

# Optimal Design for Improving Drag and Cooling Performance of KSAE E-Mobility through CFD Analysis

G. H. An, C. S. LEE

Handong Global University Mechanical Control Engineering



## Introduction

- This project aims to reduce drag and enhance the cooling efficiency of the electronic circuits in an electric vehicle designed for the KSAE (Korea Society of Automotive Engineers) university student competition through CFD (Computational Fluid Dynamics) analysis. The existing vehicle design has a structure that suffers from high drag and ineffective cooling of the electronic circuits. To address these issues, we utilized Ansys Fluent to propose a model that minimizes drag while improving the cooling of the electronic circuits. Additionally, we aim to verify the reliability of the CFD analysis results through wind tunnel experiments. The key factors determining the drag force( $F_D$ ) of a vehicle are the projected area in the direction of fluid flow( $A$ ), the velocity of the fluid ( $v^2$ ), and the drag coefficient ( $C_D$ ) due to the shape, as shown in Equation (1).

$$F_D = \frac{1}{2} \rho v^2 C_D A \quad (1)$$

- Therefore, to minimize the flow separation phenomenon (Fig 1), which causes drag, the vehicle was designed with a streamlined shape to reduce the drag coefficient ( $C_D$ ) and to decrease the projected area ( $A$ ), aiming to minimize the drag force.

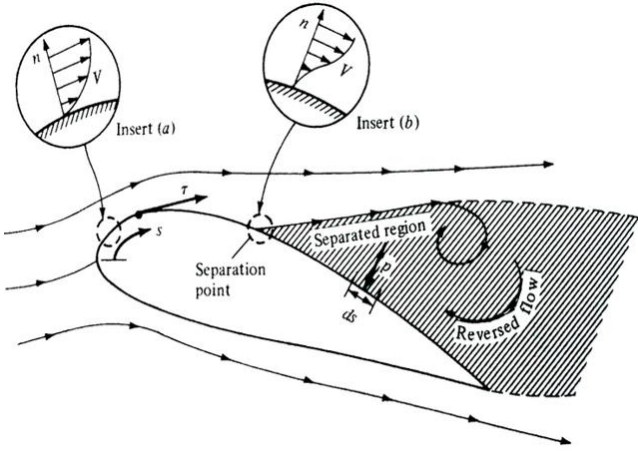


Fig 1. Separation Phenomenon

## Model Description

### Advanced 1 (Frame Unmodified)

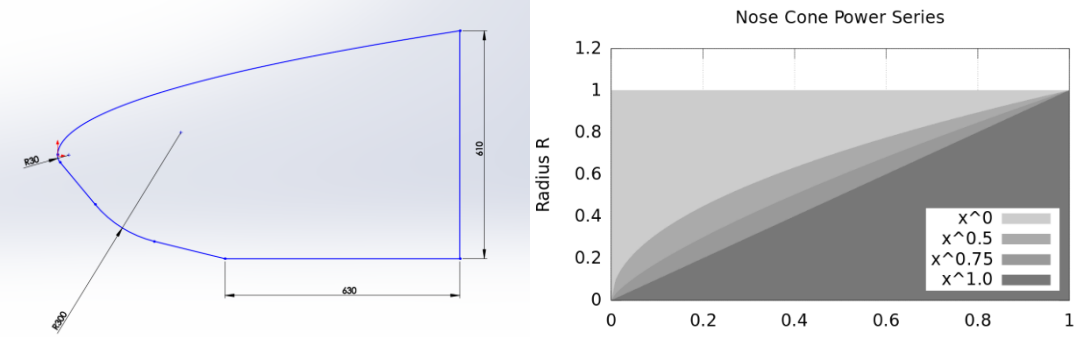


Fig 2. Power 0.5 Nose Cone Design

- The model was inspired by the Power 0.5 model of Nose Cone Design (1), and the front part of the vehicle was designed with a streamlined shape.

$$y = R \left( \frac{x}{L} \right)^2 \quad (1)$$

### Advanced 2 (Frame Modified)

Table 1. Lift-to-Drag Ratio and Drag Coefficient of NACA Airfoil at Different Angles of Attack

$\alpha^\circ$	NACA 4412		NACA 2412		NACA 6409	
	$\frac{C_L}{C_D}$	$C_D$	$\frac{C_L}{C_D}$	$C_D$	$\frac{C_L}{C_D}$	$C_D$
0	69.63	0.00682	42.5	0.00560	99.15	0.00704
-1	52.57	0.00697	21.58	0.00604	83.95	0.00701
-2	35.56	0.00712	3.27	0.00648	66.85	0.00714
-3	19.25	0.00737	-12.80	0.00694	48.86	0.00749
-4	3.87	0.00777	-26.42	0.00751	30.35	0.00836
-5	-9.82	0.00830	-37.48	0.00820	14.68	0.00964
-6	-21.17	0.00910	-46.07	0.00901	2.64	0.01125
-7	-30.24	0.01001	-52.17	0.01001	-	-

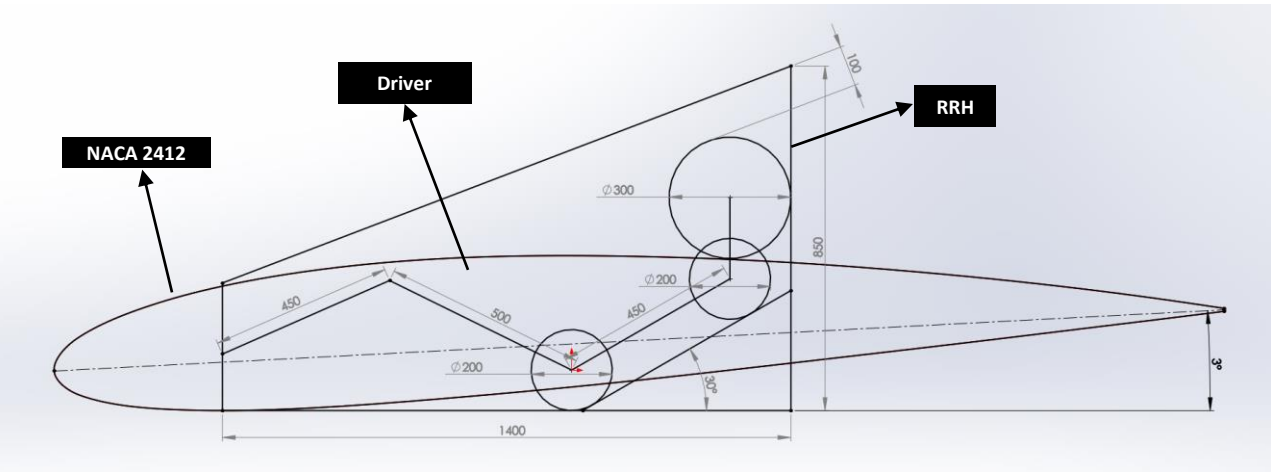


Fig 6. Advanced 2 Model Sketch (NACA 2412)

- The design is based on the NACA 2412 airfoil (at AOA,  $\alpha = -3^\circ$ ), which has a low lift-to-drag ratio, and the smallest drag coefficient ( $C_D$ ).
- The driver's seating position was changed from a sitting posture to a 30° reclining posture, reducing the height of the RRH by 350mm.

## Model Comparison

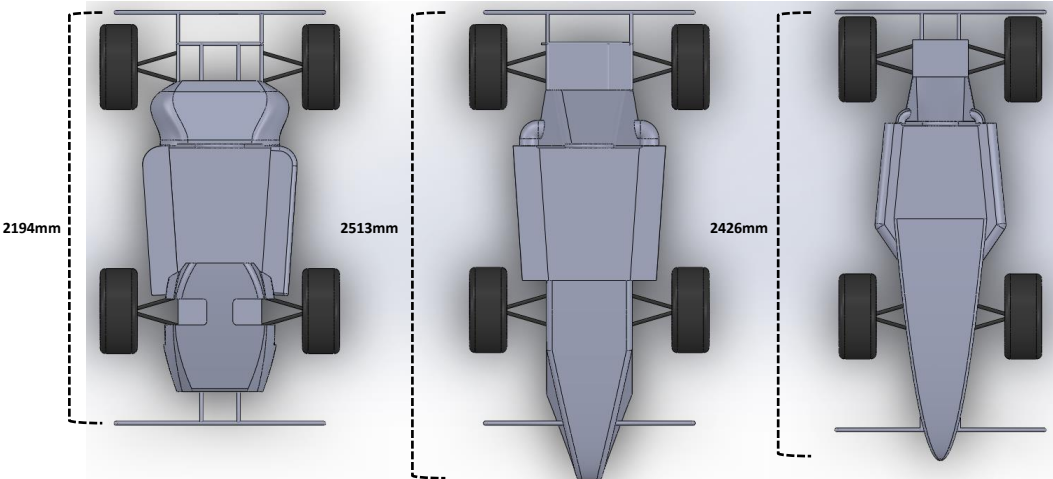


Fig 7. Original, Advanced 1, Advanced 2 Models (Top View)

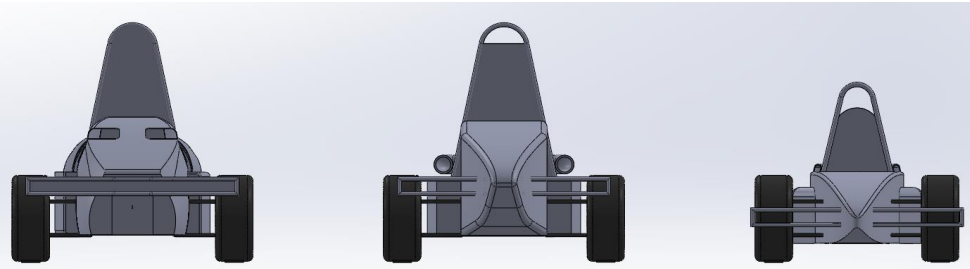


Fig 8. Original, Advanced 1, Advanced 2 Models (Front View)

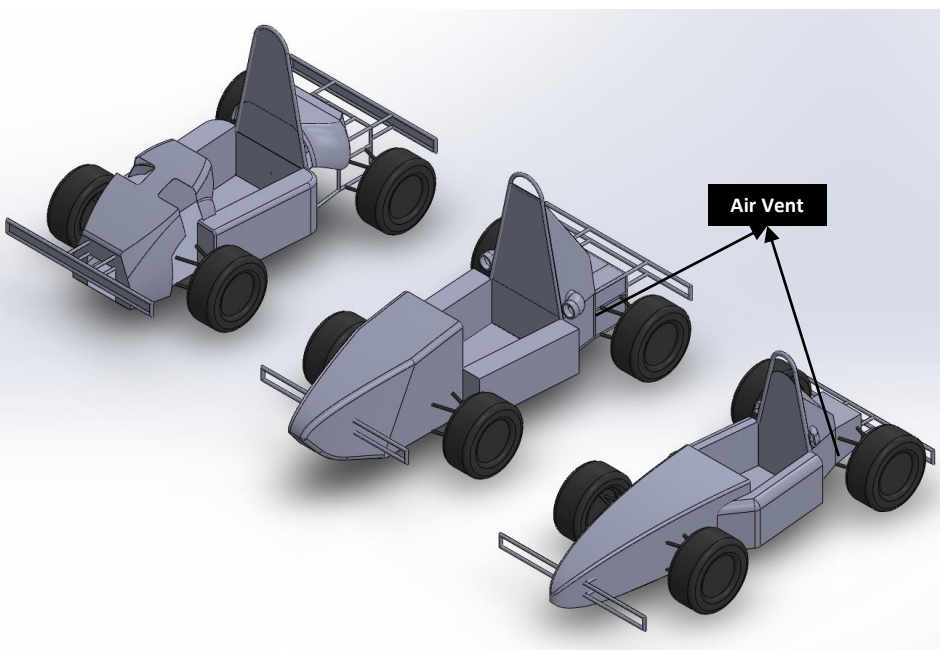


Fig 9. Original, Advanced 1, Advanced 2 Models (Left to Right)

Table 2. Projected Area of Models

	Original	Advanced 1	Advanced 2
$A(m^2)$	0.40	0.385	0.3

## Meshing Method

Table 3. Mesh Conditions

Region	Type	Size (mm)
BOI-nearfield	Body of Influence	40
BOI-farfield	Body of Influence	90
Bumper and Rods	Curvature	Min:4 / Max:5
Cowl	Curvature	Min:5 / Max:30
Wheel	Curvature	Min:3.5 / Max: 5
Boundary Layer	Last-ratio	10 layers / Ratio: 0.2
Surface	Curvature & Proximity	Min: 0.5 / Max: 256
Volume	Poly-hexcore	Min: 0.5 / Max: 512

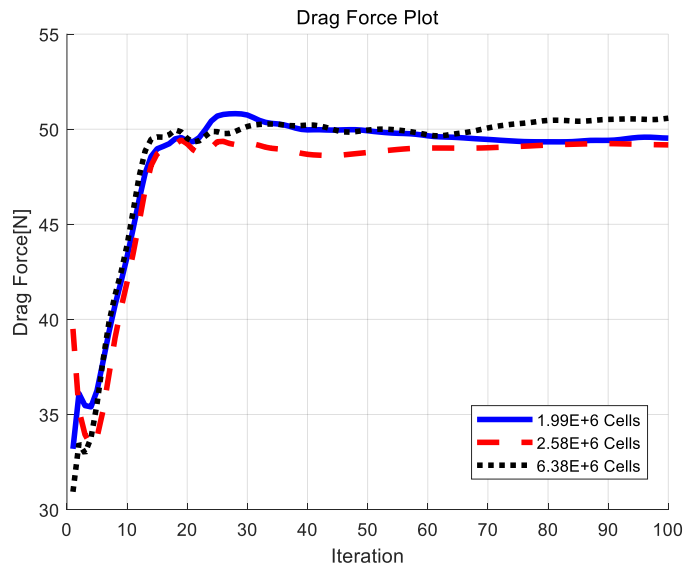


Fig 10. Mesh Convergence

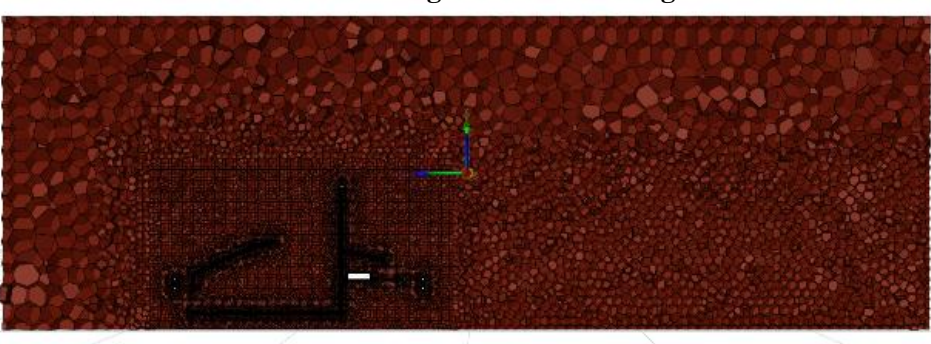
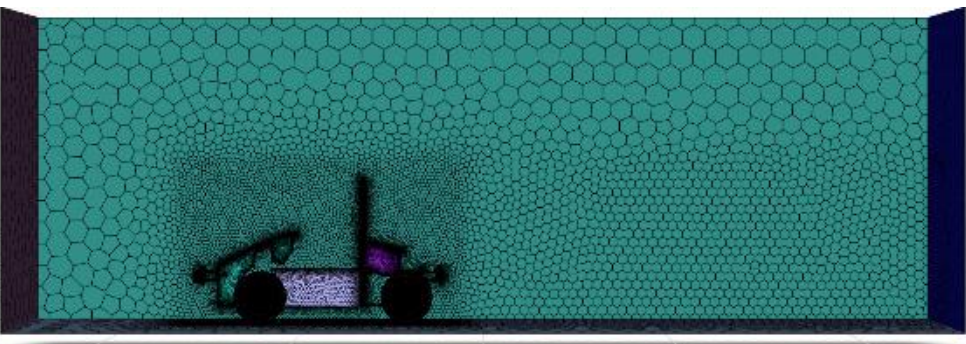


Fig 11. Surface Mesh (Left), Volume Mesh (Right)

## Turbulence Model & Boundary Conditions

### Used Turbulence Model: Spalart-Allmaras (1-eqn)

- Single Transport Equation -(3)
- Suitable for External Boundary Layer Flow Analysis
- Low Computational Cost

$$\frac{\partial v_t}{\partial t} + U_j \frac{\partial v_t}{\partial x_j} = P - D + T \quad (3)$$

$$F_D = \int (v + v_t) \frac{\partial u}{\partial y} \Big|_{wall} dA \quad (4)$$

$v_t$ : turbulent viscosity coefficient  
 $U_j$ : velocity component  
 $x_j$ : coordinate component  
 $P$ : production term,  
 $D$ : dissipation term,  
 $T$ : diffusion term

Table 4. Boundary Conditions used in CFD Simulation

Type	Condition	Speed
Inlet	Velocity-Inlet	20 [m/s]
Outlet	Pressure-Outlet	-
Cowl	Stationary-Wall	-
Frame	Stationary-Wall	-
Wheel	Rotational	87.489 [rad/s]
Ground	Translational	-20 [m/s]
Tunnel-Wall	No Shear	-

## CFD Result (Drag Force)

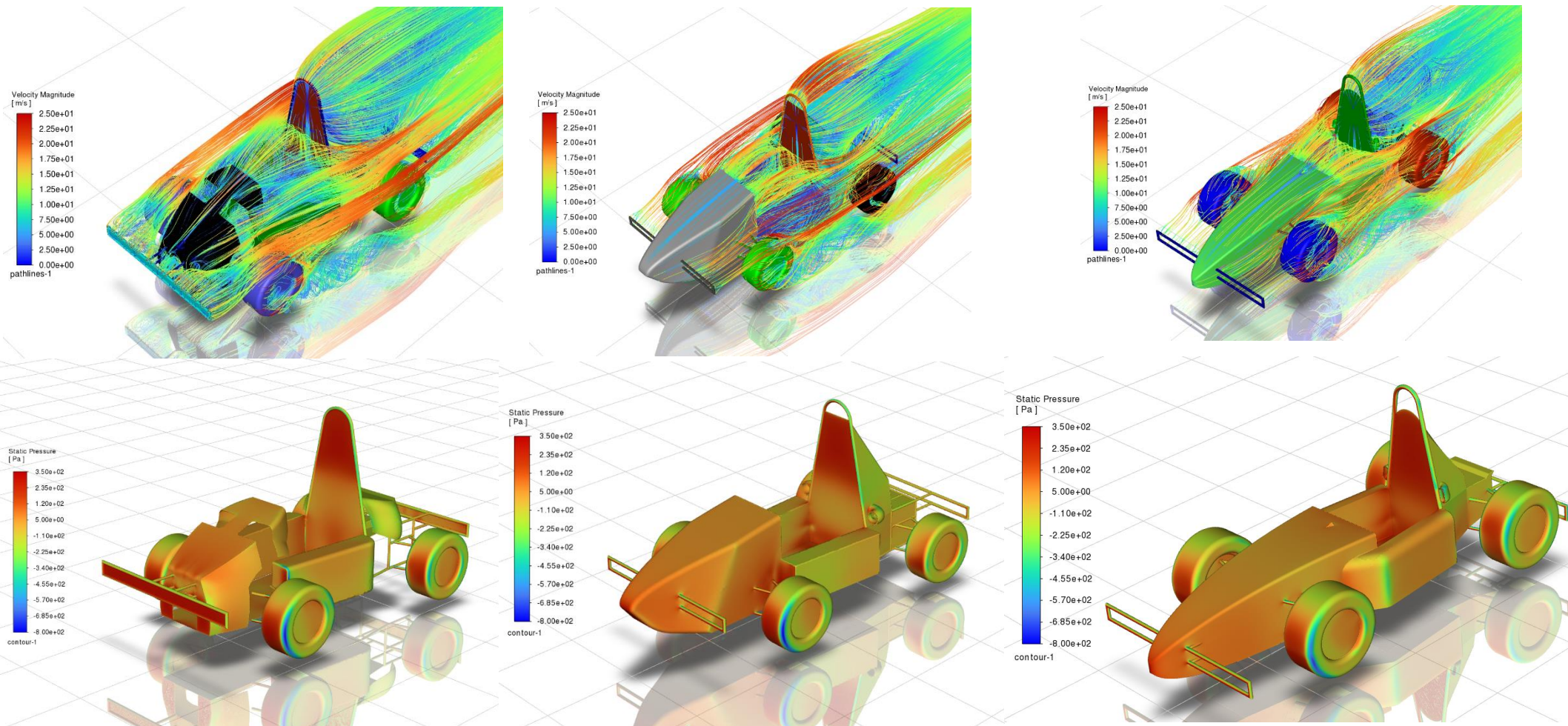


Fig 12. Velocity pathlines and Pressure Contour of Models

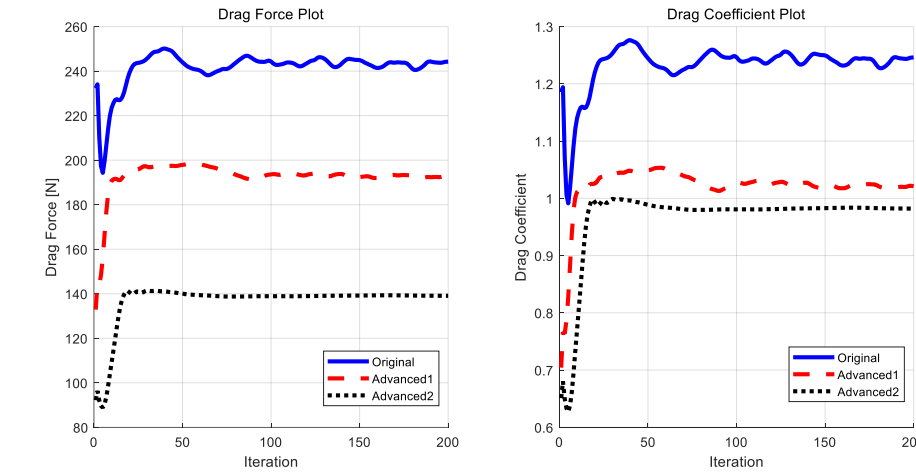


Fig 13. Drag Force and Drag Coefficient

- Advanced1 and Advanced2 reduced drag by 21.1% and 43%.
- Drag Coefficient reduced by 17.7% and 21%.
- RRH and front wheels caused the most drag, with vortices around wheels and rear.
- Streamlined design reduced the drag coefficient.
- Air vent drag did not significantly impact overall drag.

## CFD Result (Cooling)

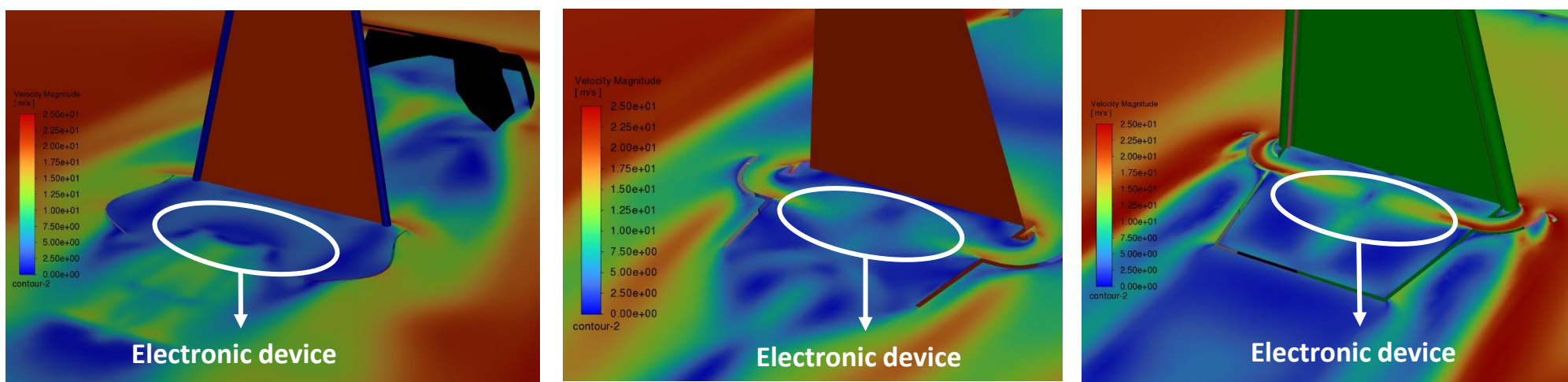


Fig 14. Rear Cowl Velocity Contour (Original, Advanced 1, Advanced 2)

- By adding an air vent shape, fast-flowing fluid through a narrow passage cools the electronic device with lower temperature fluid. ( $2.5m/s \rightarrow 10m/s \rightarrow 16m/s$ )

## Wind Tunnel Experiment Setting [Further Study]

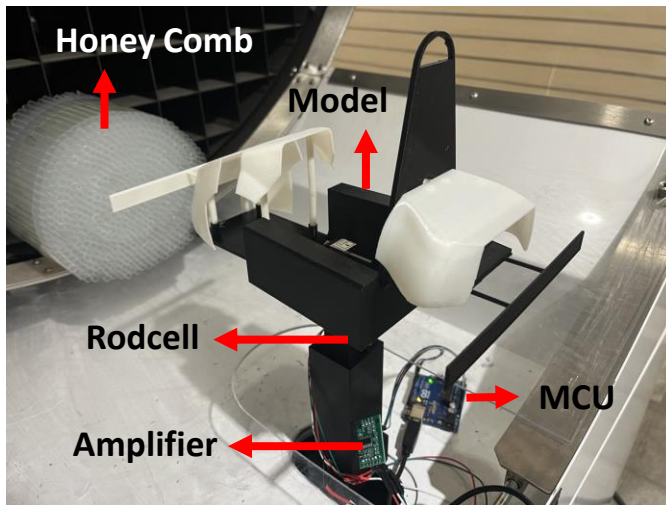


Fig 15. Wind Tunnel Experiment Settings

- Uniform fluid flow is formed after passing through the honeycomb.
- The model's air resistance deforms the rod cell, changing its electrical resistance
- The amplified signal is sent to the MCU to calculate drag.
- The drag coefficient is calculated using the anemometer's fluid velocity and atmospheric conditions, and compared with simulation results

## Discussion & Conclusion

Table 5. DIY Car Spec Sheet

Power [KW]	8.6
Torque [Nm]	33.9
$\omega$ [rad/s]	254.59
Wheel Radius [m]	0.115
$Z_A: Z_B$	17:50
$m$ [kg]	250
$f_t$ : friction loss	0.7
$c_t$ : circuit loss	0.85

$$F_{thrust} = \frac{T}{R} \times \frac{Z_B}{Z_A} \times f_t \times c_t = 515.87[N]$$

- The reduced drag (105[N]) is significant compared to the drive force (515.87[N]) from Table 4.
- Reduced drag can improve top speed and acceleration.
- Drag is proportional to the square of speed, so reducing the projected area and drag coefficient reduces drag.
- Safety device (RRH) regulations limited drag coefficient reduction ( $1.25 \rightarrow 0.98$ ).
- Adding air vents improved cooling and motor efficiency.
- Running time: 90min. Proper geometry and mesh generation are crucial (16-core process, 2.9 million cells).