

Article

CFD Analysis of Aerodynamic Characteristics in a Square-Shaped Swarm Formation of Four Quadcopter UAVs

Ahmet Talat İnan and Berkay Çetin * 

Institute of Pure and Applied Sciences, Marmara University, Istanbul 34722, Türkiye; ainan@marmara.edu.tr

* Correspondence: brkycn@hotmail.com

Abstract: The aerodynamic behavior of a square-shaped formation of four quadcopter UAVs flying in a swarm is investigated in detail through three-dimensional computer simulations utilizing Computational Fluid Dynamics (CFD) methodology. The swarm configuration comprises four UAVs positioned with two in the upper row and two in the lower row along the same propeller axes. The flow profile generated by the UAV propellers rotating at 10,000 revolutions per minute is analyzed parametrically using the Multiple Reference Frame (MRF) technique. UAVs within the swarm are positioned at 75 cm from the motion centers of adjacent propellers. This distance, the effects of horizontally and vertically positioned UAVs on each other, and the collective behavior of the swarm are thoroughly examined. Pressure, velocity, and turbulent kinetic energy values are meticulously analyzed. This research represents a milestone in understanding the aerodynamic characteristics of UAV swarms and the optimization of swarm performance. The findings highlight effective factors in swarm flights and their consequences for UAVs. Additionally, the article describes the “near-UAV phenomenon”. Furthermore, the methodology developed for CFD simulations provides an approach to analyzing close flight scenarios and evaluating their performance in various swarm configurations. These achievements contribute to the future development of UAV technology.

Keywords: unmanned aerial vehicle (UAV); quadcopter; computational fluid dynamics (CFD); swarm; flock



Citation: İnan, A.T.; Çetin, B. CFD Analysis of Aerodynamic Characteristics in a Square-Shaped Swarm Formation of Four Quadcopter UAVs. *Appl. Sci.* **2024**, *14*, 6820. <https://doi.org/10.3390/app14156820>

Academic Editor: Francesca Scargiali

Received: 8 July 2024

Revised: 20 July 2024

Accepted: 23 July 2024

Published: 5 August 2024



Copyright: © 2024 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) license (<https://creativecommons.org/licenses/by/4.0/>).

1. Introduction

The widespread use of unmanned aerial vehicles (UAVs) can be attributed to ongoing technological advancements, meeting evolving needs and demands. Unmanned aerial systems (UASs) have broad utility, particularly in military contexts but also in civilian, industrial, and academic sectors [1–4]. These vehicles are preferred due to their advantages such as prolonged flight in the air, operation without crew members, vertical takeoff and landing capabilities, rapid maneuverability, and the ability to accomplish hazardous missions [2,5]. Unmanned aerial vehicles are classified into fixed-wing and rotary-wing aircrafts based on their wing functionalities [6]. This classification includes various types such as helicopters, multi-copters, and tiltrotors [2,7–9]. There has also been a significant increase in the amount of research on UAVs in recent years [10,11]. Many studies exist focusing on improving the flight performance of fixed wing unmanned aerial vehicles. In these studies, investigations have been conducted on wings and wingtips, and research has been carried out on UAV components affecting flight.

Computational Fluid Dynamics (CFD) analyses were performed to determine the dynamic response of an unmanned aerial vehicle [12]. These analyses were made with dynamic mesh using active, passive, and deformable mesh regions, and the oscillatory movements that emerged in the analyses were examined. Error rates were determined by comparing simulation results with experimental data obtained from test flights. In another research study [11] fixed-wing UAV analyses were made using a one-way Fluid–Solid interaction. It was demonstrated that successful and reliable results were achieved using artificial neural networks.

In the research of Kapsalis, Panagiotou, and Yakinthos, CFD analyses of body-wing UAVs were optimized using Taguchi methods and analyzed. It has been shown that performance can be improved with design variables [13]. Additionally, in another research study investigated by Panagiotou, Kaparos, and Yakinthos, the winglet structures with different design criteria of fixed-wing UAVs were analyzed using CFD. It was revealed that an approximate 10% increase in flight duration was achieved due to increased aerodynamic performance [5].

Fixed-wing models are preferred for their long flight range, while multi-rotor UAVs are more advantageous for short range flights. Multi-rotor vehicles are favored due to their ease of operation, flight stability, and advanced maneuverability capabilities [2]. Due to these advantageous features, multi-rotor UAVs find applications in tasks such as search and rescue, area surveillance, and smart agriculture practices including pesticide spraying, seeding, and spraying [10,14,15]. It is observed that there is an increasing interest in the use of multi-rotor platforms for spraying and pesticide applications.

Research has also been conducted on components that affect the performance of UAVs, such as the landing gear. Experimental data obtained from wind tunnel tests of the landing gear were compared with CFD analyses by Götten and Havermann, leading to design suggestions for reducing frictional forces [16]. Moreover, different types of wings have been examined to determine the performance of the airfoil [17].

Fluid analyses were carried out using the $k-\epsilon$ turbulence model in a study by Wen et al., which investigated a tilt-rotor UAV model with three rotors. The study examined the performance of vehicles operating at 2100 rpm with various rotor angles, thus verifying the applicability of wing movements [7]. Additionally, aerodynamic characteristics were compared by Wan, Sun et al. through wind tunnel experiments and CFD simulations, and the results were shown to be compatible [8]. In addition to these studies, analyses have also been performed on three-rotor VTOL (Vertical Take-Off and Landing) systems [9].

In the literature, UAVs with configurations of four, six, and eight rotors have been an extensive subject of research. In a study conducted by Yang et al. [18], three-dimensional CFD analyses were performed on a 6-rotor UAV used for agricultural purposes using the $k-\epsilon$ turbulence model. The analyses examined the downwash created by the vehicle at heights of 1 to 2 m above ground level. Within these analyses, the rotational speed of the propellers was taken as 288 rad/s (2750 rpm), and according to the analysis results, the maximum velocity generated by the ground effect was determined to be 10 m/s. Chen et al. performed unsteady CFD analyses of a 6-rotor UAV equipped with a spray system. The UAV was positioned at a height of 2 m and operated at a rotational speed of 3600 rpm. In these analyses, the distribution of downwash effects at different distances was determined and comparisons between experimental data and CFD results were conducted [19].

CFD analyses of an eight-rotor UAV configuration were performed by Ni, Wang et al. The UAV model was simplified and was positioned at a height of 3 m above ground level. Additionally, rotation speed was set at 3500–4000 rpm, and crosswind and ground effects were used. As a result, it was shown that the spray systems were successful [14]. The research carried out by Yang, Tang, et al. performed CFD analyses to investigate the downwash effect at different levels using a six-rotor UAV. The analyses were based on the $k-\epsilon$ turbulence model, and the rotational speed of the small-sized UAV was selected as 1500 rpm. To investigate the downwash, the UAV was positioned at different heights (0.2 m; 0.5 m; 2 m, and 5 m). The results from the small-scale UAV suggest that it could potentially replace the full-scale UAV [10].

Research on aeroacoustics related to UAVs has also been conducted. In a study by Dbouk and Drikakis, the aeroacoustics characteristics of a swarm consisting of six multi-copters were examined. The swarm was positioned horizontally in V-shaped and rectangular-shaped (U-shaped) configurations. Three-dimensional CFD analyses were performed, and the flight speed was set to 9 m/s. The aeroacoustics characteristics of the swarm were determined [20].

Paz, Suarez et al. carried out CFD analyses on a four-rotor UAV model, which is also the subject of this study, using MRF (Multiple Reference Frame) and Sliding Mesh methods. In their research, the ground effect was also examined by taking the rotation speed of propellers as 1000 rad/s and the analyses based on the initial positions of the propellers [1]. The evaluations indicated that the MRF method was a suitable option for calculating the relative positions of the propellers and the MRF method could be preferred over the Sliding Mesh method for ground effect analyses. In another study by Paz, Suarez et al. [2], the aerodynamic effects occurring from passing over an obstacle using a quad-rotor UAV model were investigated. The study successfully determined the ground effect that may occur as a UAV moves at speeds of 1.5, 7.5, and 10 m/s when passing over an obstacle.

The focus of this paper is CFD analyses of four multi-rotor UAVs arranged in a rectangular formation, with two UAVs positioned above and two below at a specified proximity distance. The objective was to calculate the flight configurations of multiple UAVs in a vertical formation using CFD analyses with the $k-\epsilon$ turbulence model and the MRF method and to investigate their flight performances. Upon examining previous research in the literature, this research represents a milestone in the aerodynamic behavior of swarm UAVs. The swarm formation that four rotary-wing unmanned aerial vehicles can create during flight was investigated in detail, their behaviors during flight were modeled and interpreted using Computational Fluid Dynamics. Additionally, the phenomenon of close UAV interaction was defined.

2. Materials and Methods

2.1. Mathematical Model

The mathematical algorithms that form the theoretical background of Computational Fluid Dynamics are typically based on the finite volume method, with ANSYS Fluent relying on this approach. The background, continuity, and momentum equations are present. Equation (1) provides the continuity equation, and Equation (2) presents the momentum equations. In the continuity equations, S_m represents the source term, ρ denotes density, t stands for time, and $\vec{v}$ represents the velocity vector.

In the momentum equation, p denotes pressure, $\vec{\tau}$ represents the stress tensor, $\vec{g}$ is the gravitational acceleration, and $\vec{F}$ represents the body forces.

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = S_m \quad (1)$$

$$\frac{\partial}{\partial t} (\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \vec{v}) = \nabla p + \nabla \cdot (\vec{\tau}) + \rho \vec{g} + \vec{F} \quad (2)$$

The moving reference frame (MRF) method was employed in the analyses, which can be expressed as the defining movement relative to a reference point. This methodology comprises two types of functions: absolute and relative. Absolute function definitions were employed for the analyses.

$$\vec{\omega} = \omega \hat{a} \quad (3)$$

$$\vec{v}_r = \vec{v} - \vec{u}_r \quad (4)$$

$$\vec{u}_r = \vec{v}_t + \vec{\omega} \times \vec{r} \quad (5)$$

Here, $\vec{\omega}$ represents angular velocity (3), $\vec{v}_r$ denotes relative velocity (4), $\vec{v}$ refers to absolute velocity, $\vec{u}_r$ stands for the velocity of the moving frame relative to the inertial reference frame (5), and $\vec{v}_t$ signifies the translational frame velocity. Using these expressions, continuity and momentum equations can be rewritten as Equations (6) and (7).

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}_r) = 0 \quad (6)$$

$$\frac{\partial}{\partial t} (\rho \vec{v}) + \nabla \cdot (\rho \vec{v}_r \vec{v}) + \rho [\vec{\omega} \times (\vec{v} - \vec{v}_t)] = -\nabla p + \nabla \cdot \vec{\tau} + \vec{F} \quad (7)$$

Equations for the Realizable k- ϵ turbulence model are given in (8) and (9).

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \epsilon - Y_M + S_k \quad (8)$$

$$\frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial x_j}(\rho \epsilon u_j) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] + \rho C_1 S \epsilon - \rho C_2 \frac{\epsilon^2}{k + \sqrt{\nu \epsilon}} + C_{1\epsilon} \frac{\epsilon}{k} C_{3\epsilon} G_b + S_\epsilon \quad (9)$$

Equation (8) describes the turbulence kinetic energy (TKE) generation due to velocity gradients G_k , buoyancy G_b , and TKE generation from fluctuating expansion Y_M . Additionally, in Equation (9), C_2 and $C_{1\epsilon}$ are constants. σ_k and σ_ϵ represent the turbulent Prandtl numbers for k and ϵ respectively [21].

2.2. Geometric Model

Quad-rotors are among the most common types of rotary-wing unmanned aerial vehicles. Therefore, the geometry of this unmanned aerial vehicle was preferred for Computational Fluid Dynamics analysis. Simplifications were applied to the solid geometries of the UAV fuselage and the four propellers and rotating flow cylinders were modeled around each propeller. The UAV body, propellers, and rotating flow cylinders were integrated into the computational domain. The flow geometry of the UAV along the Y-X and X-Z axes is illustrated in Figure 1 of this research.

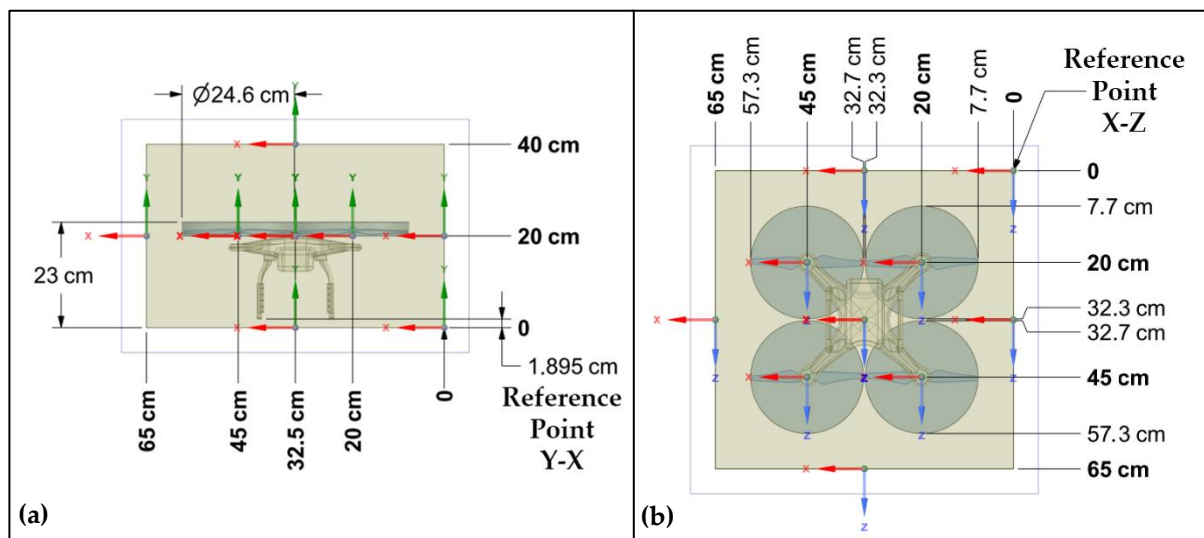


Figure 1. Quad-rotor unmanned aerial vehicle geometry: (a) Y-X view; (b) X-Z view.

A specific reference point was defined in the fluid model of the UAV. The relative positions of the UAV with respect to this reference point are illustrated in Figure 1. In the CFD geometry used for multiple flight analyses, the positionings were defined relative to this specified reference point. According to this reference point, the distances along the Y-X and X-Z axes are shown as local coordinates.

2.3. Numerical Methodology and Boundary Conditions

The objective of this research was to identify a swarm configuration consisting of four unmanned aerial vehicles and investigate their spatial positioning within the swarm. The spatial positions within the swarm were referenced with respect to a designated reference point. Four UAV models, each equipped with four rotating fluid domains, were utilized. A total of 17 fluid domains, comprising 16 rotating fluid domains and 1 external fluid domain,

were defined in the CFD geometry. Additionally, parameters for each of the 16 UAV propellers, including their local origin and orientation vectors, were individually defined.

The External Fluid Volume in the CFD geometry, illustrated in Figure 2a, had a height of 10 m. The width dimensions of the external flow volume were defined to be located 100 cm away from the central axis of the propeller adjacent to the surface. The spatial arrangement of UAVs is shown in Figure 2b, whereas Figure 2c illustrates the positioning of UAVs with dimensions. The distance between the propeller axes of adjacent UAVs was 75 cm.

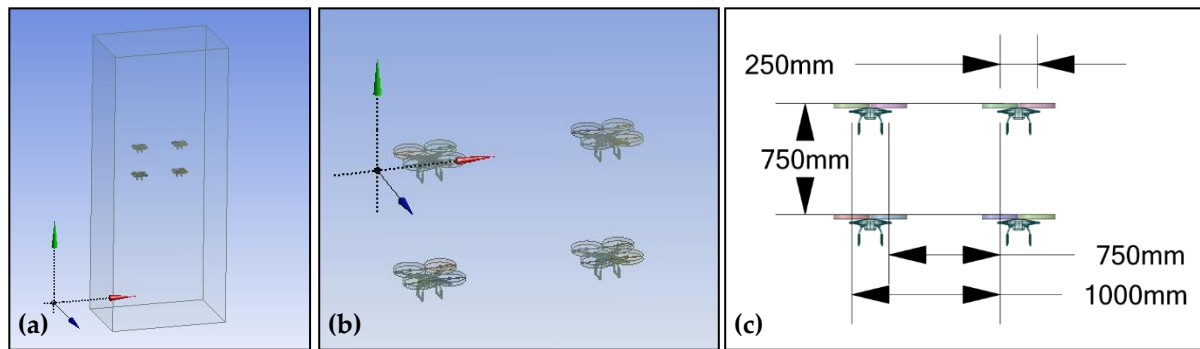


Figure 2. CFD geometries: (a) external fluid volume with quadrotors; (b) swarm formation; (c) distances of UAVs.

Figure 3 illustrates the mesh of the CFD geometry. ANSYS-Mesh (v19) software was used for mesh generation purposes. Figure 3a shows a cross-sectional view of the CFD geometry, Figure 3b displays the arrangement of UAVs, and Figure 3c illustrates the meshes of the four-rotor model.

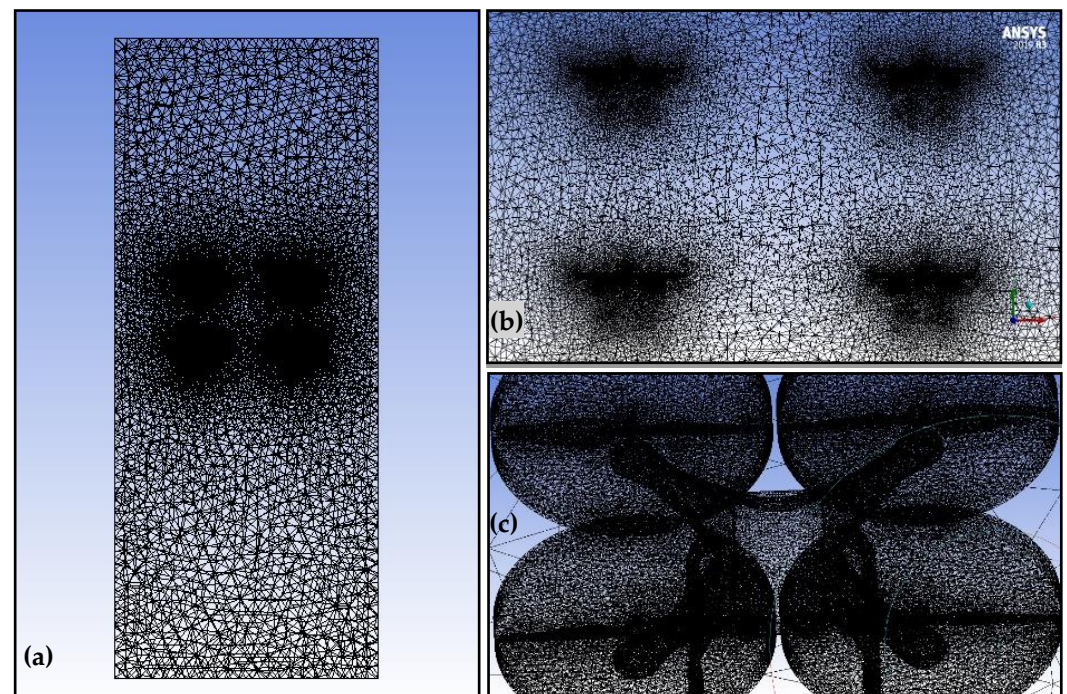


Figure 3. Mesh images: (a) cross-sectional view; (b) mesh of the swarm; (c) UAV mesh.

The parameters of the mesh are given in Table 1. Proximity and curvature captures were enabled, the growth rate was set as 1.1, the number of cells across the gaps was set as 5, and the target skewness was taken as 0.8. The total number of nodes in the meshes was approximately 23 million, with a total of about 16 million elements.

Table 1. Global mesh features.

Physics and Solver Preference	CFD, Fluent
Element Order	Quadratic
Growth Rate	1.1
Mesh Defeature Size	1, mm
Target Skewness	0.8
Smoothing	High
Pinch Tolerance	1.8 mm
Capture Proximity and Curvature	Yes
Curvature Min Size	2 mm
Curvature Normal Angle	18°
Proximity Min Size	2 mm
Num Cells Across Gap	5
Nodes	23,272,029
Elements	16,841,588

ANSYS-Fluent (v19) software was used as the CFD software. Moving mesh and moving reference frame methods were used as the solution methodology [2]. Simulations were conducted using the moving reference frame method. The cylindrical volumes shown in Figure 1 have been referred to as either moving or rotating zones. The cylindrical volumes shown in Figure 1 have been referred to as either moving or rotating zones. These zones were used to determine the rotation directions, rotation axes, and origin of the propellers.

The rotational speeds of UAV propellers may vary depending on the type of UAV, the cruise missions and the equipment carried. For instance, in research involving a fixed-wing UAV, the rotational speed was set at 6500 rpm [12]. However, the rotational speeds of propellers in rotary-wing UAVs may vary. Similarly, in quadcopter UAVs, the rotational speed of the propellers can change depending on factors such as propeller size, motor specifications, weight, and payload. In this study [2], a rotational speed of 1000 rad/s, which corresponds to 80% of the maximum rotational speed specified in the technical specifications of the UAV, was used for the simulation. It was assumed that the propellers rotate at speeds of 10,000 rpm in both clockwise and counterclockwise directions.

The Realizable $k-\epsilon$ model turbulence model in conjunction with enhanced wall treatment was selected, and a gravitational acceleration was given as 9.81 m/s^2 [17,18]. The fluid was defined as air, with a density of 1.225 kg/m^3 and a viscosity of $1.7894 \times 10^{-5} \text{ kg/(m} \cdot \text{s)}$. Within the swarm model, there were a total of 17 fluid zones with 16 rotating fluid domains and 1 external fluid domain. According to the global coordinate system on CFD, the origins, orientation vectors (X, Y, Z variables), and rotational speeds were separately defined for each of rotating fluid domain. In these definitions, orientation vectors were assumed to be in the Y+ direction, and 3 variables for rotating origin and 1 variable for rotational velocity were used. While 16 variables were used for a single UAV, a total of 64 parametric variables were defined for the 4 swarm UAVs within the CFD geometry. According to the general coordinate system on CFD, these variables including X, Y, and Z coordinate parameters and rotational velocities are listed in Table 2. It should be underlined that both horizontally and vertically adjacent UAVs maintained 75 cm between their closest propeller axes.

Table 2. Parameters of rotating-fluid domains: rotating-axis origin and rotational velocity.

Variables	X	Y	Z	Rev
Unit	m	m	m	RPM
UAV-1 Propeller Rear-Left	2.2	6.2	2.2	−10,000
UAV-1 Propeller Rear-Right	2.45	6.2	2.2	10,000
UAV-1 Propeller Front-Left	2.2	6.2	2.45	10,000
UAV-1 Propeller Front-Right	2.45	6.2	2.45	−10,000
UAV-2 Propeller Rear-Left	3.2	6.2	2.2	−10,000
UAV-2 Propeller Rear-Right	3.45	6.2	2.2	10,000
UAV-2 Propeller Front-Left	3.2	6.2	2.45	10,000
UAV-2 Propeller Front-Right	3.45	6.2	2.45	−10,000
UAV-3 Propeller Rear-Left	2.2	5.45	2.2	−10,000
UAV-3 Propeller Rear-Right	2.45	5.45	2.2	10,000
UAV-3 Propeller Front-Left	2.2	5.45	2.45	10,000
UAV-3 Propeller Front-Right	2.45	5.45	2.45	−10,000
UAV-4 Propeller Rear-Left	3.2	5.45	2.2	−10,000
UAV-4 Propeller Rear-Right	3.45	5.45	2.2	10,000
UAV-4 Propeller Front-Left	3.2	5.45	2.45	10,000
UAV-4 Propeller Front-Right	3.45	5.45	2.45	−10,000

3. Results and Discussion

The Computational Fluid Dynamics analyses of four unmanned aerial vehicles with a swarm configuration were performed in this research, and the resulting data were evaluated. The processing of CFD results is referred to as post-processing, and this procedure was carried out using CFD-Post (v19) software. Due to factors such as mesh characteristics and solution methods, numerical discrepancies may occur in CFD results. Additionally, in the CFD-Post software, data regions were defined using Pearl, CCL, and CEL programming languages to facilitate visualization and graphing.

First, all data lines were generated to create the plots of CFD results. Figure 4 displays these data lines, which were obtained from both horizontal and vertical lines drawn over the front and rear propeller axes. Plots obtained over the front propeller's axes are labeled with the "F" identifier, while those obtained over the rear propeller's axes are labeled with the "B" identifier. Additionally, the same colors which are used in the data lines are used in the plots as well. There may be differences in the data taken over the front and rear propeller's axes. Therefore, the data obtained over the front propeller's axes were chosen to align with the overall structure.

In Figure 4, horizontal lines and planes representing the Y general coordinate values are displayed on the left side. In these horizontal data lines, the range of X values was taken from 100 cm to 465 cm, and the Z value was taken as 245 cm (F) or 220 cm (B). Additionally, on the upper side of the figure, vertically created data lines are shown together with X values. In the vertical lines, the Y variables were taken within the range of 0 cm to 1000 cm, and the Z variables were taken as either 245 or 220; the X variables were taken same values in the figure. The propeller axes X220, X245, X320, and X345 shown in the top-left and top-right of Figure 4 correspond to UAV-1 and UAV-2. All plots and contours were generated using these definitions.

Figure 5 depicts the contours of velocity distributions in planes defined by the front and rear propeller axes as well as the midpoint of the UAVs. Figure 5a represents the contours from the rear propeller plane, Figure 5b from the midplane of the UAVs, and Figure 5c from the front propeller plane. In the velocity distribution, UAVs positioned below were affected by the airflow generated by the UAVs positioned above. When examining the velocity contours, it has been determined that velocity distributions approximately 2.5–3 m below the UAVs exhibited homogeneity along the propeller axes and vertical axes taken between the UAVs. Velocity distributions are shown by taken over the horizontal lines in Figure 8 and taken over the vertical lines in Figure 9. It has been observed that a phenomenon similar to the "Near-field mixing phenomenon", which occurs in near-jet

flows, manifests in the airflow produced by the UAVs [22]. This study introduces a new phenomenon named the “Near-UAV Phenomenon” resulting from the investigation of close UAV effects.

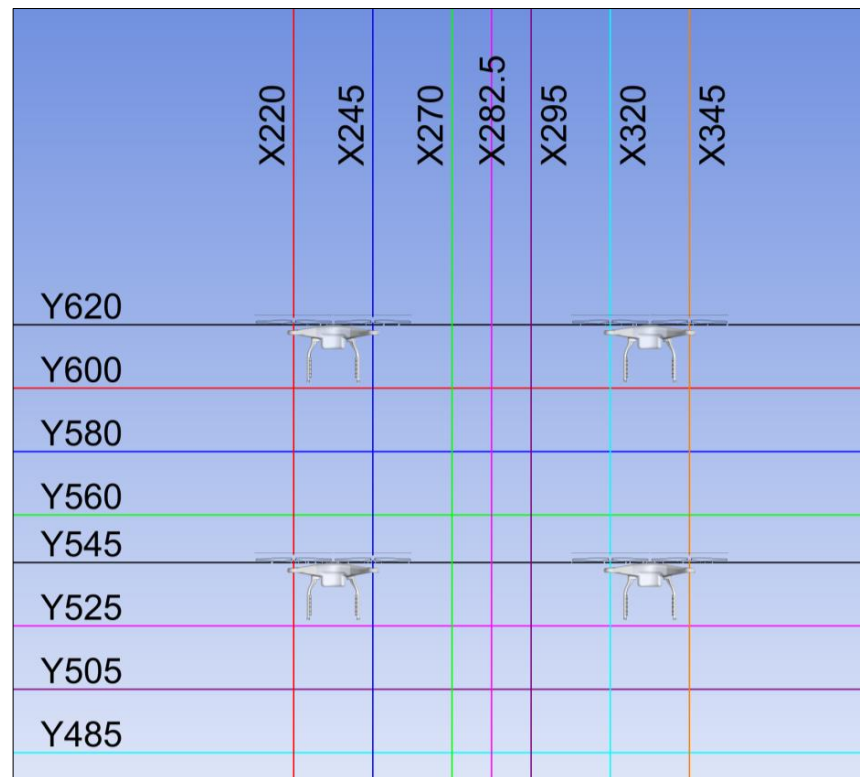


Figure 4. Vertical and horizontal data lines of swarm UAVs.

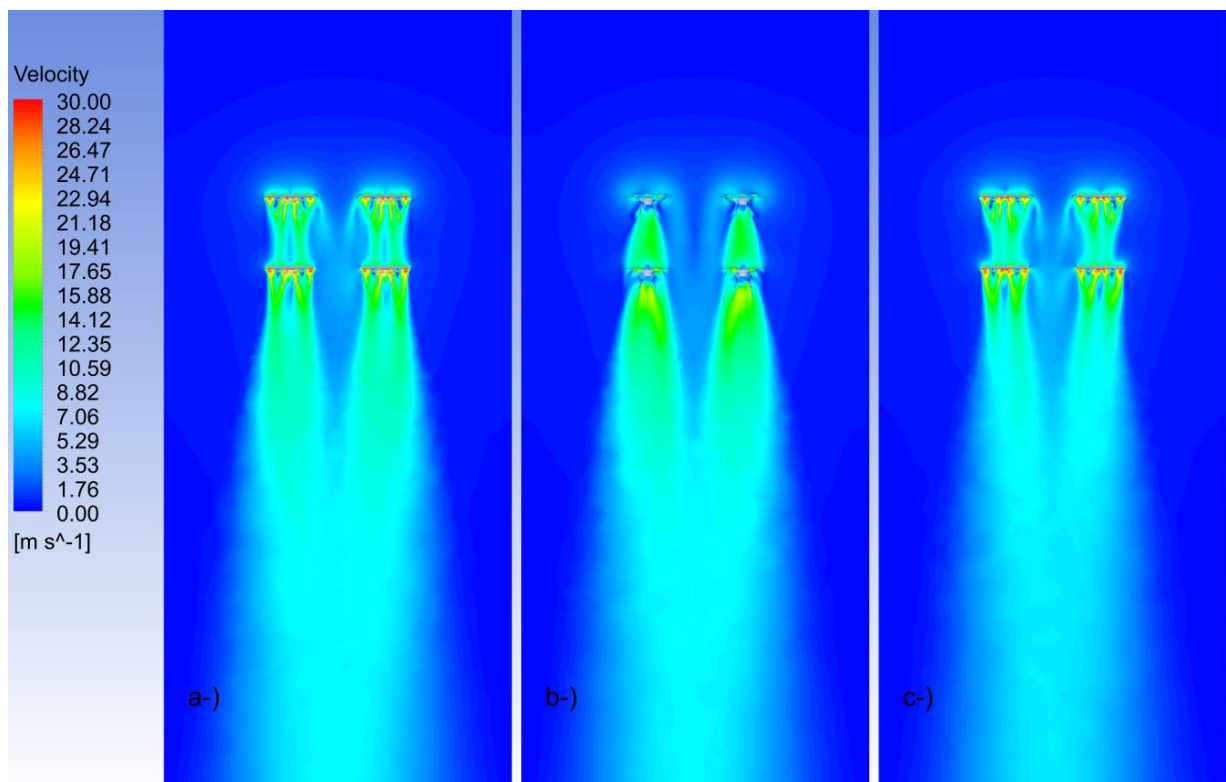


Figure 5. Velocity contours: (a) rear plane; (b) midplane of the UAVs; (c) front plane.

Velocity contours obtained over vertical planes in Figure 6 are presented comparatively, while the corresponding locations of these planes are indicated in Figure 4. The distances of these planes, relative to the propeller planes of the UAVs positioned above and below are specified as follows; Y560 and Y485 60 cm, Y580 and Y505 40 cm, Y600 and Y525 20 cm, and Y535 and Y610 10 cm under. The velocity value ranges in these contours are 1 to 22 m/s.

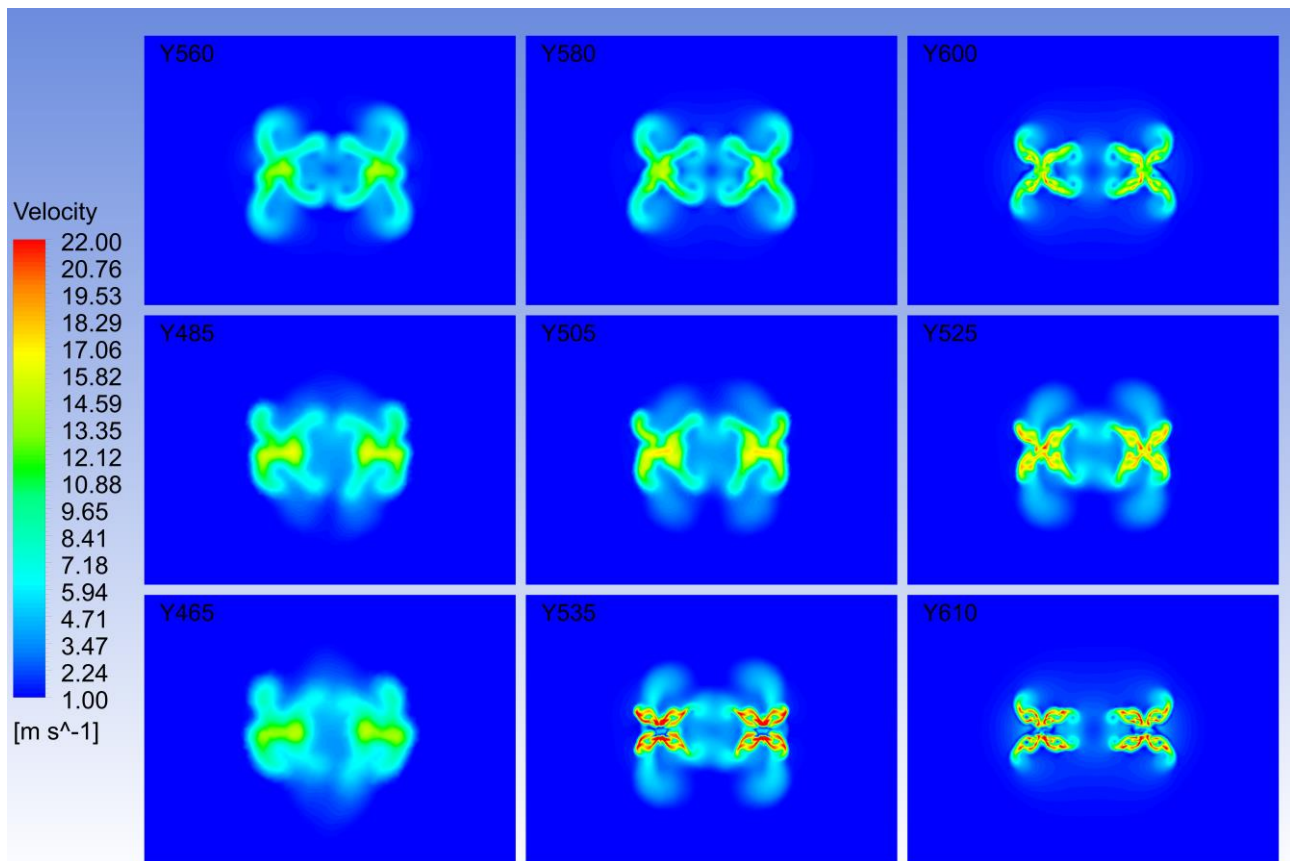


Figure 6. Velocity contours in vertical planes.

When examining the Y600 and Y525 vertical planes, it is clearly observed that the velocity distribution increased after the vehicles positioned below. The formation of vortices was observed between the UAVs at Y610, Y600, and Y580 vertical planes. In the Y560 contour, the outward flow generated by the propellers of the UAVs positioned above was affected by the flow generated by the UAVs positioned below, resulting in distribution up to Y505 while passing near the vehicles. This distribution is visually represented with streamlines in Figure 7.

The streamlines generated by the UAVs are illustrated in Figure 7, where Figure 7a shows the top view and Figure 7b shows the front view. By following the paths of the streamlines and the resulting trajectories, the flows around the vehicles were determined, and the velocities along the streamlines were also shown. The turbulence occurring between the UAVs was visualized. Additionally, it was observed that the flow generated by the outer propellers of the UAVs positioned above passed through between the vehicles.

The velocity values are presented as plots in Figure 8, representing the front propellers. The horizontal lines Y600 and Y525 are located 20 cm below the rotation centers of the UAV propellers. In these regions, velocities measured from the tip of the propellers are approximately 18–19 m/s, and the velocity distributions are high due to the proximity to the propellers.

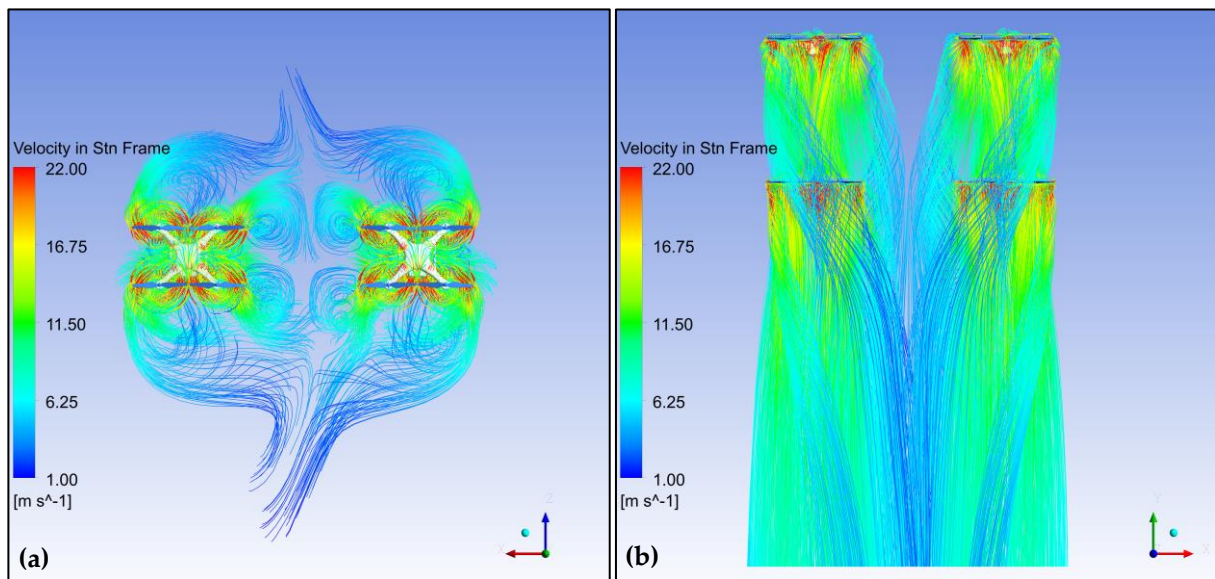


Figure 7. Streamline velocity: (a) top view; (b) front view.

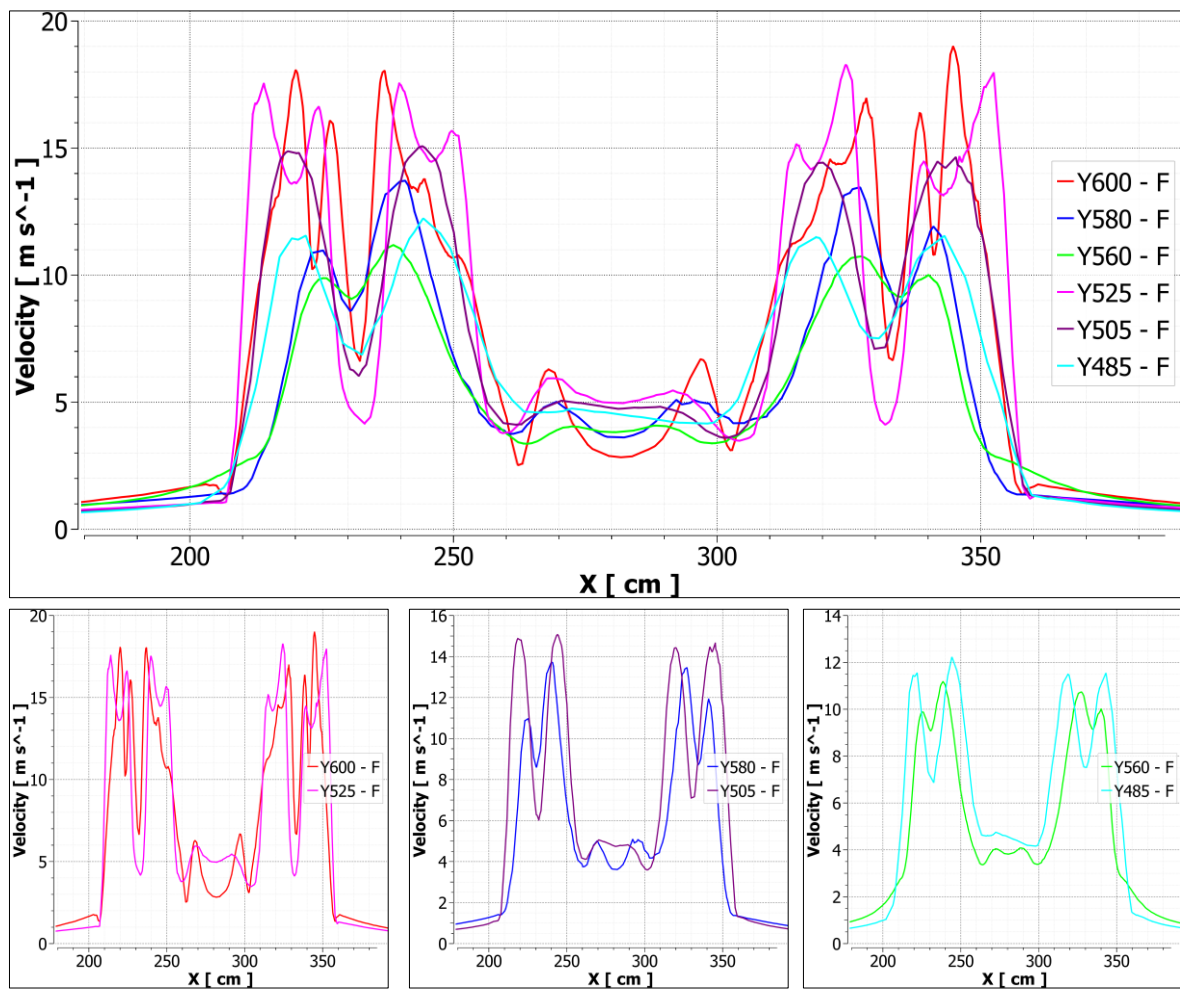


Figure 8. Velocity plots of front(F) propeller taken from horizontal lines.

The Y580 to Y505 lines are 40 cm below the rotation centers. The velocities under the propeller tips are approximately 14 m/s in the UAVs positioned above, while velocities are approximately 15 m/s in the vehicles positioned below. The distribution of velocities

increases towards the propeller tips. Additionally, when examining the airflow between adjacent vehicles horizontally, velocity distributions are increased.

The Y560 to Y485 lines are located 60 cm below the centers of rotation. The velocity distribution at Y485 is increased compared to Y560 due to the downward flow. While the speed is 12 m/s at Y560, it has increased to 14 m per second at Y485. It is clearly seen that the airflow produced by the upper UAVs affects the lower UAVs with the horizontal lines.

Velocity distributions on vertical data lines are shown in Figure 9. Data were collected along vertical lines ranging from X220 to X345. The values X220, X245, X320, and X345 represent the vertical lines taken from the propeller axes, while X270, X282.5 (midpoint), and X295 are used to determine the vertical lines between the UAVs. The velocity plot obtained along these vertical lines is illustrated in Figure 9. The values of Y at 620 cm and 545 cm correspond to the rotating-axis origin of the propellers.

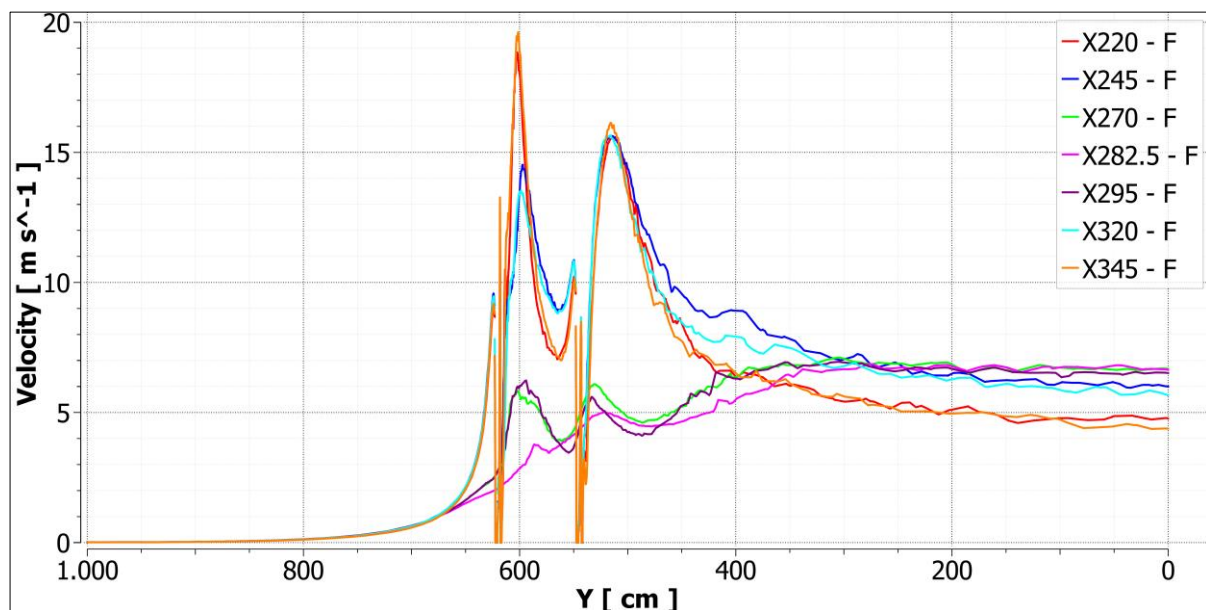


Figure 9. Velocity on vertical lines.

Examining the graph of data obtained from vertical lines indicates that the velocity under the upper vehicles at Y = 600 cm reaches approximately 19 m/s on the X220 and X345 lines, while it is up to about 14 m/s on the X245 and X320 lines. On the X220, X245, X320, and X345 axis lines, this velocity under the lower vehicles is approximately 15 m/s at Y = 525 cm, while it reaches its highest value around Y = 515 cm, reaching approximately 15.5 m/s.

The velocity distribution between UAVs is examined both horizontally in Figure 8 and vertically in Figure 9. It is observed that speeds reach up to 7 m/s at X270 to X295 near the propeller's axis and up to 4 m/s at the midpoint X282.5. The developing airflow converges towards the airflow on the propeller axes, approximately 2.5 m under the UAVs-swarm, around Y = 300 cm, reaching speeds ranging between 7 and 8 m per second. Furthermore, at Y = 0 cm, the velocity at the outer axes (X345–X220) is approximately 1 m/s lower compared to other lines. All of this illustrates the impact of close-range UAV swarm flights on velocity.

Figure 10 illustrates the total pressure contours taken from the front and rear propeller planes, as well as the mid-plane of the UAVs. Figure 10a illustrates the contours taken from the rear plane, Figure 10b shows the contours from the mid-plane, and Figure 10c displays the contours from the front plane in a comparative manner. Upon detailed examination of these contours, it is evident that low-pressure regions formed in the middle of the UAVs. Approximately −50 Pa pressure is observed on the opposing propeller surfaces of the UAVs positioned at the top. While it can be anticipated that the UAVs positioned above could affect those below, it has also been determined that they are affected by

the airflow generated as they approach each other horizontally. The total pressure lines resulting from the interaction of near-UAVs are presented both horizontally and vertically in Figures 11 and 12. The total pressure below 2.5 m from the UAVs is approximately 40 Pa.

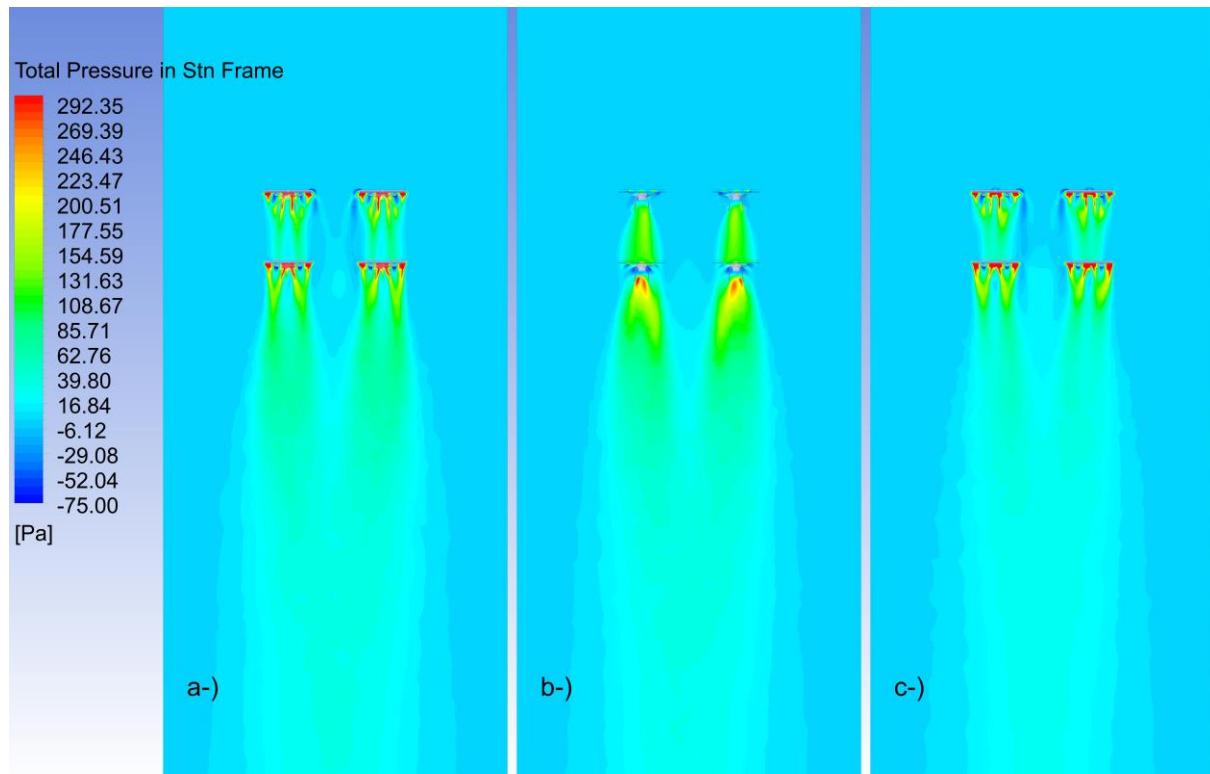


Figure 10. Total pressure contours: (a) rear plane; (b) mid-plane; (c) front plane.

The distributions of total pressure are illustrated in Figure 11. Upon examination of the data lines Y600 and Y525, located 20 cm under the origins of the propeller rotating axis at $X = 220$ cm and 345 cm, approximately 200 Pa is observed at Y600, while it is around 130 Pa at Y525. As the propeller tips progress towards, this pressure increases up to 200 Pa. On the lines located 40 cm under the propeller origins, a maximum total pressure of 110 Pa at Y580 and 130 Pa at Y505 is observed. As the downward flow progresses, the distribution of pressure lines is observed to spread. On lines Y560 to Y485, under the lower row of UAVs, the pressure reaches up to 85 Pa at Y485, while under the upper row of UAVs, it remains under 65 Pa at Y560.

The total pressure plot derived from vertical data lines is displayed in Figure 12. These vertical data lines are taken from a range of X220 to X345. Between Y 620 cm and 600 cm, the total pressure is increased at the UAVs positioned above. Y = 600 cm, the pressure reaches approximately 210 Pa on the X220 and X345 lines, while it is around 125 Pa on the X245 and X320 lines. It remains below 100 Pa at Y = 580 cm and is around 40 Pa at Y = 560 cm. Subsequent to the UAVs positioned below, the pressure is measured at 120 Pa at Y = 525 cm and reaches around 145 Pa near Y = 515 cm. The pressure between the axis lines ranges between 30 and 40 Pa at Y = 300 cm.

Upon examination of the central axis lines (X270, X282.5, and X295), under the upper row of UAVs, it is approximately 10 Pa at Y = 600 cm, and a vacuum effect is observed as we approach towards the lower row of UAVs. The pressure converges at the same level as the other axis lines towards Y = 300 cm. Pressure distributions become more uniform with distance from the UAVs.

Dynamic and static pressure values are also illustrated in Figure 13. The pressure and dynamic pressure plots taken from the front and rear axis planes are comparatively presented. The planes representing the front are denoted as “F”, whereas those representing

the rear are denoted as “B”. The Y-axis range in these graphs is taken between 700 cm and 300 cm.

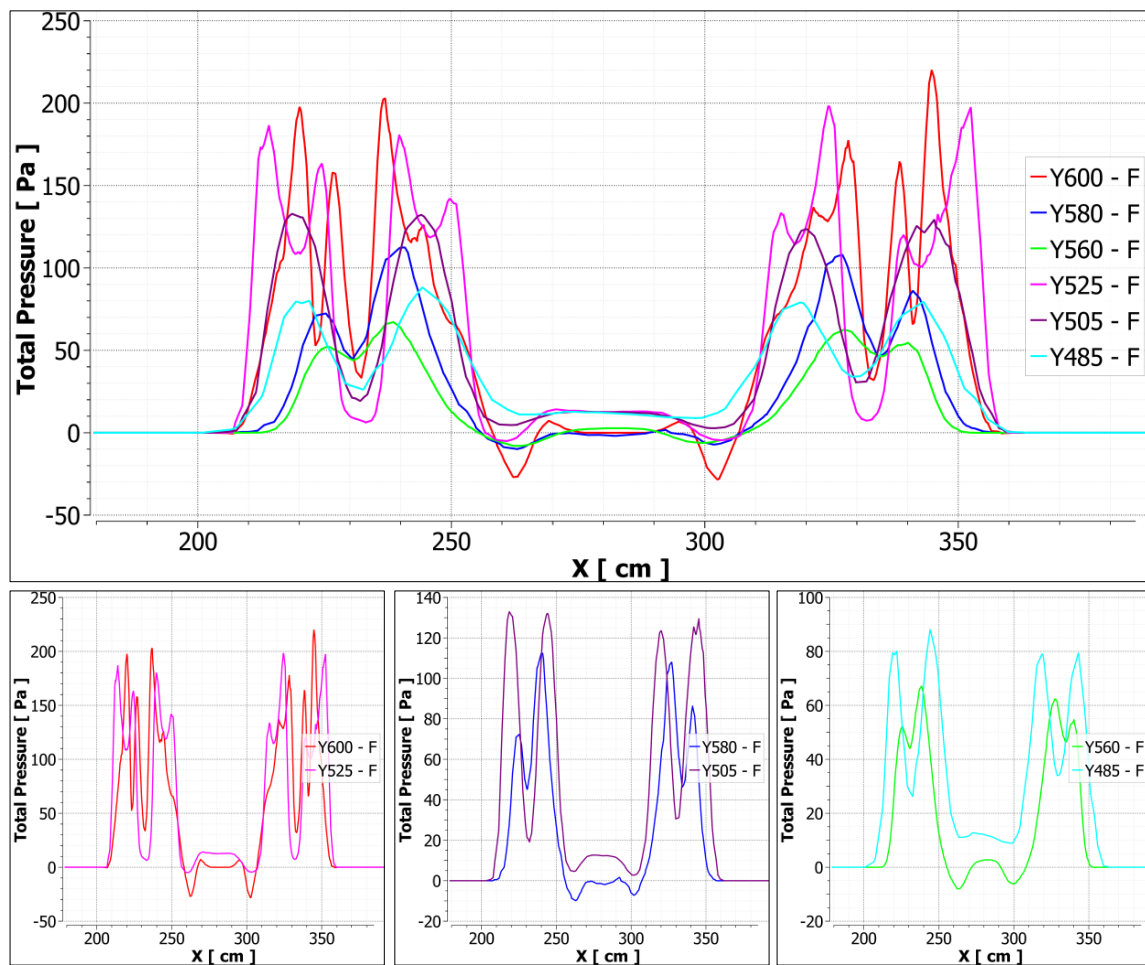


Figure 11. Total pressure on horizontal lines.

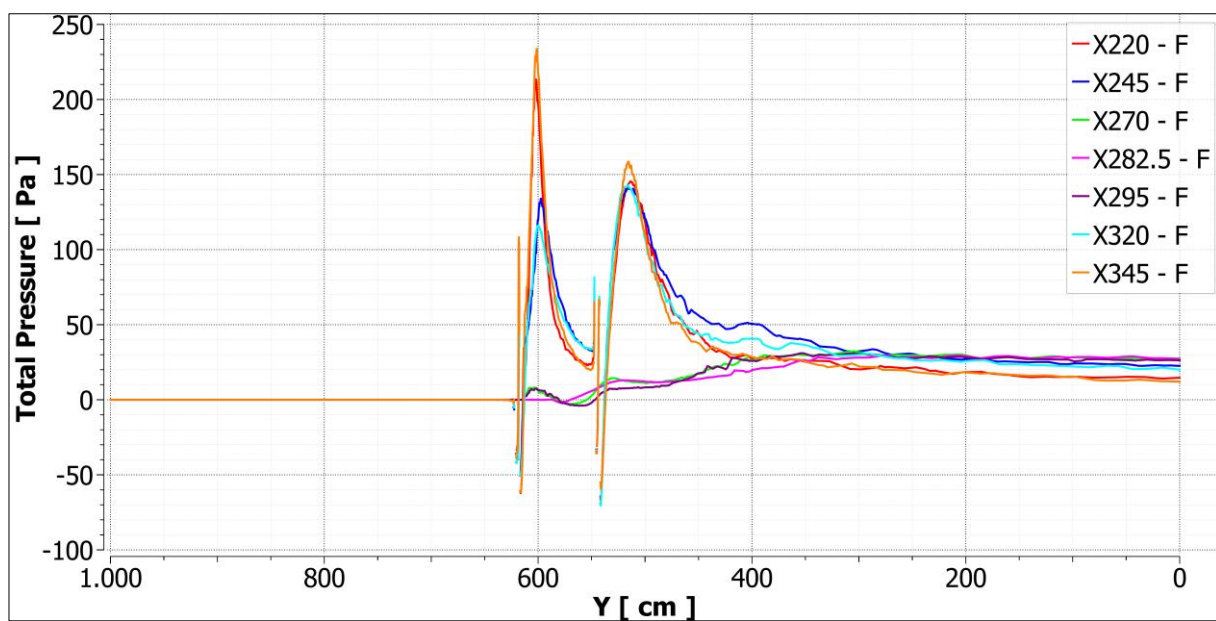


Figure 12. Total pressure on vertical lines.

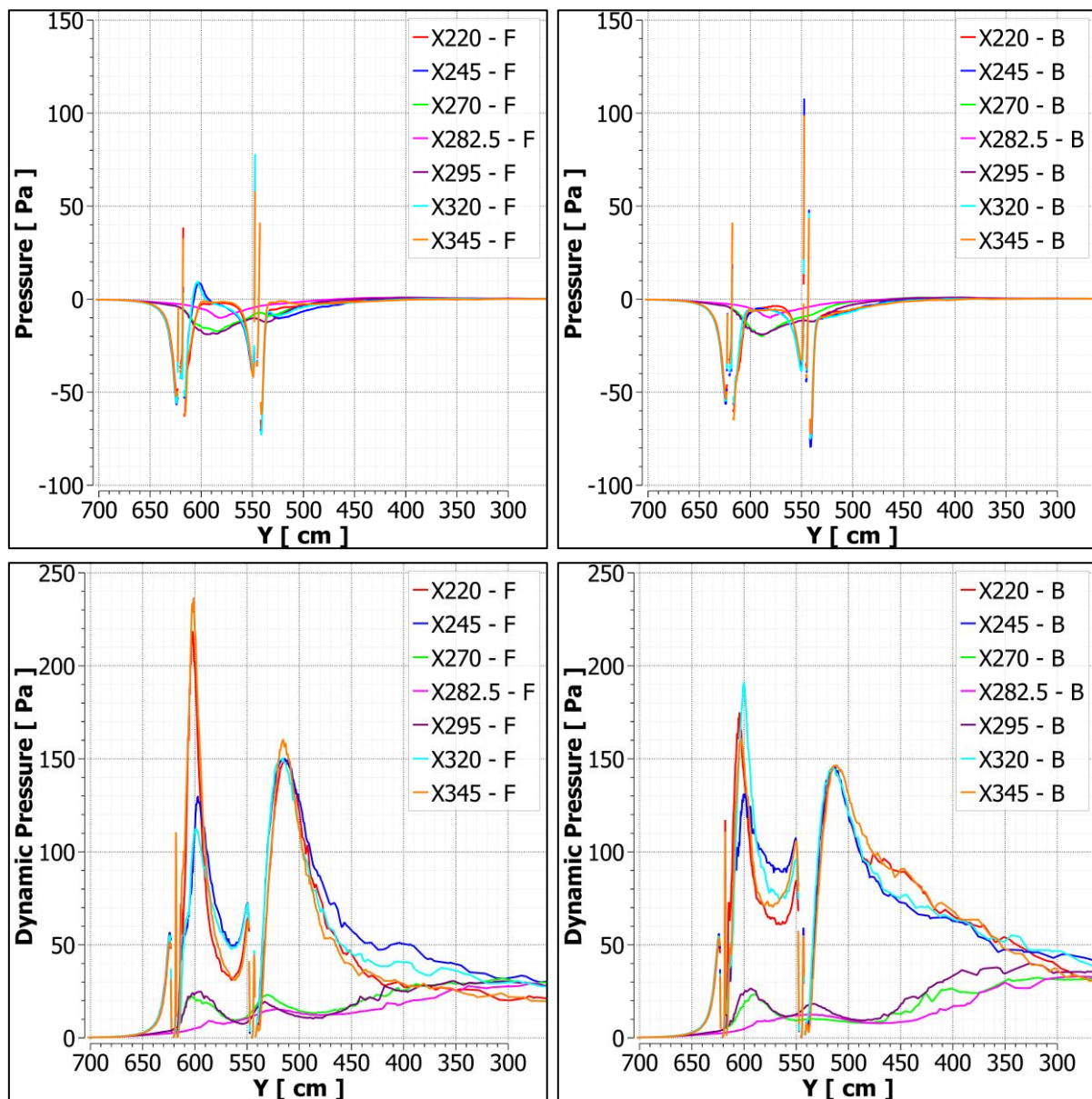


Figure 13. Pressure and dynamic pressure on vertical lines.

The dynamic pressure at $Y = 600$ cm is above 200 Pa on the “F” lines, while it remains below 200 Pa on the “B” lines. At $Y = 520$ cm, the “F” lines can reach up to 150 Pa, contrasting with the “B” lines which stay below 150 Pa. At $Y = 400$ cm, the “F” lines stay below 50 Pa, while the “B” lines are above 50 Pa. For the pressure plot, similarities are seen on the front and rear axis plane. Variations between the front (F) and rear (B) data are observed due to the structural design of the UAVs illustrated in Figure 1.

The turbulence kinetic energy (TKE) contours generated from three distinct planes are shown in Figure 14. The contours obtained from the rear plane are shown in Figure 14a, those from the mid-plane in Figure 14b, and those from the front plane in Figure 14c. A discernible increase in turbulence kinetic energy is observed amidst the UAVs. Upon close examination of the contours derived from the mid-plane, it is observed that the TKE can reach up to approximately $80 \text{ m}^2/\text{s}^2$ in proximity to the UAV fuselages. The TKE values obtained over horizontal and vertical lines are presented in Figures 15 and 16, respectively. Notably, the peak levels of TKE are observed under the UAV body due to the flow dynamics developing under the propellers.

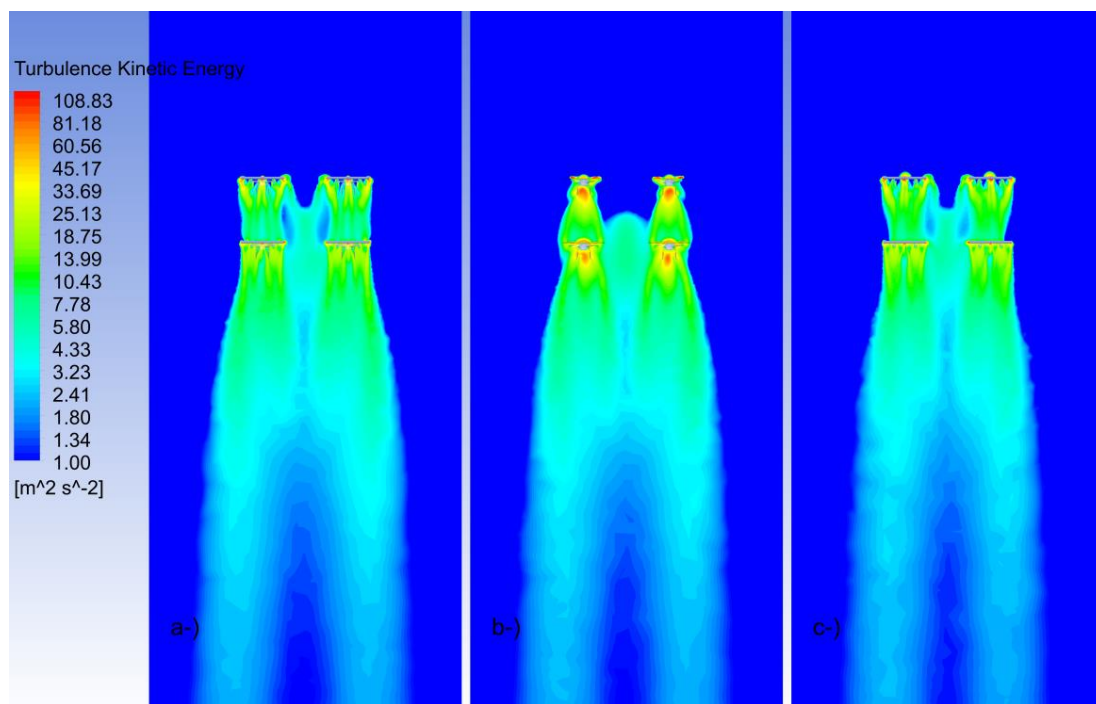


Figure 14. Turbulence kinetic energy contours: (a) rear plane; (b) mid-plane; (c) front plane.

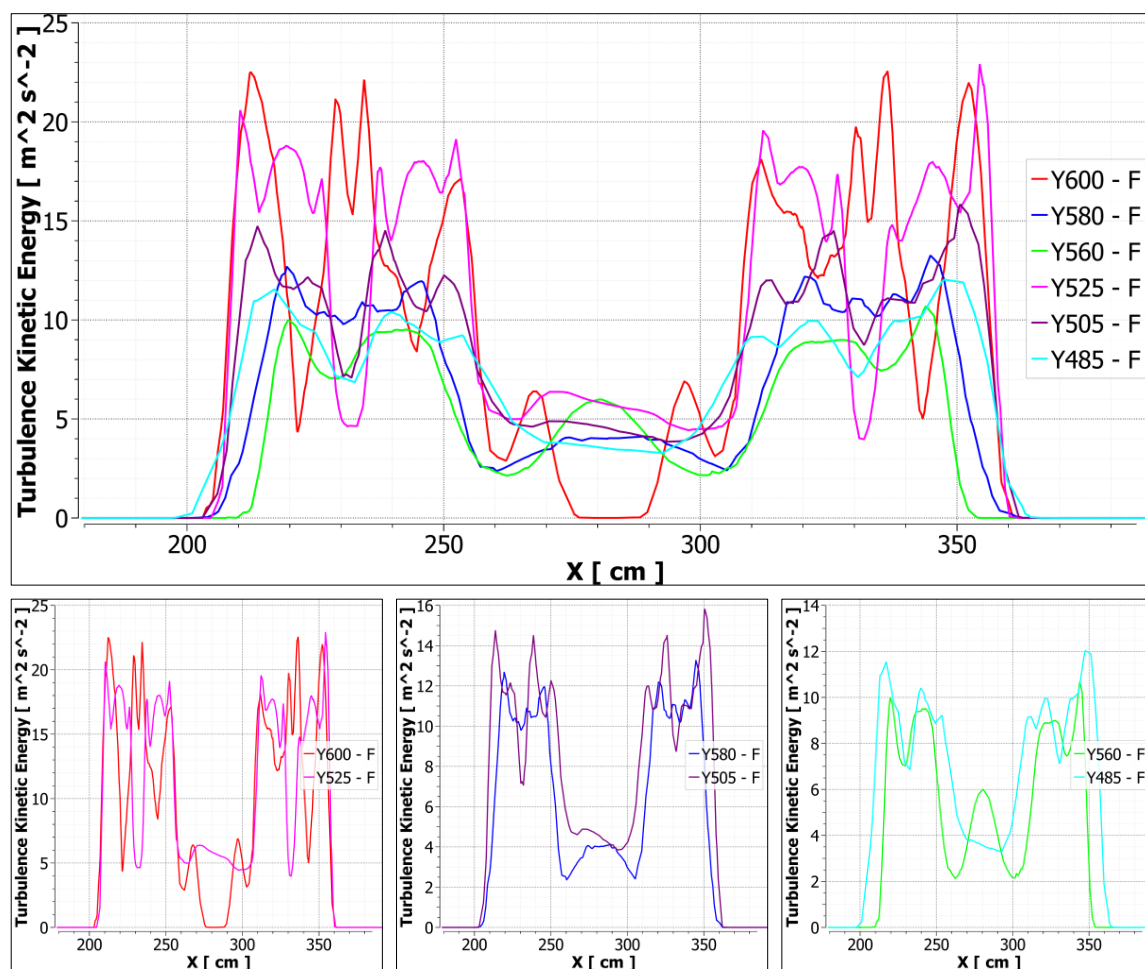


Figure 15. Turbulent kinetic energy on horizontal lines.

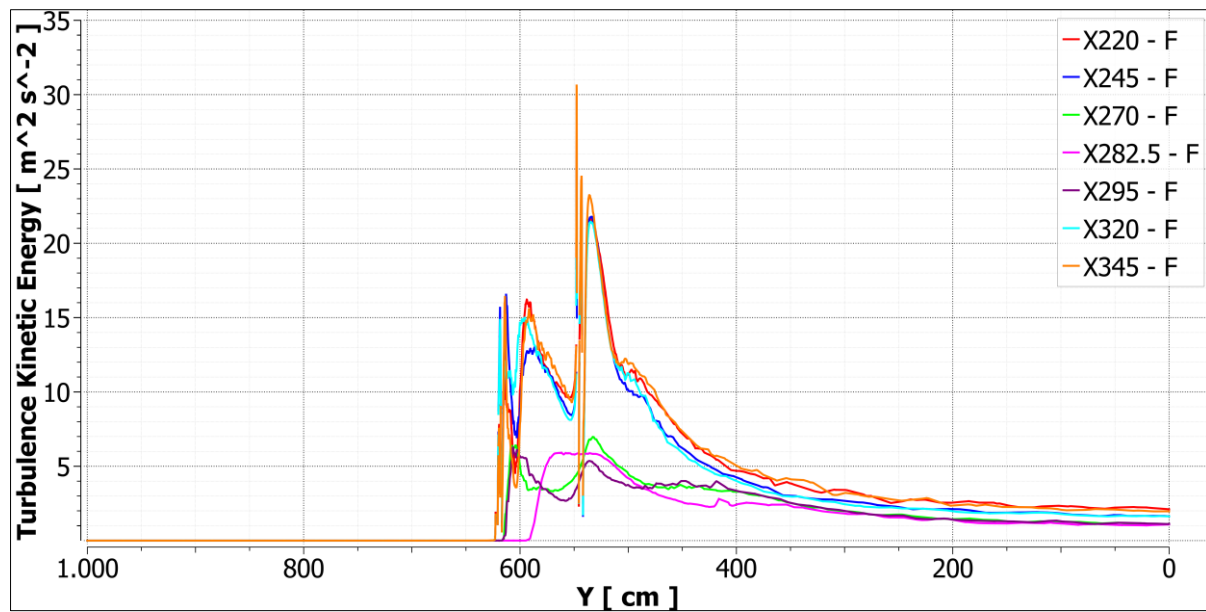


Figure 16. Turbulent kinetic energy on vertical lines.

The distribution of turbulence kinetic energy (TKE) over horizontal lines is illustrated in Figure 15. The lines Y600–Y525, Y580–Y505, and Y560–Y485 are positioned 20, 40, and 60 cm below the rotation centers of the propellers, respectively. The X positions of the propeller centers are at 220, 245, 320, and 345 cm. In these regions, the TKE reaches a maximum of $23 \text{ m}^2/\text{s}^2$. Also, TKE values are increased closer to the propeller tips of the UAVs.

When the graphics are examined, at the center of the vehicles (on X 282.5 cm), the TKE is zero on the Y600 line while it increases to $4 \text{ m}^2/\text{s}^2$ on Y580 and up to $6 \text{ m}^2/\text{s}^2$ on Y560. After the vehicles positioned below, the TKE reaches $6 \text{ m}^2/\text{s}^2$ at Y525, $\sim 4.5 \text{ m}^2/\text{s}^2$ at Y505, and rises to $6 \text{ m}^2/\text{s}^2$ at Y485. In addition, the TKE distribution that developed on Y580 decreases close to the propellers of lower-row vehicles at X 260 cm and 305 cm on the Y560 line. Also, it is observed that the TKE distribution of the lower row vehicles (Y485) increases compared to the vehicles at the upper row (Y560).

The distribution of turbulence kinetic energy obtained from vertical lines is illustrated in Figure 16. In this plot, the UAV propeller axes are positioned at Y values 620 cm and 545 cm. After the upper row vehicles, TKE values reach up to $16 \text{ m}^2/\text{s}^2$, while after the lower vehicles, these values are increased to over $20 \text{ m}^2/\text{s}^2$. The TKE between UAVs reaches up to $6 \text{ m}^2/\text{s}^2$ at the X282.5 line. The TKE generated by the swarm UAVs decreases below $5 \text{ m}^2/\text{s}^2$ at Y = 400 cm.

4. Conclusions

In this research, a specific configuration of multiple unmanned aerial vehicles (UAVs) in a swarm was investigated and evaluated. The arrangement of four UAVs, with two positioned above and two below along the same propeller axis, was extensively studied. The analysis focused on the aerodynamics profile generated by UAV propellers rotating at 10,000 revolutions per minute. Particularly, interactions occurring in close flights were observed at 75 cm between adjacent axes. The effects of UAVs positioned above on UAVs positioned below were meticulously examined and their impact on swarm behavior was investigated. Moreover, investigations into pressure, velocity, and turbulent kinetic energy profiles based on these distances between UAVs revealed the presence of turbulence among them.

This study differentiates itself by being the first to model and examine the specific configuration of multiple unmanned aerial vehicles within a swarm using Computational Fluid Dynamics. It focused on the aerodynamic behavior of a square-shaped swarm formation,

which adds a unique dimension to the analysis. Additionally, advanced CFD techniques, including Multiple Reference Frame (MRF), the $k-\epsilon$ turbulence model, parametric placement, and analyses were employed to simulate the swarm configuration.

The methodology employed in this research for analyzing swarm formations can serve as a valuable tool for future research and development in UAV swarm technology. It lays the groundwork for further advancements in swarm coordination, navigation, and mission planning.

This research marks a significant milestone in understanding the aerodynamic behavior of UAVs in a square-shaped swarm formation flight and optimizing their aerodynamic performance. Additionally, the “near-UAV phenomenon” observed during formation/swarm flights was first defined in this research. Defining proximity zones between UAVs should be considered for ensuring the safety of flight operations in UAV swarms. In future studies, proximity distances between drones in swarm formations should be considered for safety.

Author Contributions: Conceptualization, A.T.İ. and B.Ç.; methodology, B.Ç.; software, B.Ç.; validation, B.Ç.; formal analysis, B.Ç.; investigation, B.Ç.; resources, B.Ç.; data curation, B.Ç.; writing—original draft preparation, A.T.İ. and B.Ç.; writing—review and editing, A.T.İ.; visualization, B.Ç.; supervision, A.T.İ.; project administration, A.T.İ.; funding acquisition, B.Ç. All authors have read and agreed to the published version of the manuscript.

Funding: This research was supported by The Scientific and Technological Research Council of Türkiye (TÜBİTAK), 2211-C National PhD Scholarship Program in the Priority Fields in Science and Technology 2022/2 (1649B032209721) as well as by The Council of Higher Education (CoHE) of the Republic of Türkiye, YÖK 100/2000 PhD Scholarship Program.

Institutional Review Board Statement: Not applicable.

Informed Consent Statement: Not applicable.

Data Availability Statement: The raw data supporting the conclusions of this article will be made available by the authors on request.

Acknowledgments: This study is based on the Ph.D. thesis titled “Computational Fluid Dynamics Analysis of Rotary-Wing Unmanned Aerial Vehicles Moving as a Swarm”, authored by Berkay ÇETİN under the supervision of Ahmet Talat İNAN at Marmara University, Institute of Science and Technology, Department of Mechanical Engineering in İstanbul, Türkiye. The author, B.Ç., extends his sincere thanks to his supervisor, Ahmet Talat İNAN, for his invaluable guidance and support throughout the research process. The author also wishes to acknowledge The Scientific and Technological Research Council of Türkiye (TÜBİTAK) and The Council of Higher Education (CoHE) of the Republic of Türkiye (YÖK) for their support. Additionally, the author expresses his deep gratitude to Özlem ÇETİN, Mustafa ÇETİN, and Hülya ÇETİN for their unwavering support and understanding throughout this academic journey.

Conflicts of Interest: The authors declare no conflicts of interest. The funders had no role in the design of the study; in the collection, analyses, or interpretation of data; in the writing of the manuscript; or in the decision to publish the results.

References

1. Paz, C.; Suárez, E.P.; Gil, C.; Vence, J. Assessment of the Methodology for the CFD Simulation of The Flight of a Quadcopter UAV. *J. Wind Eng. Ind. Aerodyn.* **2021**, *218*, 104776. [\[CrossRef\]](#)
2. Paz, C.; Porto, E.S.; Gil, C.; Baker, C. CFD Analysis of the Aerodynamic Effects on the Stability of the Flight of a Quadcopter UAV in the Proximity of Walls and Ground. *J. Wind Eng. Ind. Aerodyn.* **2020**, *206*, 104378. [\[CrossRef\]](#)
3. Atmaca, M.; Çetin, B.; Yılmaz, E. CFD Analysis of Unmanned Aerial Vehicles (UAV) Moving in Flocks. *Acta. Phys. Pol. A* **2019**, *135*, 694–696. [\[CrossRef\]](#)
4. Hari, B.; Brouwer, J.P.; Dhir, A.; Steinberger-Wilckens, R. A Computational Fluid Dynamics and Finite Element Analysis Design of a Microtubular Solid Oxide Fuel Cell Stack for Fixed Wing Mini Unmanned Aerial Vehicles. *Int. J. Hydrogen Energy* **2019**, *44*, 8519–8532. [\[CrossRef\]](#)
5. Panagiotou, P.; Kaparos, P.; Yakinthos, K. Winglet Design and Optimization for a Male UAV Using CFD. *Aerosp. Sci. Technol.* **2014**, *39*, 190–205. [\[CrossRef\]](#)

6. Abichandani, P.; Lobo, D.; Ford, G.; Bucci, D.; Kam, M. Wind Measurement and Simulation Techniques in Multi-Rotor Small Unmanned Aerial Vehicles. *IEEE Access* **2020**, *8*, 54910–54927. [\[CrossRef\]](#)
7. Wen, G.-L.; Wu, D.; Yin, H.; Zhang, D.-B. Coupled CFD/MBD Method for a Tilt Tri-rotor UAV in Conversion of Flight Modes. *Int. J. Comput. Fluid Dyn.* **2020**, *34*, 363–379. [\[CrossRef\]](#)
8. Kim, C.; Chung, J. Aerodynamic Analysis of Tilt-Rotor Unmanned Aerial Vehicle with Computational Fluid Dynamics. *J. Mech. Sci. Technol.* **2006**, *20*, 561–568. [\[CrossRef\]](#)
9. Wang, H.-P.; Sun, W.-H.; Zhao, C.-L.; Zhang, S.-J.; Han, J.-D. Dynamic Modeling and Control for Tilt-Rotor UAV Based on 3D Flow Field Transient CFD. *Drones* **2022**, *6*, 338. [\[CrossRef\]](#)
10. Yang, S.; Tang, Q.; Zheng, Y.; Liu, X.; Chen, J.; Li, X. Model Migration for CFD and Verification of a Six-Rotor UAV Downwash. *Int. J. Agric. Biol. Eng.* **2020**, *13*, 10–18. [\[CrossRef\]](#)
11. Mazhar, F.; Khan, A.M.; Chaudhry, I.A.; Ahsan, M. On Using Neural Networks in UAV Structural Design for CFD Data Fitting and Classification. *Aerosp. Sci. Technol.* **2013**, *30*, 210–225. [\[CrossRef\]](#)
12. Sandoval, P.; Cornejo, P.; Tinapp, F. Evaluating The Longitudinal Stability of an UAV Using a CFD-6DOF Model. *Aerosp. Sci. Technol.* **2015**, *43*, 463–470. [\[CrossRef\]](#)
13. Kapsalis, S.; Panagiotou, P.; Yakinthos, K. CFD-Aided Optimization of a Tactical Blended-Wing-Body UAV Platform Using the Taguchi Method. *Aerosp. Sci. Technol.* **2021**, *108*, 106395. [\[CrossRef\]](#)
14. Ni, M.; Wang, H.J.; Liu, X.D.; Liao, Y.L.; Fu, L.; Wu, Q.Q.; Mu, J.; Chen, X.Y.; Li, J. Design of Variable Spray System for Plant Protection UAV Based on CFD Simulation and Regression Analysis. *Sensors* **2021**, *21*, 638. [\[CrossRef\]](#)
15. Zhu, H.; Li, H.Z.; Zhang, C.; Li, J.X.; Zhang, H.H. Performance Characterization of the UAV Chemical Application Based on CFD Simulation. *Agronomy* **2019**, *9*, 308. [\[CrossRef\]](#)
16. Götten, F.; Havermann, M.; Braun, C.; Marino, M.; Bil, C. Wind-Tunnel and CFD Investigations of UAV Landing Gears and Turrets—Improvements in Empirical Drag Estimation. *Aerosp. Sci. Technol.* **2020**, *107*, 106306. [\[CrossRef\]](#)
17. Siddiqi, Z.; Lee, J.W. A Computational Fluid Dynamics Investigation of Subsonic Wing Designs for Unmanned Aerial Vehicle Application. *Proc. Inst. Mech. Eng. Part G J. Aerosp. Eng.* **2019**, *233*, 5543–5552. [\[CrossRef\]](#)
18. Yang, F.B.; Xue, X.Y.; Zhang, L.; Sun, Z. Numerical Simulation and Experimental Verification on Downwash Air Flow of Six-Rotor Agricultural Unmanned Aerial Vehicle in Hover. *Int. J. Agric. Biol. Eng.* **2017**, *10*, 41–53. [\[CrossRef\]](#)
19. Chen, Q.; Zhang, J.; Zhang, C.; Zhou, H.; Jiang, X.; Yang, F.; Wang, Y. CFD Analysis and RBFNN-Based Optimization of Spraying System for a Six-Rotor Unmanned Aerial Vehicle (UAV) Sprayer. *Crop Prot.* **2023**, *174*, 106433. [\[CrossRef\]](#)
20. Dbouk, T.; Drikakis, D. Quadcopter Drones Swarm Aeroacoustics. *Phys. Fluids* **2021**, *33*, 057112. [\[CrossRef\]](#)
21. ANSYS Inc. *Ansys Fluent Theory Guide*; ANSYS Inc.: Canonsburg, PA, USA, 2019.
22. Xie, Z.; Xie, J.; Jiang, W.; Du, W.Z. Numerical and Experimental Investigation of Near-Field Mixing in Parallel Dual Round Jets. *Int. J. Aerosp. Eng.* **2016**, *2016*, 7935101. [\[CrossRef\]](#)

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.