Aerodynamic Analysis of a High-Powered, Multi-Stage Rocket

Parker T. Blenk[1]
University of Georgia, Athens, GA, 30602, United States

Jason S. Rupram[2]
University of Georgia, Athens, GA, 30602, United States

The Intercollegiate Rocket Engineering Competition (IREC) is one of the world's largest student-built rocket competitions, connecting students from across the United States and around the globe. IREC is built upon motivating and encouraging students to pursue science, technology, engineering, and math (STEM) based careers, allowing students to develop real-world engineering problems through designing, building, and flying multistage rockets. The University of Georgia (UGA) has designed, built, and launched IREC-ready rockets for three years with different design focuses. This year, the primary focus has been on the integration and deployment of air brakes. IREC challenges teams to reach a target altitude of 10,000 ft, and as a result, teamwork and engineering innovation are required to achieve precise apogee control, ultimately determining the competition scoring. A CFD model, using ANSYS Fluent with the k-omega SST turbulence model, was developed to analyze the effectiveness of air brakes in controlling the rocket's velocity, trajectory, and apogee during flight by accurately capturing high-speed flow characteristics. This CFD-based analysis establishes a reliable framework for integrating air brakes and ensuring an optimal balance between aerodynamic performance and flight control for the 2025 UGA IREC rocket.

Nomenclature

AR = aspect ratio
C_D = drag coefficient
$Mach$ = speed of sound
v = velocity

I. Introduction

Rocketry Associations such as Experimental Sounding Rocket Association (ESRA) hosts the Intercollegiate Rocket Engineering Competition (IREC) which challenges teams to create an innovative design that balances performance, stability, and safety. One of the scoring metrics in IREC is apogee; how closely a team's rocket can get to an altitude of 10,000 feet. Teams receive higher scores based on accuracy to this apogee, making it important to achieve the target altitude consistently. Rocketry airbrakes have been used by schools and other IREC competition teams in the same advantage this paper hopes to achieve [1] [2]. They are essential in decelerating the rocket before reaching the targeted apogee by increasing the total drag exerted onto the rocket. The University of Georgia has developed a rocket to compete with, which utilizes this airbrake subsystem that is critical in controlling the rocket's flight profile. This subsystem features a 3-petal design that will deploy linearly and radially outwards under the power of a single motor. Overall, the integration of high-fidelity CFD analysis using ANSYS establishes a framework essential for optimizing the airbrake system and ensuring that UGA's rocket meets the performance requirements of IREC.

II. Methodologies

---

[1] Graduate Student, College of Engineering, AIAA Student Member (ID: 1572880)
[2] Graduate Student, College of Engineering, AIAA Student Member (ID: 1785154)

The base model was developed using OnShape, a free open-source computer automated design (CAD) software that allows group collaboration. This CAD software works ideally for group projects such as this as it allows individual subsystems and teams to combine their sub-assemblies into a main assembly. This is evident on the CAD of the rocket when implementing the airbrakes to the airframe. The detailed model of the airbrakes allows for much further understanding of the subsystem. The airflow behind the airbrakes is highly chaotic and turbulent in nature due to the flow separation and backflow. This inherent turbulence and chaos make it difficult to accurately predict using low-fidelity computational fluid dynamics (CFD) software. OpenRocket is a widely adopted rocket simulation tool that is useful for preliminary and low-fidelity flight trajectories. Flight trajectories include metrics such as max apogee, C_D, max velocity, etc. Additionally, OpenRocket uses a 6 Degree of Freedom Runge-Kutta-4 method which provides accurate correlations to real-world flight dynamics. This method is great for its high accuracy in low computational time but can often struggle with highly oscillatory/chaotic systems, like the one described in this paper. The air brakes have a major effect on airflow, resulting in a substantial increase in turbulent flow as well as flow separation, which as mentioned before is something that this method struggles with. As such, the need for more accurate analysis like high-fidelity simulations is important in determining the overall control of the rocket to reach the 10,000 ft criteria. To conduct these high-fidelity simulations, ANSYS Fluent was considered with the K-ω Reynolds-Averaged Navier-Stokes (RANS) turbulence model, which will help capture the turbulent flow structures behind the airbrakes. The K-ω RANS model is effective in resolving near-wall effects and complex turbulent flow features characteristic of chaotic regions, thus providing a more realistic prediction of the aerodynamic forces acting on the rocket.

To appropriately model this case study, a few assumptions were made to reduce computational resources and time. The assumptions are as follows.

1) Air is assumed to be compressible ($M_\infty > 0.3$)
2) Air is assumed to be viscid
3) Assumed to be a steady state simulation as the rocket will not have any additional thrust or cross-winds
4) Turbulent flow
5) Geometry is symmetrical therefore airflow is symmetrical

Additionally, the boundary conditions include a pressure far-field, no-slip condition, and by extension viscous flow. In this simulation, the walls of the rocket will act as a no-slip boundary condition. This means that the velocity of the fluid that touches the rocket, in this case, air will have zero velocity. An initial condition of these simulations is the velocity at the inlet, this is assumed to be Mach 0.8 as this is what OpenRocket claims. Air properties are to be derived using ideal gas, so density varies as the flow develops.

For the retracted air brakes simulation, the nose cone, tail cone, antennas, and transition regions were meshed with a local mesh size of 0.005 m to ensure that critical geometric features were well-resolved. Due to the fins' significant influence on aerodynamic characteristics, a target mesh size of 0.0025 m was applied to the fins. The resulting surface mesh incorporates these localized sizing parameters to capture the flow features over the rocket's surface. To further enhance mesh quality, 10 inflation layers were added with a growth rate of 1.2 and a transition ratio of 0.272, ensuring a smooth resolution of the boundary layers adjacent to the rocket's surfaces. Overall, the mesh quality and cell count are deemed sufficient to produce a high-fidelity simulation while maintaining reasonable convergence times. Similarly, for the full deployed airbrakes model, all mesh sizing and features stay consistent, however, an additional sizing of 0.005 m was added to the face and edges of the airbrakes, which will enhance the model's capability to capture the intricate flow behavior where the flow will be disrupted. The volume mesh incorporated polyhedral cells, with both models resulting in a minimum orthogonal quality of 0.25 and over 3.6×10^6 cells.

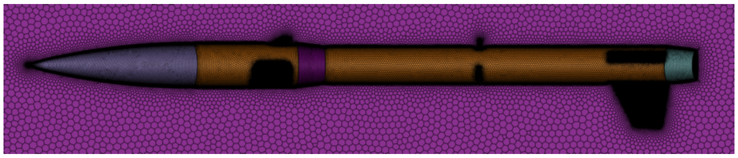


Fig. 1 Polyhedral Mesh

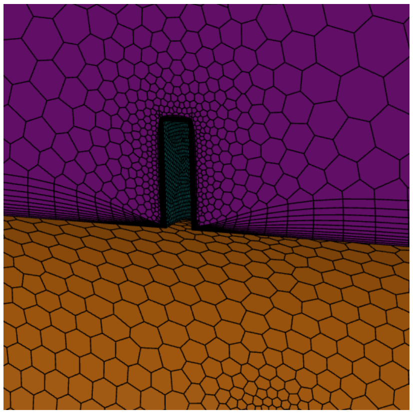


Fig. 2 Mesh Refinement Around AB

III. Design Details

The initial rocket configuration, with the airbrakes retracted, is the first concept focused on. To enable a good mesh quality for CFD, the airframe is modeled with a smooth exterior, such as assuming all mating surfaces of the rocket have no lip, screw holes are filled, and the cavity of the airbrake is filled. The rocket features a nose cone with a 157 mm diameter design in a 5 to 1 von Karman shape. The rocket's airframe is composed of two distinct sections. The first section features a 6-inch diameter and spans 1 foot in length, while the second section has a 5-inch diameter and extends 4.5 feet. This multi-section configuration is designed to balance structural integrity with aerodynamic efficiency, ensuring that the rocket meets the performance targets. This nose cone profile was used due to its stable and efficient aerodynamic properties. The stability of the rocket is also due to the three fins on the aft section of the 5-inch airframe. A critical design parameter for the fins is its span, defined as the measurement from the fin root chord to the tip chord, which is set at 162.74 mm. Another notable surface that induces a significant drag penalty on the rocket is the camera bump and antenna shrouds. All these factors will play into the overall performance and aerodynamic stability of UGA's rocket. The second design iteration introduces a configuration in which the airbrakes are extruded outward from the airframe. The three-airbrake petals which sit 14 inches behind the transition will extrude 1 inch radially out of the airframe. In this model, the small gap between the airframe and the petals will be filled in for simplicity during the meshing process of CFD.

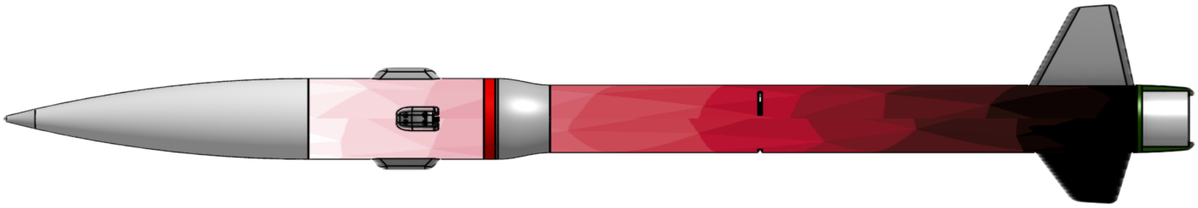
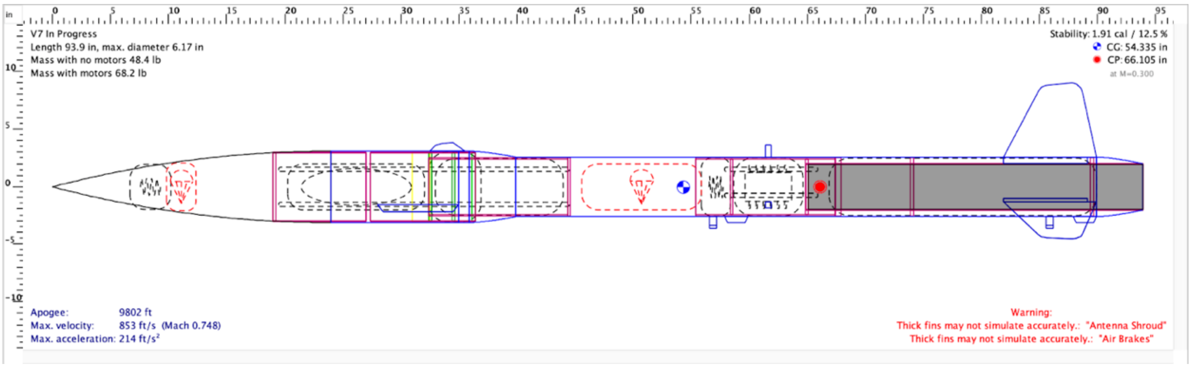


Fig. 3 CAD Model of IREC Rocket



V7 In Progress
Length 93.9 in, max. diameter 6.17 in
Mass with no motors 48.4 lb
Mass with motors 68.2 lb

Stability: 1.91 cal / 12.5 %
CG: 54.335 in
CP: 66.105 in
at M=0.300

Apogee: 9802 ft
Max. velocity: 853 ft/s (Mach 0.748)
Max. acceleration: 214 ft/s^2

Warning:
Thick fins may not simulate accurately.: "Antenna Shroud"
Thick fins may not simulate accurately.: "Air Brakes"

Fig. 4 OpenRocket Model of IREC Rocket

IV. Simulation Setup

For each rocket model, a Fairfield boundary condition was defined using a half-sphere with a radius of 15 m. To enhance the resolution in regions immediately around the rocket, a body of influence was introduced. The body of influence was created using a box, with dimensions of 6 × 1 × 1 m. This was created around the rocket to promote local mesh refinement. The increased cell density within this box allows for a more accurate capture of the flow areas immediately around the rocket where the aerodynamic behavior is most complex and significant. This approach ensures that the flow features, especially in the vicinity of the rocket, are resolved with greater precision without unnecessarily increasing the overall cell count of the entire computational domain. Named selections were then created for components of the rocket to better control meshing. For the base model, selections include the Fairfield, Nosecone, Airframe, Antennas, Transition, Tail Cone, and Fins. The rocket's symmetry was leveraged by simulating only half of the full geometry. For the Airbrakes model, a similar workflow was followed with the addition of an Airbrakes selection. The named selections include Fairfield, Nosecone, Airframe, Antennas, Transition, Tail Cone, Fins, and Airbrakes which allows for a high-fidelity analysis of the impact of the airbrakes on the aerodynamic profile of the rocket.

A. Retracted Airbrakes

Air was selected as the fluid flowing through the domain and then all initial and boundary conditions were inputted into the solver. Since the primary goal is to analyze the drag characteristics caused by the air brakes, a drag plot was created on the entirety of the rocket. This was done on the entirety of the rocket as this will eventually help predict the trajectory of the rocket. After initializing the system, 300 iterations were computed to monitor and check for convergence of the drag report and the scaled residuals: continuity, x-velocity, y-velocity, z-velocity, k, and omega. The system converges after 125 iterations when all residuals reach their final values and the plot flattens out to values ranging between $1\ x\ 10^{-2}$ and $1\ x\ 10^{-8}$, as seen in Fig 5**.** This convergence indicates confidence about the results as the error throughout the system is negligible. These periodic oscillations can arise from several factors, the biggest being unsteady flow. In this simulation, there is a lot of turbulence caused by the rocket which can cause cyclic behaviors in residuals.

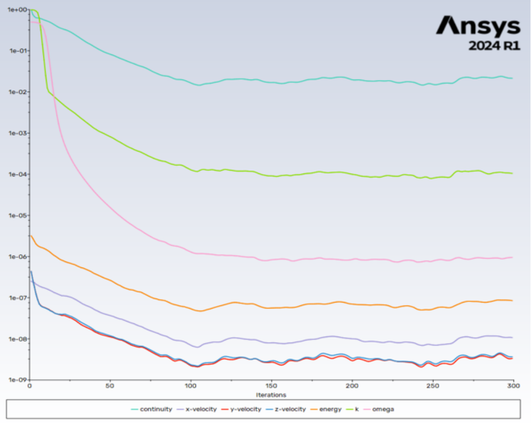


Fig. 5 ANSYS Simulation Residuals Retracted AB

B. Fully Deployed Airbrakes

After initializing the simulation system, 300 iterations were simulated to achieve convergence of the drag coefficient report and the scaled residuals for continuity, x-velocity, y-velocity, z-velocity, turbulence kinetic energy (k), and the specific dissipation rate (omega). Convergence was achieved after about 125 iterations, with all residuals stabilizing to values between 1×10^{-2} and 1×10^{-8} as depicted in Figure 6. This stabilization indicates that the numerical error within the system is marginal. Overall, the convergence behavior provides strong evidence of the simulation's accuracy and reliability.

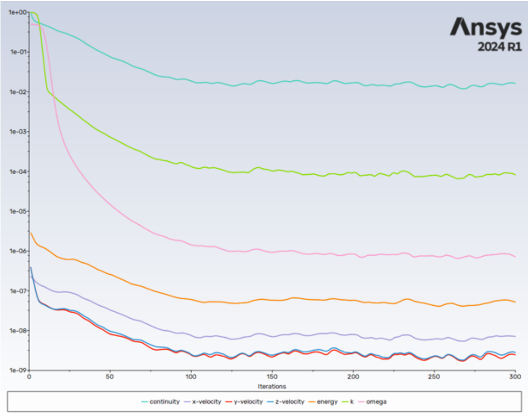


Fig. 6 ANSYS Simulation Residuals Full Deployment AB

V. Simulation Results

A. Retracted Airbrakes

When analyzing the airflow over the rocket, the first thing to look at is the pressure differential over the body. In Fig. 7, the high-pressure regions occur at the tip of the nose cone and the leading edges of the fins. The leading edge of the fins creates a stagnation region which also results in an increase in static pressure. There is also a low-pressure wake region behind the aft end. This turbulent low-pressure region can be seen in Figs. 7 & 8.

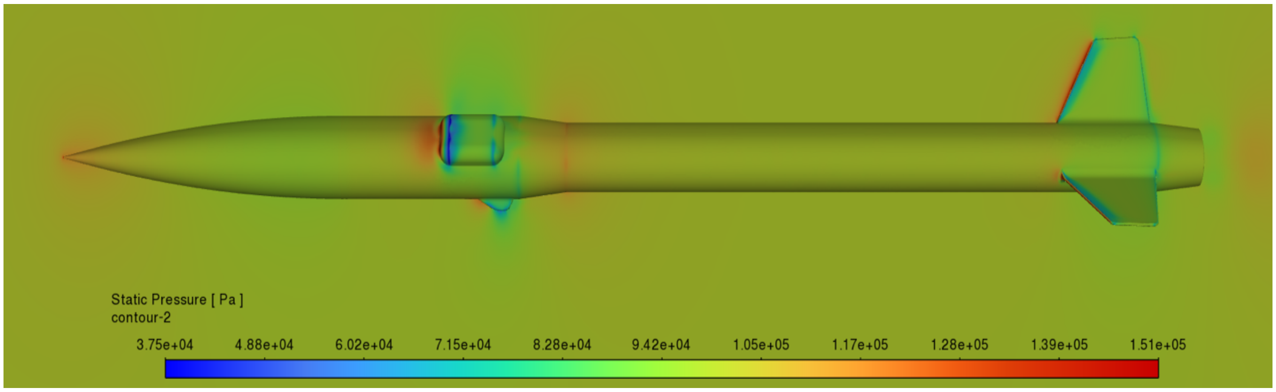


Fig. 7 Static Pressure Around Base Rocket

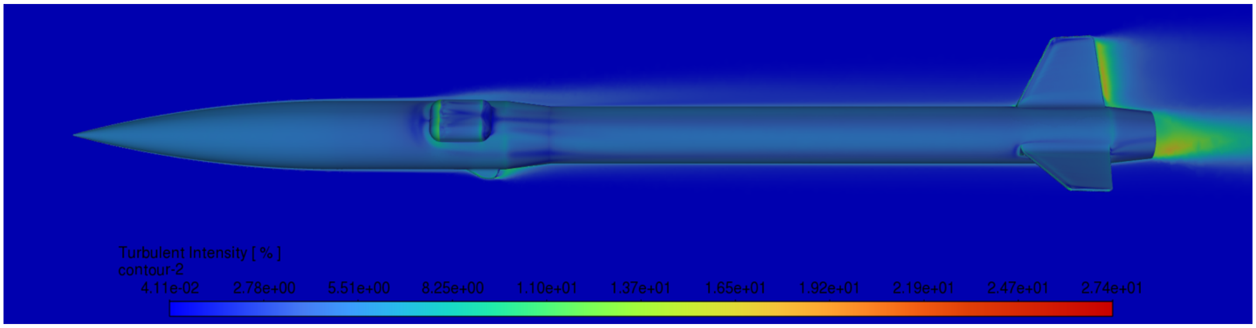


Fig. 8 Turbulence Intensity Around Base Rocket

B. Fully Deployed Airbrakes

In analyzing the airflow over the rocket, the initial focus is on the pressure differential across the rocket. As seen in Fig. 9, the frontal face displays high-pressure regions due to the formation of a stagnation zone, which is reflected via an increase in static pressure. A low-pressure region is also observed in the area aft of the air brakes. In Fig. 9 there is a clearly visible turbulent low-pressure wake. The velocity behavior behind the air brakes is particularly noteworthy. Additionally, Fig. 9 illustrates high-pressure regions at the tip of the nose cone and along the leading edges of the fins. These features are associated with significant airflow deceleration. The nose cone functions as a stagnation point, where the airflow slows considerably.

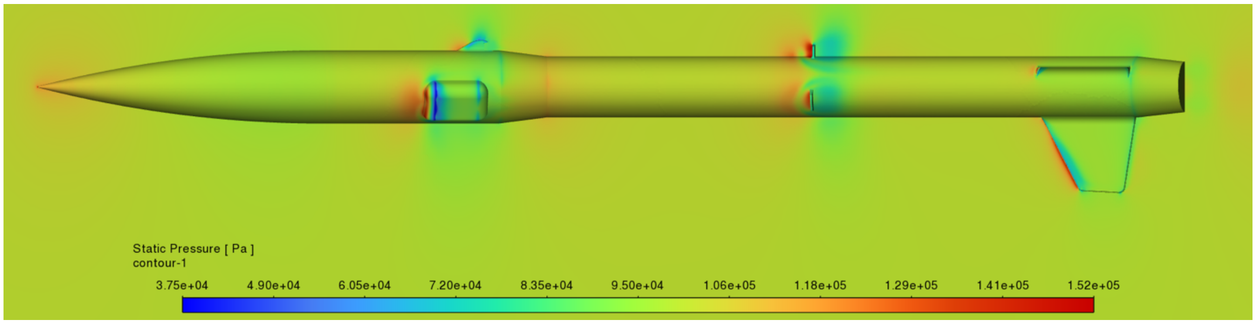


Fig. 9 Static Pressure Around Base Rocket

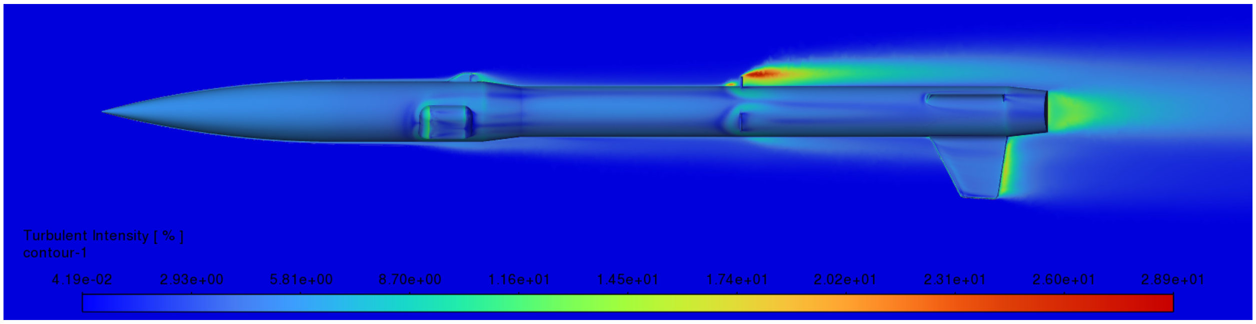


Fig. 10 Turbulence Intensity Around Airbrakes Rocket

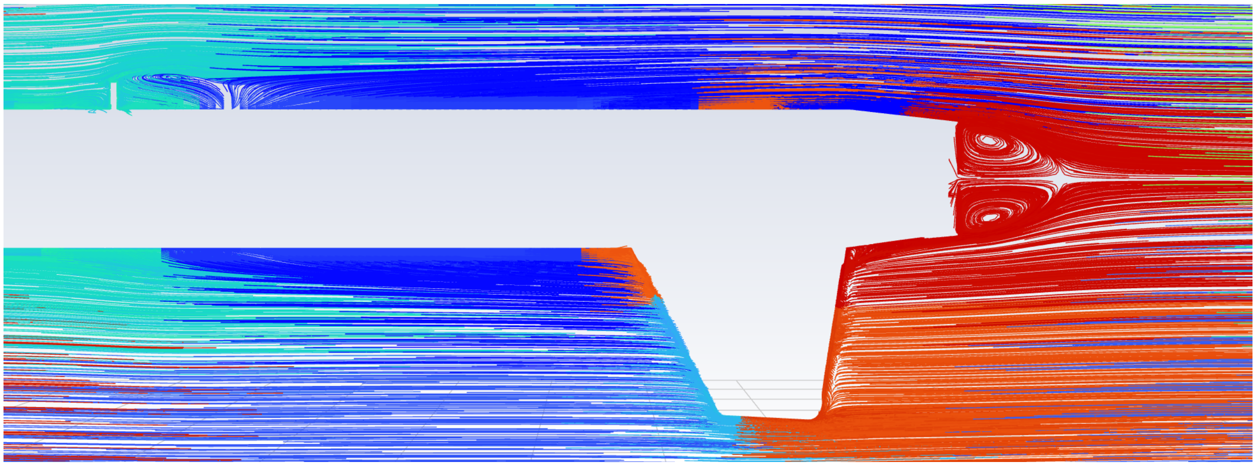


Fig. 11 Particle Pathlines around Airbrakes Rocket

VI. Simulation Results

When determining the effectiveness of airbrakes in decelerating the rocket to reach the target apogee, this paper assessed the total increase in drag on the rocket. Looking at Table 1, it can be seen that when the airbrakes are fully deployed, the total C_D increased by roughly 0.12. This in large is not a drastic increase in total drag, which could be attributed to two things. First, due to the aluminum transition frame designed into the rocket to connect the 6-inch airframe to the 5-inch airframe, a barrier was created that blocked radio signals from reaching to the ground, in what is commonly known as the Faraday Cage effect (https://www.gamry.com/application-notes/instrumentation/faraday-cage/). This resulted in the UGA IREC team moving the antennas outside the frame into two adjacent shrouds. In order to not impact the airflow around the fins, ultimately making the rocket unstable, two airbrake petals were placed directly in line with the antenna shrouds. As seen in Fig. 7 & 9, these shrouds have a huge impact on airflow around the rocket, causing a large increase in static pressure as well as flow separation. The flow does not have enough time to attach to the rocket before contacting with the airbrakes, which decreases the effectiveness of the airbrakes. Lastly, due to manufacturing constraints, the total area of each airbrake petal was reduced to securely slot inside the five-inch airframe, which resulted in a decreased C_D.

Additionally, when looking at the values calculated by OpenRocket and ANSYS, the inaccuracy of OpenRocket to fully capture highly turbulent flows dramatically shows. For both retracted and fully deployed airbrakes, OpenRocket overestimates the value of C_D by almost 0.3.

Table 1. Comparison of Drag Coefficients on IREC Rocket

Analytical Model	OpenRocket	ANSYS	Percent Difference
Retracted AB C_D	0.513	0.275	60.41
Fully Deployed AB C_D	0.770	0.395	64.38

As seen in fig. 7 & 9 there are high-pressure regions at the nose. This is indicative of a stagnation point, in which the air has a lower velocity which increases static pressure at the cost of dynamic pressure. In those same images, there are further high-pressure regions at the front of the antennas and fins. In Fig. 7 & 9 there are also low-pressure regions on the fillets of the fins, antennas, and tail cone due to local airspeeds accelerating thus causing a drop in static pressure but an increase in dynamic pressure. This local increase in airspeed over the fins and antennas are caused by the curved nature of the fillets/bevels on the geometry. The drop in pressure can also be explained by flow separation as illustrated in Figs 8 & 10. These levels of adverse pressure gradients can induce flow separation, thus leading to higher drag values and unsteady effects. Turbulent intensity (TI) is a percentage measurement of how unsteady/chaotic the flow is. As seen in Figs. 8 & 10, there is low TI percentage along the nose cone until the antennas. This initial region with little turbulence indicates the boundary layer is still stable. Once the airflow reaches the antennas it becomes less stable and grows in intensity due to the relatively sharp changes in the directions of the antennas which causes vortices as seen in Fig. 11. Just past the airframe, for the retracted model, there is some slight flow separation and instability before it has a chance to interact with the surrounding freestream and mixes and eventually reattaches and stabilizes before reaching the fins. Once the airflow reaches the fins the flow remains stable until the end of the fins and tail cone. Immediately after these two ends, the air has no surface to attach to which causes an immense amount of flow separation, vortices, and recirculation. This behavior is represented by the large amount of turbulence in the TI contour. For the air brakes deployed model the flow does not have enough time to restabilize as seen in Fig. 10. The lack of time to restabilize hinders the theoretical max performance of the airbrakes as turbulent air does not create predictable pressure contours thus creating oscillating drag values as seen in fig. 6. Much like the airflow after the antennas there is even greater turbulence and fails to restabilize before it reaches the fins. The max pressure on the fins is less than the air brakes retracted model but not by much because the fins are clocked 45 degrees off the airbrakes which helps give it cleaner and slightly cleaner air to interact with compared to if they were in line with each other. The growth in turbulence does impede slightly with stability as vortices can oscillate, however, the rocket should be stable enough with full airbrake deployment to maintain a predictable and consistent flight portfolio. The particle path lines shown in Fig. 11 gives insight into flow direction, flow separation, vortices, and recirculation. The key behavior capture is the flow separation on the 5-inch section of the airframe for and the recirculation/backflow immediately behind the airbrakes and tail cone. Because there is a blunt end to the geometry the flow separates and creates a multitude of vortices in the wake of the rocket. Overall, the behavior witnessed in CFD is feasible and correlates to previous aerodynamic analysis studies.

VII. Conclusion

The airbrake system utilized by the University of Georgia's IREC team is essential in reaching the target apogee of 10,000 feet and maximizing the competition score. High-fidelity CFD analysis plays an important role in determining the overall effectiveness of each airbrake petal by precisely predicting C_D, flow separation, and turbulence intensity levels across the entire rocket. This study not only emphasized the unpredictable and turbulent flow at high speeds but also provided valuable insights into future design recommendations.

By leveraging mesh sizing refinements and incorporating the k-omega SST turbulence model, the rapid changes in fluid properties over key components of the rocket were captured, highlighting the large flow separations and static pressure around the airbrakes. Additionally, this CFD study helped the UGA IREC team understand the importance of airbrake placement on the rocket. It detailed the flow separation from the antenna shroud to the airbrake petal, ultimately decreasing the effectiveness of the airbrakes. These results will ultimately help the rocket maintain stable flight while also controlling its maximum height.

References

[1] Eyre, E and Ding Nicole. "Air Brakes", MIT Rocket Team, 2024. https://wikis.mit.edu/confluence/display/RocketTeam/Air+Brakes

[2] Jack, S. "The Evaluation of Various Controller Architectures for an Air Brake on a High-Powered Model Rocket", AIAA, 2024. https://doi.org/10.2514/6.2024-83924