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

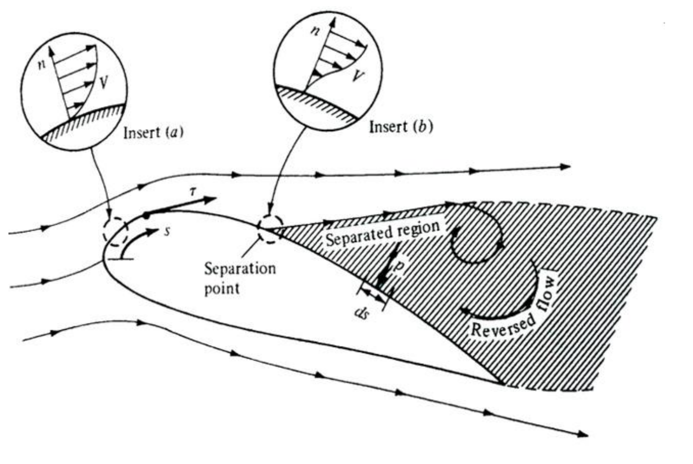


Fig 1. Separation Phenomenon

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

- **Purpose:** Improve driving performance and cooling of electronic devices of KSAE E-mobility
- **Objective:** To minimize the flow separation phenomenon (Fig 1), the vehicle was designed with a streamlined shape to reduce the drag coefficient (C_D) and decrease the vehicle frontal area (A)

Model Description

- **Advanced 1 (Frame Unmodified)**

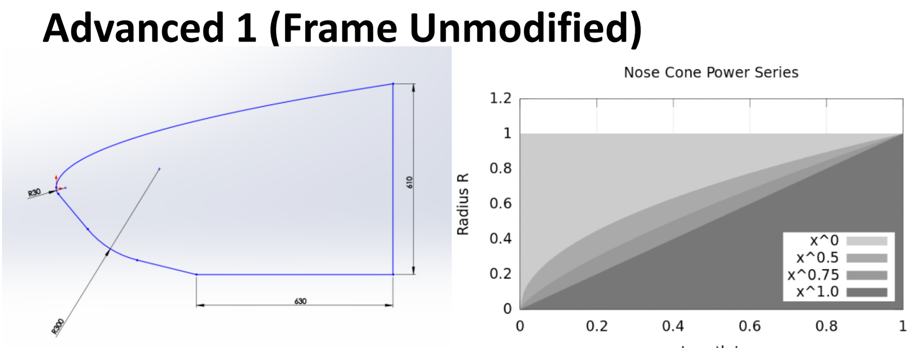


Fig 2. Power 0.5 Nose Cone Design

- 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.

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

- **Advanced 2 (Frame Modified)**

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

	NACA 4412		NACA 2412		NACA 6409	
$\alpha°$	$\frac{C_L}{C_D}$	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	-	-

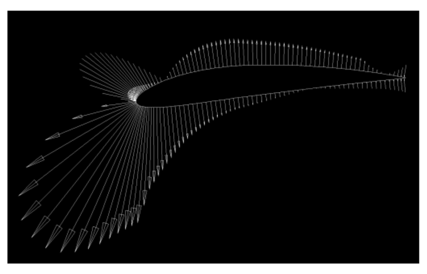


Fig 3. NACA 4412 ($\alpha = -6°$)(XFOIL)

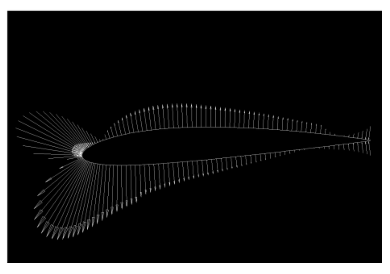


Fig 4. NACA 2412 ($\alpha = -3°$)(XFOIL)

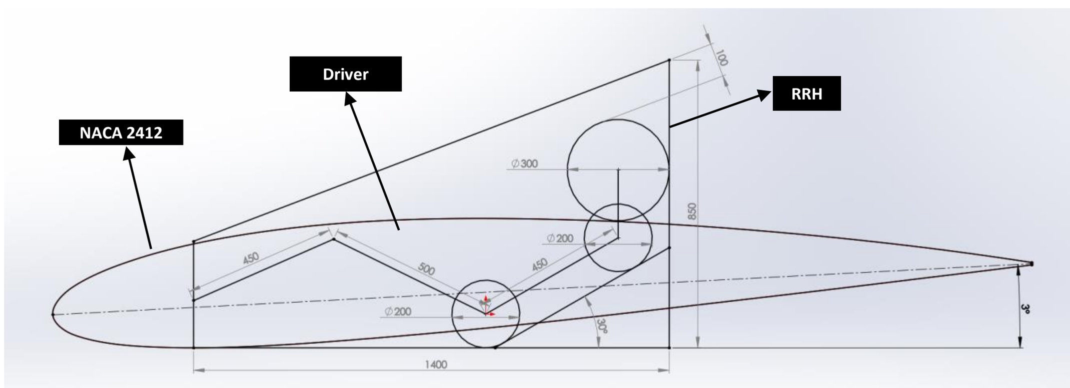


Fig 6. Advanced 2 Model Sketch (NACA 2412)

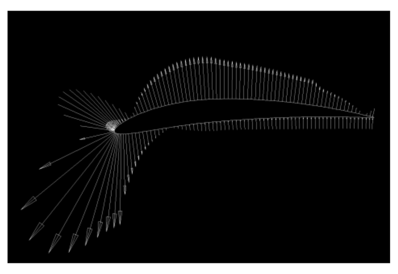
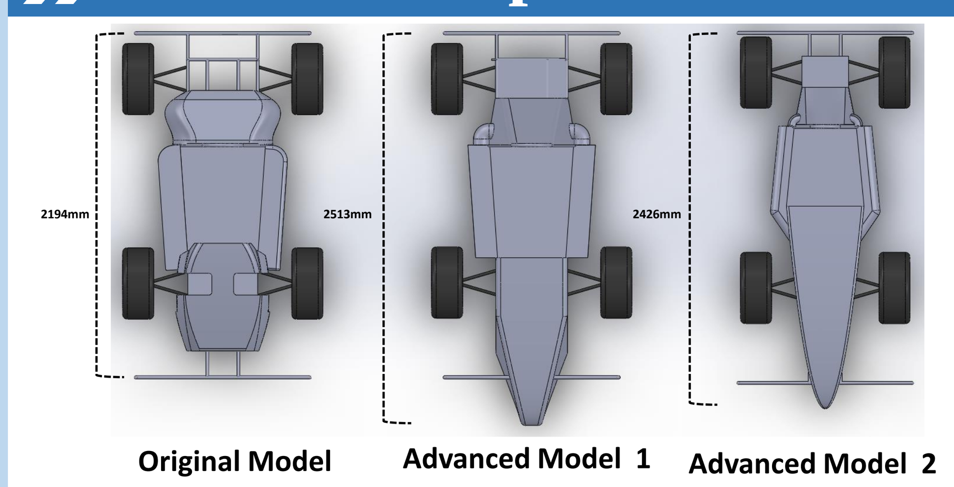
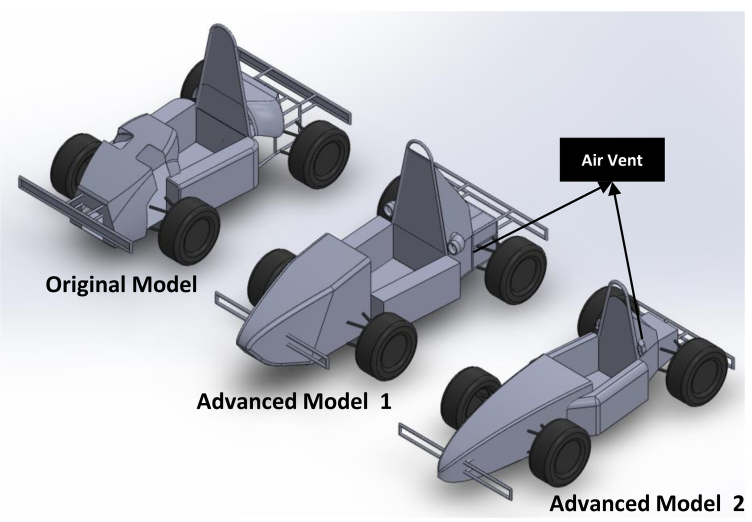


Fig 5. NACA 4412 ($\alpha = -3°$)(XFOIL)

- The design is based on the NACA 2412 airfoil (at AOA, $\alpha = -3°$), 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





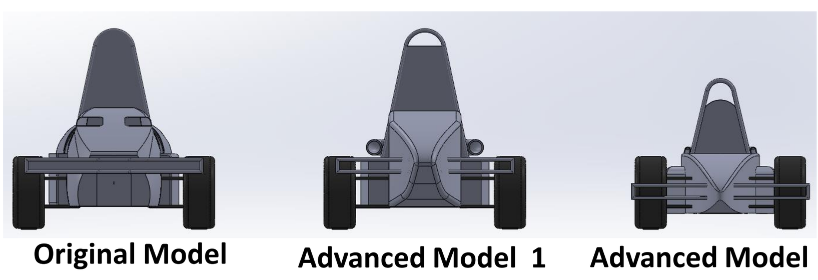


Table 2. Projected Area of Models

	Original Model	Advanced Model 1	Advanced Model 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

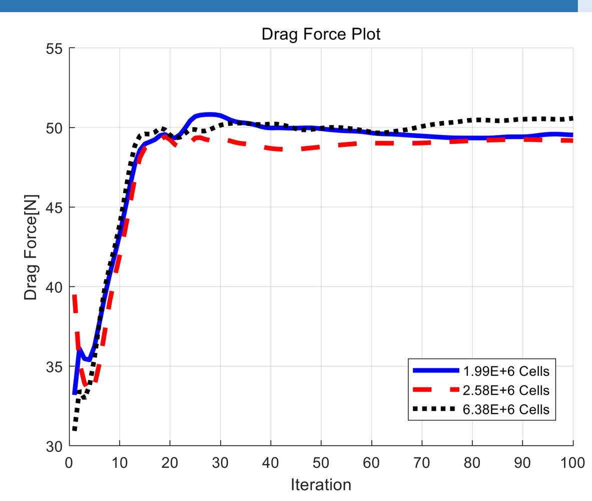


Fig 7. Mesh Convergence

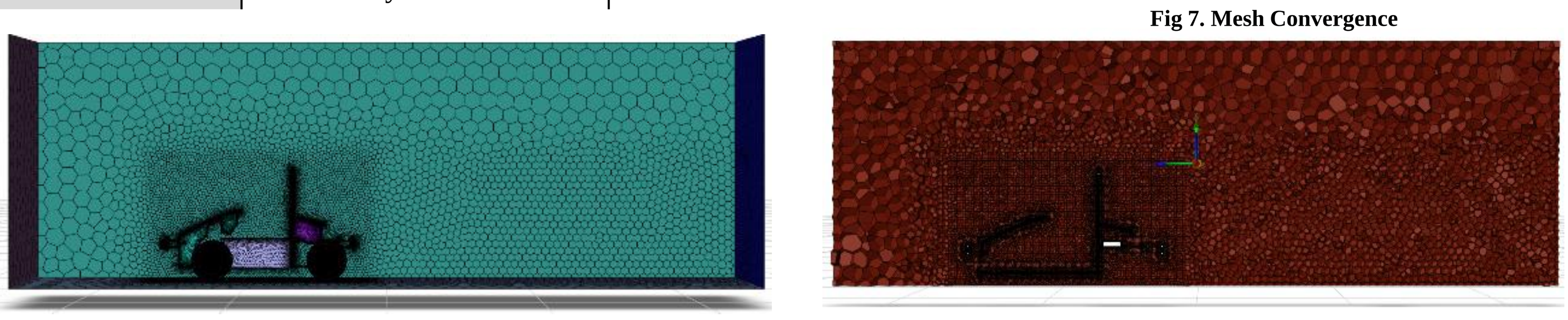


Fig 8. Surface Mesh (Left), Volume Mesh (Right)

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

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

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

v_t: *turbulent viscousity 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)

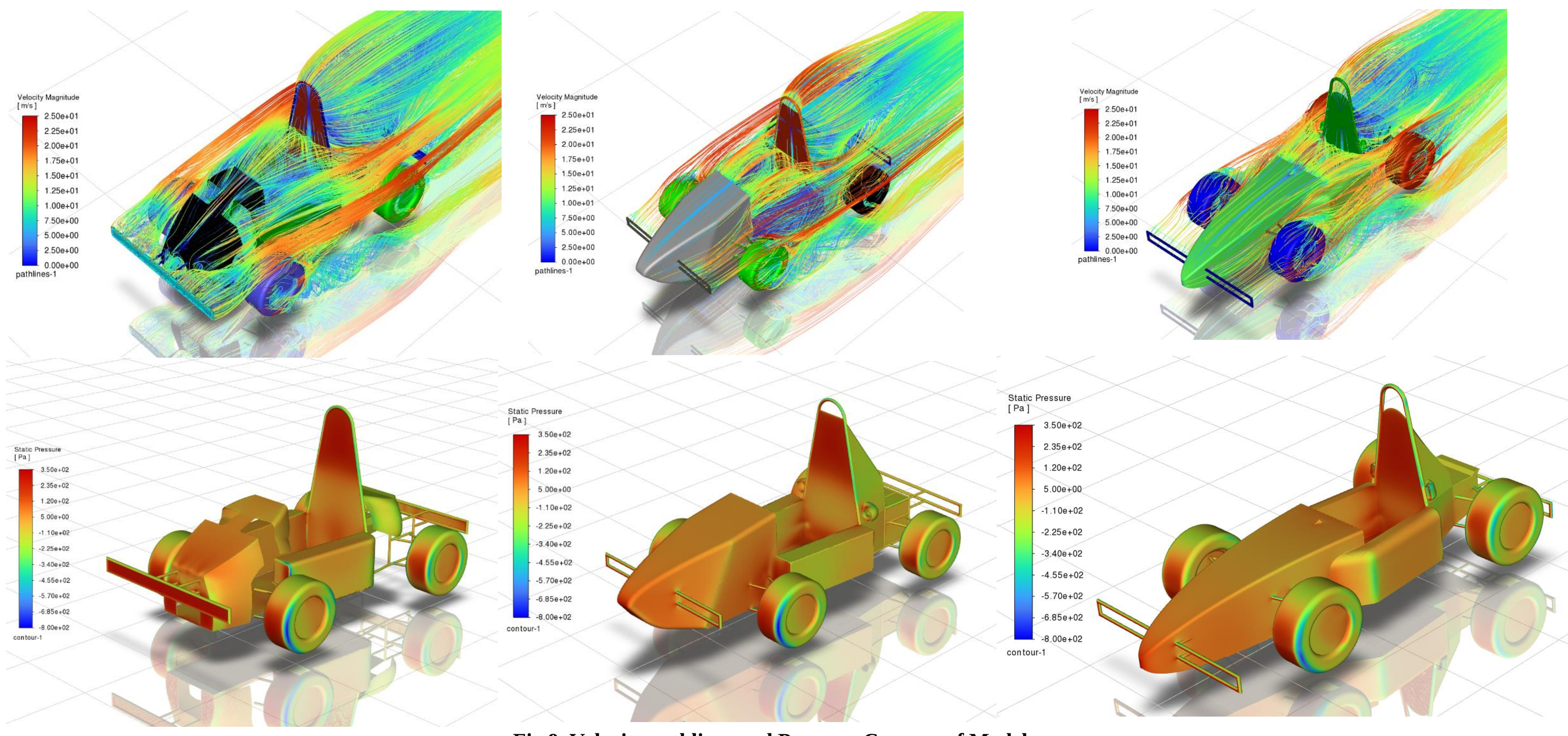


Fig 9. Velocity pathlines and Pressure Contour of Models

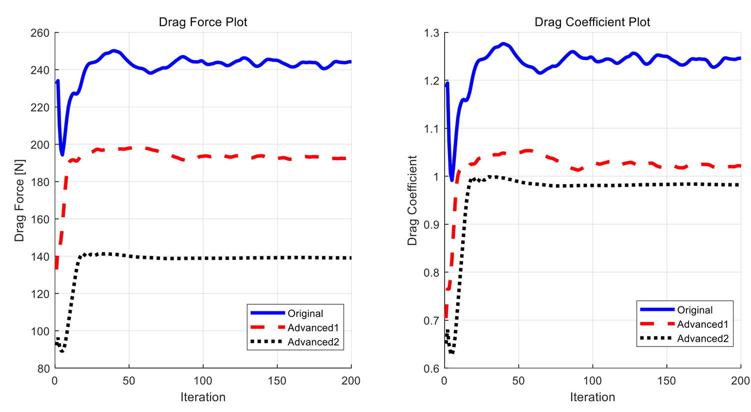


Fig 10. Drag Force and Drag Coefficient

- Drag force reduced by 21.1% for Advanced 1 model and by 43% for Advanced 2 model.
- Drag Coefficient reduced by 17.7% and 21%.
- RRH(Rear Roll Hoop) and front wheels caused the most drag, due to vortices in rear area.
- Streamlined design for advanced models reduced the drag coefficient.
- Reduced frontal area mostly contributed to reduction of drag force.
- Air vent drag did not significantly impact overall drag.

CFD Result (Cooling)

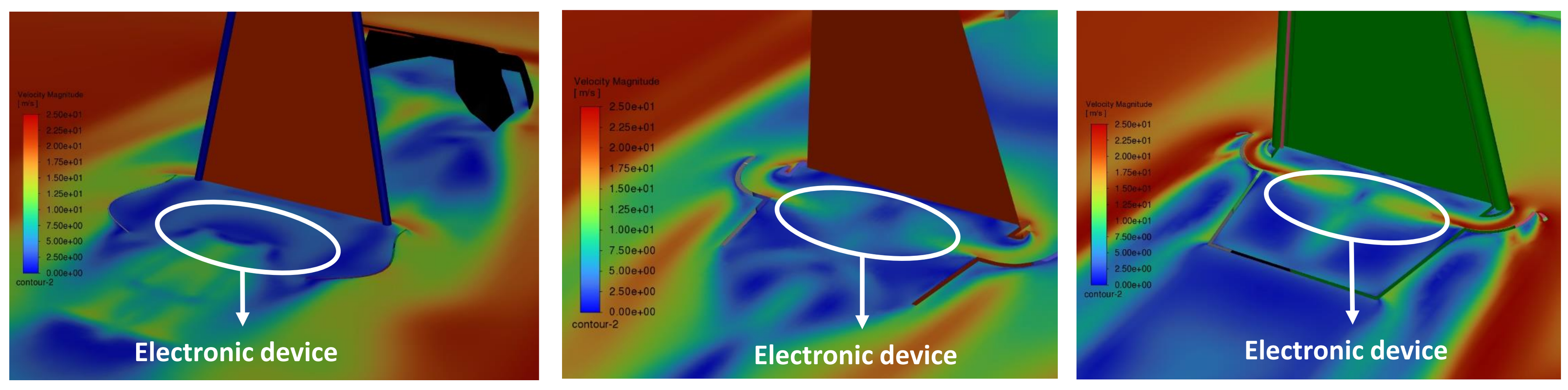


Fig 11. Rear Cowl Velocity Contour (Original, Advanced 1, Advanced 2)

- 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

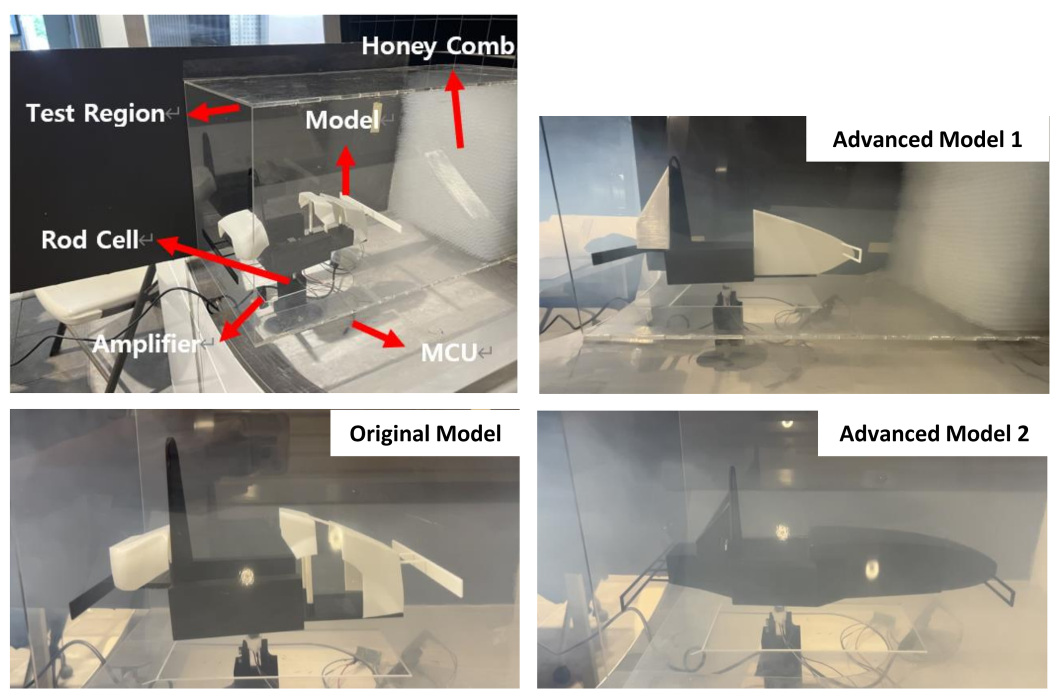


Fig 12. Wind Tunnel Experiment

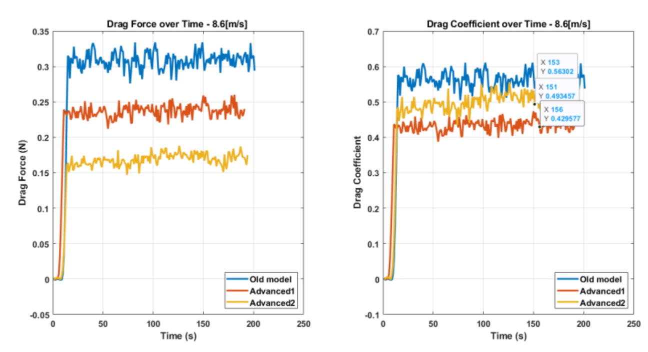


Fig 16. Drag Force & Drag Coefficient (8.6[m/s])

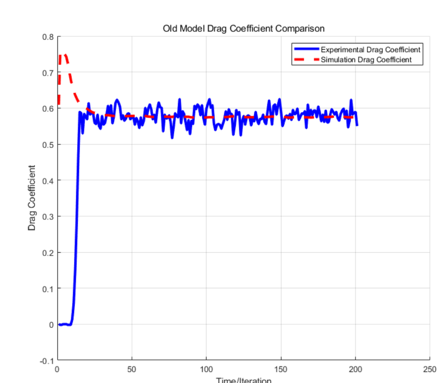


Fig 13. Comparison of Experimental and Simulation Results

Discussion & Conclusion

Table 5. DIY Car Spec Sheet

Power [KW]	8.6
Torque [Nm]	33.9
ω [rad/s]	254.59
Wheel Radius [m]	0.115
Z_A:Z_B	17:50
m [kg]	250
f_l: friction loss	0.7
c_l: circuit loss	0.85

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

- The reduced drag (105[N]) is significant compared to the traction force of the vehicle (515.87[N]) in Table 5.
- Reducing the projected area and drag coefficient resulted in decreased drag force.
- Reduced drag can improve top speed and acceleration.
- Safety device (RRH) regulations limited drag force reduction up to 43%.
- Adding air vents improved cooling and motor efficiency.
- Simulation time: 90min. Proper geometry and mesh generation are crucial (16-core process, 2.9 million cells).