Drone Design and Aerodynamic Analysis Project Report

Project Overview

This project focuses on the **design and analysis of a compact drone** by Prathmesh Deepak Gondkar. The primary objectives were to achieve **aerodynamic optimization**, verify the **stability** of the design, and study the interaction between various drone components during operation. The drone was modelled in 3D and subjected to simulation studies to refine its performance. Through iterative design and simulation, the project evaluated how well the drone's components (frame, impeller, gears, etc.) work together to maintain stable flight and efficient airflow. Overall, the project demonstrates a comprehensive approach to drone development, from CAD modelling to dynamic motion testing and computational fluid dynamics (CFD) analysis.

Software Used

This project was executed using the SolidWorks suite for both design and simulation tasks. Key software tools and modules included:

- **SolidWorks CAD** – Used for 3D modelling of all parts and creating the full assembly of the drone. It allowed precise geometry creation for components like the impeller blades, gears, frame, etc., and integration of these parts into a single assembly (Assembly.SLDASM).
- **SolidWorks Motion (Motion Study)** – Utilized to simulate the drone's mechanical motion. This module enabled applying motors and mates to replicate real-world movements (e.g., rotating blades via a motor and belt drive) and to animate the assembly for visual analysis.
- **SolidWorks Flow Simulation (CFD)** – Employed for aerodynamic analysis of the drone. This CFD tool was used to simulate airflow through and around the drone's propeller (impeller) and body, providing insights into pressure distribution, flow trajectories, and overall aerodynamic performance.

By using a single platform (SolidWorks) for design, motion, and fluid simulation, the project ensured a seamless workflow from initial concept through to detailed analysis.

Assembly Details

The drone's assembly (Assembly.SLDASM) is composed of several custom-designed components that fit together to form the final product. Each part was modelled individually in SolidWorks and then brought into the assembly with appropriate mates to define their relative positions and allowed movement.

- **Impeller Blades** (1-Impeller Blades.SLDPRT): A set of aerodynamic blades that function as the drone's propeller system. These blades are designed to impel air and generate lift. In the assembly, the impeller blades attach to a central hub and are driven by the motor via a gear or belt mechanism. They are critical for thrust and were a focus for aerodynamic optimization.

- **Arm Gear** (2-Arm Gear.SLDPRT): A gear component connected to the arm or rotor hub that interfaces with the motor's drive system. The arm gear transmits power from the motor (or a central gear) to the impeller blades. This ensures that when the motor spins, the arm gear (and attached impeller) rotates accordingly. Proper meshing and alignment of this gear in the assembly were important for smooth power transmission.

- **Gearing System** (3-Gearing.SLDPRT): The main driving gear or pulley mechanism in the drone. This part is mounted to the **Main Structure** and is driven directly by the motor. A belt or gear train connects the Gearing System to the Arm Gear, establishing the **motor-to-propeller connection**. The gearing ensures the impeller blades spin at the desired RPM and may also provide a gear ratio if needed to adjust torque/speed.

- **Main Structure** (5-Main Structure.SLDPRT): The central frame or body of the drone. It houses the motor and supports the arms and other components. The Main Structure is the backbone of the assembly, providing mounting points for the impeller mechanism, gears, legs, and camera. It was designed for structural integrity and minimal aerodynamic drag. All other parts are assembled onto this structure with appropriate mates (fixing its position as the base of the assembly).

- **Legs** (4-Legs.SLDPRT): Four landing legs attached to the underside of the Main Structure. These legs are positioned to stabilize the drone during takeoff and landing, keeping the body and propellers elevated off the ground. They were integrated into the assembly ensuring symmetric placement and enough clearance for the rotating impeller. Sturdy leg design contributes to overall stability when the drone is stationary on a surface.

- **Camera** (6-Camera.SLDPRT): A camera module mounted on the drone (typically at the front or bottom of the Main Structure). The camera is included to simulate payload or sensing capability for the drone. In the assembly, the Camera part is mated to the frame such that it remains fixed relative to the main body (or possibly on a gimbal, though here likely static). Its inclusion demonstrates how additional equipment can be integrated without interfering with aerodynamic performance or stability.

Each component was precisely integrated into the Assembly (Assembly.SLDASM) using appropriate mates—concentric for shafts, coincident for mounts, and gear/belt mates for the drive. The motor drives the gearing system, which spins the impeller blades via the arm gear, while the legs support the structure and the camera is mounted onboard. This realistic assembly formed the basis for both motion and CFD analyses.

Motion Study (Dynamic Simulation)

A **motion study** was conducted on the assembled drone to simulate its mechanical operation and to test the stability of moving parts. Using SolidWorks Motion, the team applied a motor and defined mates to replicate the real-life movement of the drone's propulsion system. The primary focus was on the rotation of the impeller blades and how the drive components interacted during operation.

In the motion analysis setup, a **motor** was configured to rotate the main drive gear at **100 RPM**. This motor was virtually attached to the Gearing System in the assembly. Through either a **belt drive mate** or a **gear mate** in SolidWorks, the rotation from the motor-driven gear was transmitted to the **Arm Gear**, and thereby to the **Impeller Blades**. This simulates the belt/chain mechanism connecting the motor to the propeller in the actual drone design. As the motor turns the main gear, the belt mate ensures the impeller spins at the same rate (or a calculated rate if a gear ratio was applied). The motion study verified that the mechanical linkage (motor → gearing → arm gear → impeller) functioned smoothly without any misalignment or slippage in the virtual assembly.

Additionally, the motion study incorporated **angle controls** to test and demonstrate any articulating features of the drone. For instance, if the drone's design included swivelling arms or a tilting camera mount, those would be driven by angular mates in the animation. In this project, the main motion was the rotation of the impeller, but the simulation could also capture slight movements or vibrations of other components (e.g., the reaction of the Main Structure and Legs to the spinning mass). By controlling the rotation angle over time (0–360° continuously at 100 RPM) and any other relevant joint angles, a realistic animation was produced.

The result of the motion study was an **animated sequence** (as captured in the provided **Motion.mp4** video) showing the drone's propeller spinning and the drive system in action. In the animation, the impeller blades rotate at a steady speed, driven by the invisible motor, while the rest of the drone remains stationary and stable. This visual proof-of-concept demonstrates that the design's moving parts work harmoniously: the gears mesh correctly, the belt (if used) stays taut, and there are no collisions between the spinning blades and the drone's body or legs. The stability of the drone during this operation is evident – the drone does not wobble in the simulation, indicating a well-balanced design. Overall, the motion study confirms the **mechanical feasibility and stability** of the drone's design under dynamic conditions, providing confidence before proceeding to aerodynamic testing.

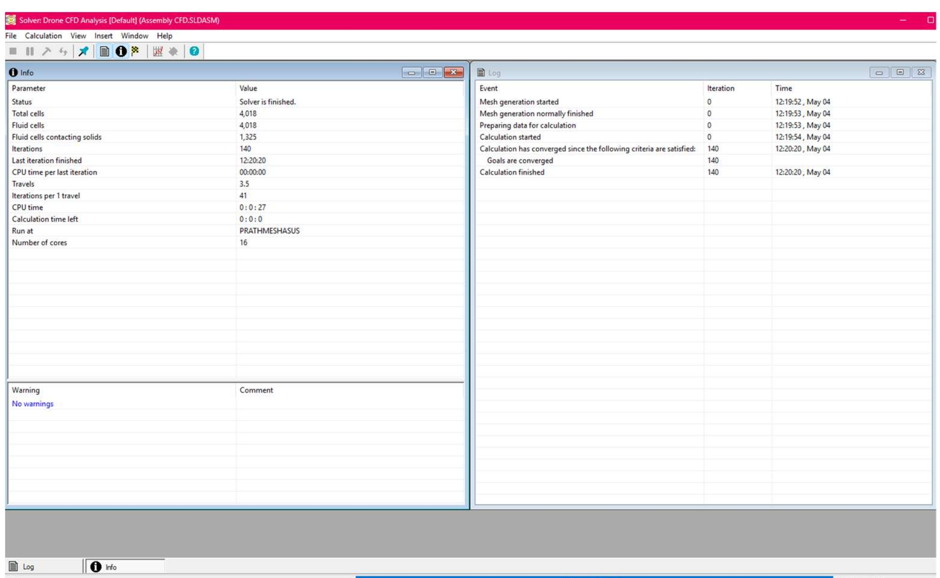
CFD Analysis (Aerodynamic Performance)

After validating the mechanical operation, a **Computational Fluid Dynamics (CFD)** analysis was performed to study the drone's aerodynamic behaviour using SolidWorks Flow Simulation. The CFD study aimed to optimize the aerodynamics of the design by examining airflow patterns, pressure distribution, and potential forces on the drone while the impeller is running. This step addresses the aerodynamic optimization objective and provides insight into how efficiently the drone would generate thrust and maintain stable flight.

For the simulation, air was used as the working fluid, and the environment was set up to reflect a plausible operating condition (e.g., the drone in hovering flight with air flow induced by the propeller). The impeller (propeller blades) was defined to rotate at **150 RPM** in the CFD model – a slightly higher speed than in the motion study, to intensify airflow for analysis. Appropriate boundary conditions were applied, likely including an inlet or opening around the impeller to allow air to be drawn in, and outlets for air to exit (for an open environment, the software automatically creates an external flow domain around the drone). The drone's surfaces (blades, body, legs) were set as solid boundaries that interact with the fluid. A rotating region or moving reference frame approach was used in SolidWorks Flow Simulation to model the effect of the spinning impeller on the air without having to physically rotate the mesh.

Mesh and Solver Details

The flow domain was discretized into a computational mesh with **4,018 cells** in total. Notably, **1,325 of these cells** directly contact solid surfaces (the drone's parts), meaning the mesh was refined around the geometry to capture boundary layer effects and accurate airflow near surfaces. Such mesh resolution is sufficient for a preliminary analysis, ensuring that key areas like the impeller blade surfaces and the edges of the Main Structure have adequate cell density. The CFD solver was run for **140 iterations**. During the simulation, monitors (goals) were likely set for quantities of interest such as pressure or velocity at certain points, or overall thrust force, to assess convergence. The solver log indicated that the **calculation converged** successfully: the goals met the specified criteria, and no further significant changes were occurring in the solution. There were **no warnings** throughout the run, signifying a stable and well-posed simulation case.



CFD solver output summary for the drone analysis. The CFD solver finished the computation in just **27 seconds** of CPU time, taking advantage of a 16-core machine for parallel processing. This rapid solution time (under half a minute) reflects the relatively modest mesh size and the efficiency of the SolidWorks Flow solver for this scenario. The solver output (shown in Figure 2) confirms the key results of the run: 4,018 total fluid cells were generated in the mesh, and the final iteration count reached 140 with all convergence goals satisfied. The log excerpt in the figure shows events like mesh generation, solution start, and that *"Calculation has converged since the following criteria are satisfied: Goals are converged"*, followed by *"Calculation finished"* with no errors or warnings. This successful convergence means the results of the simulation are reliable and can be used to interpret the drone's aerodynamic characteristics.

Flow Trajectories and Results

With a converged solution, flow visualization tools were used to interpret the data. *Flow trajectories* (streamlines) were generated in the **Flow Simulation.mp4** to show how air moves through and around the drone. These streamlines illustrate the path of air particles as they are pulled by the rotating impeller. The video and analysis indicate that air is drawn from the surroundings into the impeller region and expelled downwards (creating thrust). The streamlines start above the drone, getting sucked into the impeller disk, and then accelerate as they pass through the blades, exiting below the drone. This creates a focused downward airflow. The trajectories are relatively smooth and symmetric about the impeller's axis, suggesting that the propeller design is balanced and produces a uniform flow pattern. There is no sign of severe turbulence or recirculation in the immediate vicinity, which implies that the drone's body and legs are not obstructing the flow significantly – an indicator of a good aerodynamic design.

In addition to streamlines, *surface plots* were examined on the drone's components to observe pressure distribution. The CFD results likely include a pressure contour plot over the surfaces of the impeller blades and the main body. From the simulation, one can infer that the **pressure is lower on the upper side of the impeller blades** (suction side) as the air is drawn in, and higher on the underside where the air is being pushed downward. This pressure difference across the blades is what generates lift. On the Main Structure and other parts of the drone, the pressure plot shows mostly ambient pressure, with slight increases on surfaces directly in the path of the airflow and slight decreases in wakes or shadowed regions behind components. For example, the top of the drone near the impeller inlet might see a small low-pressure region, and the underside where the fast-moving air flows out may see a pressure drop (Bernoulli effect), while the legs might have higher pressure on their wind-facing sides. These surface pressure visuals confirm that the impeller is effectively creating the intended thrust, and that the drone's structure can handle the aerodynamic loads (since no extreme pressure hotspots are observed). The **flow simulation** overall demonstrates that the drone's design is aerodynamically capable: air moves through the system as expected, and the distribution of pressure and flow velocities is in line with stable hovering flight.

Finally, the CFD analysis provides quantitative data such as the thrust force generated by the impeller at 150 RPM, and the drag on each component. While those specific numbers are not detailed in this report, the converged simulation would allow extraction of such values. This data can guide further design optimizations – for instance, tweaking the impeller blade shape for more thrust or adjusting the frame to reduce drag. In summary, the CFD study validated the aerodynamic performance of the drone and identified no major issues, aligning with the project's goal of aerodynamic optimization and stable flight dynamics.

Comparison of Motion and CFD Conditions

It is noteworthy that the **rotational speed of the impeller differs** between the motion study and the CFD analysis. The **motion study** (mechanism animation) was conducted with the motor at **100 RPM**, whereas the **CFD simulation** used a speed of **150 RPM** for the impeller. This difference means that the airflow conditions analyzed in CFD correspond to a faster-spinning propeller than what was shown in the kinematic animation. In practical terms, a higher propeller RPM in the CFD would generate greater airflow, higher pressure differences, and consequently more lift force than the 100 RPM case. The design was tested under this more intense condition (150 RPM) perhaps to ensure it can handle a range of operating speeds or to examine its performance at an expected real-life throttle setting (many drones operate propellers at several thousand RPM; 150 RPM in CFD could have been a scaled-down test or an artifact of the simulation setup to keep flow speeds moderate).

For the purpose of the project report, this RPM discrepancy highlights a few points. First, it shows **flexibility in simulation parameters** – the designer explored multiple scenarios (one for mechanical motion, one for aerodynamics). Second, it underlines the importance of correlating mechanical and aerodynamic analyses: if the motion study and CFD study are to be directly compared, they should ideally use the same operating conditions. In future iterations, one might run the motion study at 150 RPM as well, to better visualize the same intensity of operation as the CFD or conversely run a CFD analysis at 100 RPM for direct comparison of results. However, even with the difference, the qualitative outcomes remain consistent – the drone operates smoothly at 100 RPM (mechanically), and aerodynamically it performs well at 150 RPM. There were no indications that the structure or stability would falter at the higher speed, which is a positive sign. In fact, testing at a higher RPM in CFD provides a safety margin, suggesting that at the lower 100 RPM (or intermediate speeds) the drone would have even less aerodynamic stress. Overall, this comparison assures that the drone's design is robust across different speeds, and it informs the designer about how scaling the motor speed might impact lift and stability.