a key assumption in its construction is that wind speed near the surface is lower due to friction, and it gradually increases with height, reaching a limiting value in the upper troposphere. In simplified engineering terms, such a profile is typically represented as an increasing curve with height.

Based on the obtained results, graphs were created, presented in Figures 16 and 17, comparing the dependence of speed on height in cases A3 (Spalart–Allmaras) and A4 (k–epsilon model).

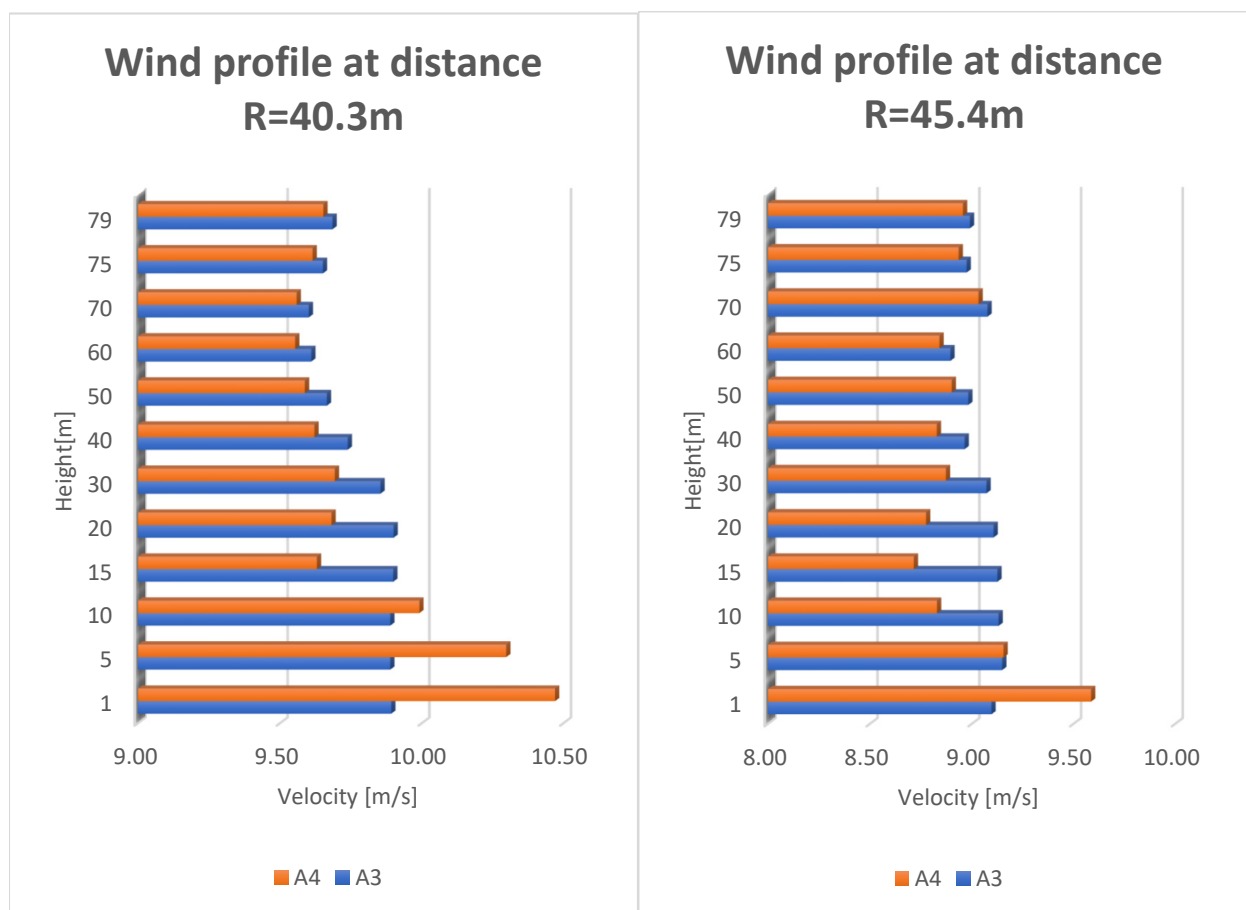


Figure 16. Classical axis wind profile. Comparison of cases A3 (Spalart–Allmaras) and A4 (k–epsilon model) at a distance of 40.3 m and 45.4 m from the vortex center.

Overlay plots of the Rankine profile confirm that the RANS-based CFD models successfully reproduce the expected height–velocity trends and the general characteristics of tornado-like vortex structures.

The profiles presented in Figure 16 indicate satisfactory agreement between CFD predictions and the expected vortex-induced wind behavior, particularly at lower and intermediate elevations.

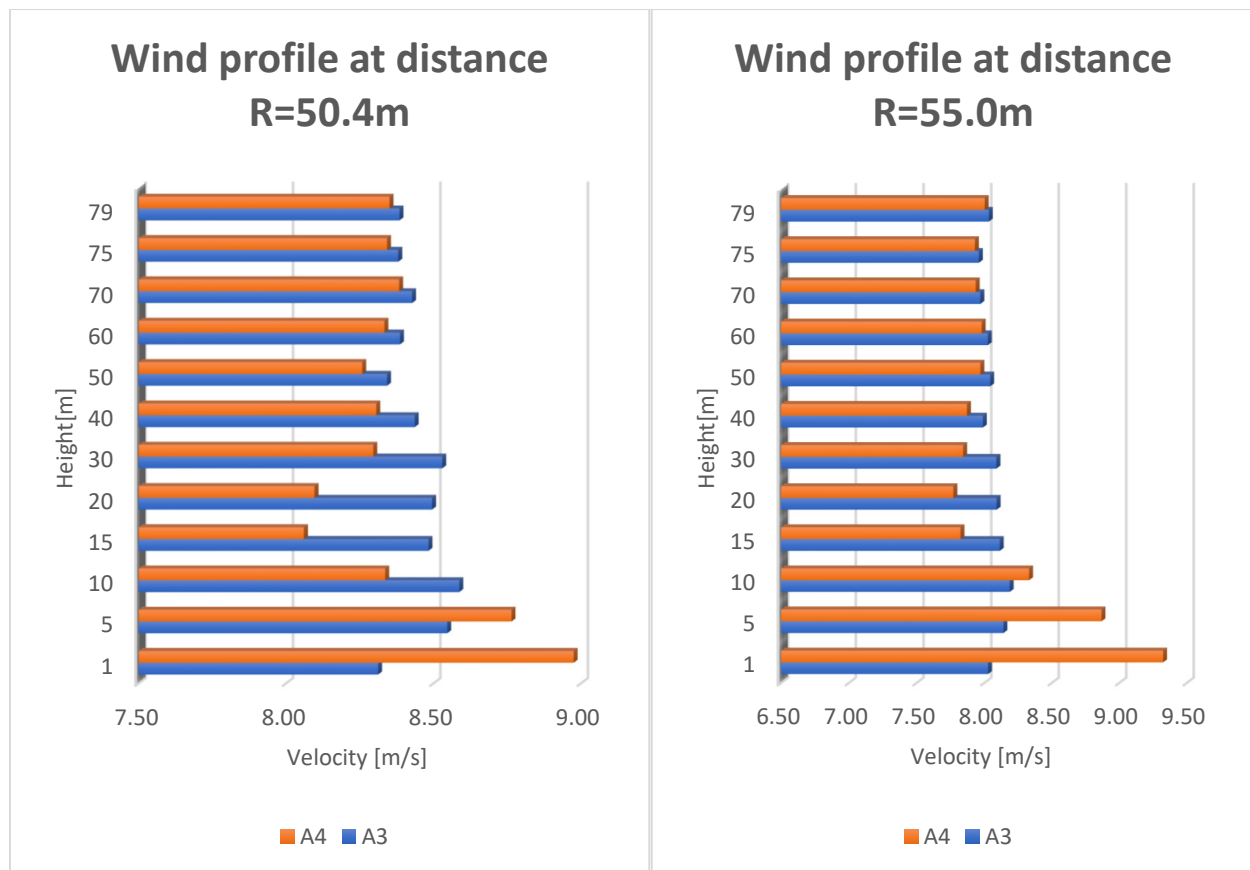


Figure 17. Classical axis wind profile. Comparison of cases A3 (Spalart–Allmaras) and A4 (k–epsilon model) at a distance of 50.4 m and 55.0 m from the vortex center.

Figure 17 confirms the persistence of the characteristic increase–drop–recovery trend in velocity distributions at larger radial distances from the vortex center.

Figures 16 and 17 further illustrate the height–velocity relationships, where CFD results replicated the increase–drop–recovery trend characteristic of theoretical tornado structures.

The A4 model shows increased velocities near the ground and a drop in speed in the middle of the altitude. In the case of the A3 (Spalart–Allmaras) model, the speed increases as expected at lower altitudes, but after half the altitude a gradual reduction in velocity is observed.

6. Conclusions

The adopted methodology combined theoretical vortex representations, CFD simulations, and parametric verification procedures to ensure reliable and reproducible results. The primary objective of the study was to develop a simplified yet reliable CFD model capable of replicating the fundamental characteristics of strong rotational wind phenomena while maintaining computational efficiency. The numerical model, developed in ANSYS Fluent 2025 R1, successfully reproduced the characteristic velocity distributions of tornado-like flow structures. The adopted computational domain dimensions ($R = 60$ m, $H = 80$ m) were selected through parametric analyses, which demonstrated that further domain enlargement produced only minor changes in velocity distributions while significantly increasing computational cost. Similarly, the triangular finite element mesh with a characteristic size of 9 m—supplemented by boundary-layer inflation near the ground—provided stable and numerically consistent results, as confirmed through grid sensitivity analysis. Verification of the velocity distribution using the Rankine vortex profile confirmed the correctness of the adopted assumptions. The analyzed RANS models reproduced the

fundamental rotational characteristics of the vortex, with deviations generally remaining within acceptable engineering limits. However, the greatest discrepancies were observed at distances below $r < 10$ m from the vortex center, while beyond this range, the simulated and theoretical profiles showed strong agreement. The simplicity of the proposed model provided several notable advantages:

- reduced computational resource requirements,
- clear and directly comparable velocity distributions,
- limited boundary-induced disturbances near the vortex core region.

Nevertheless, the research also identified several limitations that define directions for future improvement: the analyzed model represented a stationary tornado, whereas real tornadoes are characterized by complex translational motion and evolving vortex structures that may significantly influence local flow behavior.

The results, obtained through parametric RANS simulations for domain sizes ranging from $R = 60\text{--}480$ m and $H = 80\text{--}200$ m, were compared with an analytical Rankine-type vortex profile. The analysis focused on velocity distributions within a simplified computational domain. These findings confirm that the proposed modeling approach enables a reliable reproduction of tornado-like flow structures while optimizing computational resources. The adopted computational domain of $60\text{ m} \times 80\text{ m}$ represents an efficient solution for further numerical investigations of vortex-dominated flows.

The presented CFD framework represents an idealized and stationary tornado-like flow configuration intended primarily for controlled engineering analyses. Although simplified, the model successfully reproduces the principal characteristics of rotational vortex flows and provides stable and repeatable numerical results across different turbulence models.

The conducted analyses demonstrate that both the selection of the turbulence model and the computational domain configuration have a significant influence on the predicted velocity distributions. In particular, the Realizable $k\text{--}\epsilon$ and SST $k\text{--}\omega$ models showed the most stable and consistent behavior in reproducing the analytical vortex profile, indicating their strong suitability for engineering-oriented simulations of tornado-like flow phenomena.

Closing Remarks

This study contributes to the continued development of CFD-based methodologies in contemporary wind engineering applications. By combining theoretical vortex models with RANS-based simulations, it provides a solid foundation for analyzing tornado-like flow structures in an engineering context. The presented methodology can serve as a reference for future numerical and experimental investigations, supporting the development of safer and more resilient building designs under extreme wind conditions.

Future work will include the implementation of advanced Reynolds Stress Models (RSM) and hybrid RANS–LES approaches, together with the consideration of moving vortex dynamics and more complex building geometries, to establish a broader and more physically representative framework for wind-related flow analysis.

Author Contributions: Conceptualization, M.D. and T.L.; Methodology, M.D.; Software, T.L.; Validation, M.D.; Formal analysis, M.D.; Investigation, T.L. and M.D.; Data curation, M.D., Resources, T.L.; Writing—original draft, T.L.; Writing—review & editing, M.D.; Visualization, T.L.; Supervision, M.D.; Project administration, M.D. All authors have read and agreed to the published version of the manuscript.

Funding: This research received no external funding.

Data Availability Statement: The data presented in this study are available on request from the corresponding author.

Conflicts of Interest: The authors declare no conflict of interest.

References

1. Lamparski, T.; Dutkiewicz, M. Analysis of Pressure Distribution on a Single-Family Building Caused by Standard and Heavy Winds Based on a Numerical Approach. *Appl. Sci.* **2024**, *14*, 1976. [\[CrossRef\]](#)
2. Lamparski, T.; Dutkiewicz, M. The Action of Tornadoes on the Structural Elements of a Wooden Low-Rise Building Roof and Surrounding Objects—Review and Case Study. *Appl. Sci.* **2023**, *13*, 4661. [\[CrossRef\]](#)
3. Lamparski, T.; Dutkiewicz, M. Integrated Aerodynamic–Structural Validation Framework for Wind-Induced Load Assessment. *Appl. Sci.* **2026**, *16*, 2986. [\[CrossRef\]](#)
4. Wurman, J.; Kosiba, K. The Three-Dimensional Structure and Evolution of a Tornado Boundary Layer. *Weather Forecast.* **2013**, *28*, 1552–1561. [\[CrossRef\]](#)
5. Bluestein, H.B.; French, M.M.; Snyder, J.C.; Houser, J.B. Doppler Radar Observations of Anticyclonic Tornadoes in Cyclonically Rotating, Right-Moving Supercells. *Mon. Weather Rev.* **2016**, *144*, 1591–1616. [\[CrossRef\]](#)
6. Wang, S.; Reed, D.A.; Lyman, G.; Estephan, J.; Irwin, P.; Chowdhury, A. Examination of wind speed based on field measurements on a low-rise building. *Front. Built Environ.* **2025**, *11*, 1498984. [\[CrossRef\]](#)
7. Feng, R.; Liu, F.; Cai, Q.; Yan, G.; Leng, J. Field measurements of wind pressure on an open roof during Typhoons HaiKui and SuLi. *Wind Struct.* **2018**, *26*, 11–24.
8. Honerkamp, R.; Yan, G.; Snyder, J.C. A review of the characteristics of tornadic wind fields through observations and simulations. *J. Wind Eng. Ind. Aerodyn.* **2020**, *202*, 104195. [\[CrossRef\]](#)
9. Haan, F.L.; Sarkar, P.P.; Gallus, W.A. Design, construction and performance of a large tornado simulator for wind engineering applications. *Eng. Struct.* **2008**, *30*, 1146–1159. [\[CrossRef\]](#)
10. Haan, F.L.; Balaramudu, V.K.; Sarkar, P.P. Tornado-Induced Wind Loads on a Low-Rise Building. *J. Struct. Eng.* **2009**, *136*, 106–116. [\[CrossRef\]](#)
11. Chen, Q.; Tang, Z.; Wu, X.; Zuo, D. Laboratory study of tornado-like loading on a low-rise building model. *J. Wind Eng. Ind. Aerodyn.* **2023**, *238*, 105443. [\[CrossRef\]](#)
12. Sabareesh, G.R.; Matsui, M.; Tamura, Y. Dependence of surface pressures on a cubic building in tornado like flow on building location and ground roughness. *J. Wind Eng. Ind. Aerodyn.* **2012**, *103*, 50–59. [\[CrossRef\]](#)
13. Zhong, M.; Lin, M.; Zhou, Z.; Li, Z.; Lu, J.; Xu, F. Wind tunnel test on low-rise buildings influenced by hillside height in typical mountain terrain. *Adv. Aerodyn.* **2024**, *6*, 10. [\[CrossRef\]](#)
14. Ishihara, T.; Oh, S.; Tokuyama, Y. Numerical study on flow fields of tornado-like vortices using the LES turbulence model. *J. Wind Eng. Ind. Aerodyn.* **2011**, *99*, 239–248. [\[CrossRef\]](#)
15. Yuan, F.; Yan, G. Numerical simulation of laboratory tornado simulator that can produce translating tornado-like wind flow. *J. Wind Eng. Ind. Aerodyn.* **2019**, *190*, 200–217. [\[CrossRef\]](#)
16. Cao, S.; Wang, M.; Cao, J. Numerical study of wind pressure on low-rise buildings induced by tornado-like flows. *J. Wind Eng. Ind. Aerodyn.* **2018**, *183*, 214–222. [\[CrossRef\]](#)
17. Verma, S.; Selvam, R.P. CFD model validation with experimental tornado wind field & comparison of wind field in different tornado chambers. *Wind Struct.* **2021**, *33*, 367–381.
18. Holton, J.R.; Hakim, G.J. *An Introduction to Dynamic Meteorology*; Elsevier: Amsterdam, The Netherlands, 2016.
19. Lorenz, E.N. *The Nature and Theory of the General Circulation of the Atmosphere*; WMO: Geneva, Switzerland, 1967.
20. Nakayama, Y. *Introduction to Fluid Mechanics*; Butterworth-Heinemann: Oxford, UK, 2018.
21. Kundu, P.K.; Cohen, I.M.; Dowling, D.R.; Capecelatro, J. *Fluid Mechanics*; Elsevier: Amsterdam, The Netherlands, 2024.
22. Tamura, Y. Wind-Induced Damage to Buildings and Disaster Risk Reduction. In Proceedings of the APCWE-VII, Taipei, Taiwan, 8–12 November 2009.
23. Dutkiewicz, M.; Lamparski, T. Ultimate and serviceability limit States of a wooden roof under the wind speed that may cause potential damage to the structure. *AIP Conf. Proc.* **2023**, *2928*, 150038. [\[CrossRef\]](#)
24. Dutkiewicz, M.; Hajyalikhani, P.; Lamparski, T.; Whitman, L.; Covarrubias, J. Structural bracing of wooden roofs under the extreme wind. *IOP Conf. Ser. Mater. Sci. Eng.* **2021**, *1203*, 022024. [\[CrossRef\]](#)
25. Huang, Z.; Fan, X.; Cai, L.; Shi, S.Q. Tornado hazard for structural engineering. *Nat. Hazards* **2016**, *83*, 1821–1842. [\[CrossRef\]](#)
26. Holmes, J.D. *Wind Loading of Structures*, 3rd ed.; CRC Press: Boca Raton, FL, USA, 2015.
27. Simiu, E.; Scanlan, R.H. *Wind Effects on Structures: Fundamentals and Applications to Design*; Wiley: New York, NY, USA, 1996.
28. Simiu, E.; Yeo, D.H. *Wind Effects on Structures: Modern Structural Design for Wind*; John Wiley & Sons: Hoboken, NJ, USA, 2019.
29. Kareem, A.; Bernardini, E.; Spence, S.M.J. Control of the Wind Induced Response of Structures. *Adv. Struct. Wind Eng.* **2013**, *377*–410. [\[CrossRef\]](#)
30. Franke, J.; Hellsten, A.; Schlünzen, K.H.; Carissimo, B. Best Practice Guideline for the CFD Simulation of Flows in the Urban Environment. Doctoral Dissertation, COST European Cooperation in Science and Technology, Brussels, Belgium, 2007.

31. Tominaga, Y.; Mochida, A.; Yoshie, R.; Kataoka, H.; Nozu, T.; Yoshikawa, M.; Shirasawa, T. AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings. *J. Wind Eng. Ind. Aerodyn.* **2008**, *96*, 1749–1761. [[CrossRef](#)]
32. Casey, M.; Wintergerste, T. *Best Practice Guidelines for Industrial CFD*; ERCOFTAC: Brussels, Belgium, 2000.
33. Blocken, B. 50 years of Computational Wind Engineering: Past, present and future. *J. Wind Eng. Ind. Aerodyn.* **2014**, *129*, 69–102. [[CrossRef](#)]
34. Schlichting, H.; Gersten, K. *Boundary-Layer Theory*; Springer: Berlin, Germany, 2016.
35. Pope, S.B. Turbulent Flows. *Meas. Sci. Technol.* **2000**, *12*, 2020–2021. [[CrossRef](#)]
36. Murakami, S. Overview of turbulence models applied in CWE–1997. *J. Wind Eng. Ind. Aerodyn.* **1998**, *74–76*, 1–24. [[CrossRef](#)]
37. Batchelor, G.K. *An Introduction to Fluid Dynamics*; Cambridge University Press: Cambridge, UK, 2000.
38. Chorin, A.J.; Marsden, J.E. *A Mathematical Introduction to Fluid Mechanics*; Springer: New York, NY, USA, 1990.
39. Mohotti, D.; Wijesooriya, K.; Dias-da-Costa, D. Comparison of Reynolds Averaging Navier-Stokes (RANS) turbulent models in predicting wind pressure on tall buildings. *J. Build. Eng.* **2019**, *21*, 1–17. [[CrossRef](#)]
40. Ansys Fluent 12.0 Theory Guide. Available online: https://www.afs.enea.it/project/neptunius/docs/fluent/html/th/main_pre.htm (accessed on 20 November 2023).
41. CFD EXPERTS. Ansys Fluent Theory Guide. Available online: <https://drive.google.com/drive/folders/1EBB8A7HSFYnVkFdx2hMcLPW4MoeVM5xu?usp=sharing> (accessed on 20 November 2023).
42. Xiong, M.; Chen, B.; Zhang, H.; Qian, Y. Study on Accuracy of CFD Simulations of Wind Environment around High-Rise Buildings: A Comparative Study of k- ϵ Turbulence Models Based on Polyhedral Meshes and Wind Tunnel Experiments. *Appl. Sci.* **2022**, *12*, 7105. [[CrossRef](#)]
43. Thordal, M.S.; Bennetsen, J.C.; Koss, H.H.H. Review for practical application of CFD for the determination of wind load on high-rise buildings. *J. Wind Eng. Ind. Aerodyn.* **2019**, *186*, 155–168. [[CrossRef](#)]
44. Richards, P.J.; Hoxey, R.P. Appropriate boundary conditions for computational wind engineering models using the k- ϵ turbulence model. *J. Wind Eng. Ind. Aerodyn.* **1993**, *46–47*, 145–153. [[CrossRef](#)]
45. Enhanced Fujita Scale. Available online: <https://www.spc.noaa.gov/efscale/> (accessed on 30 March 2026).
46. EN 1991: Eurocode 1: Actions on Structures (EN 1991) Parts 1-1 to 1-7. Available online: <https://www.phd.eng.br/wp-content/uploads/2015/12/en.1991.1.1.2002.pdf> (accessed on 30 November 2022).
47. Roache, P.J. *Verification and Validation in Computational Science and Engineering*; Hermosa: Albuquerque, NM, USA, 1998.
48. Oberkampf, W.L.; Trucano, T.G. Verification and validation benchmarks. *Nucl. Eng. Des.* **2008**, *238*, 716–743. [[CrossRef](#)]

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.