

California State University, Pomona
Electromechanical Engineering Technology
College of Engineering

**CFD Aerodynamic Analysis of a 1995
Mercedes-Benz E320: Stock vs. Modified
Bumper**

I. Introduction

Aerodynamics plays a major role in how vehicles behave at speed: affecting stability, efficiency, and real-world performance. Mercedes-Benz, in particular, has a long history of treating aerodynamics as both an art and a science. Beginning with the 1938 540K Streamliner and evolving through experimental concepts like the C111, the brand set industry benchmarks decades before aerodynamics became mainstream in consumer vehicles. By 1984, the Mercedes W124 became the first production car to break a drag coefficient of 0.30, landing at $C_d = 0.29$. More recently, the EQS achieved $C_d = 0.20$, and the VISION EQXX reduced drag even further to $C_d = 0.173$, which is lower than an American football. Reducing drag by just 0.01 increases EV highway range by roughly 2.5%, translating to over 230 extra miles per year for an average driver [1].

This is the motivation behind this project. I wanted to simulate the airflow characteristics of my own car, a 1995 Mercedes-Benz E320 (W124 chassis), and evaluate how modifying the front bumper impacts its drag coefficient. The goal wasn't to design a revolutionary aerodynamic kit, but to quantify whether practical, real-world modifications actually measurably change airflow behavior.

Relevant CFD Background

External vehicle airflow is governed by the Reynolds-Averaged Navier–Stokes (RANS) equations, which describe conservation of mass and momentum:

Continuity equation (mass conservation):

$$\nabla \cdot (\rho u) = 0$$

Momentum equation:

$$\nabla \cdot (\rho u u) = - \nabla p + \nabla \cdot (\mu \nabla u) + F$$

where ρ is density, u is velocity, p is pressure, μ is dynamic viscosity, and F represents body forces. For a full-scale car at 20 m/s, the Reynolds number is very high, so the flow is fully turbulent. Directly resolving all turbulent structures (DNS or LES) would require an enormous mesh, which is not practical for this project. Instead, the RANS equations are closed by a turbulence model.

The standard k – ϵ turbulence model was chosen. It solves two additional transport equations for the turbulent kinetic energy k and its dissipation rate ϵ , which allows the effect of turbulence on the mean flow to be modeled without resolving every eddy [2], [3], [5].

The main aerodynamic metric of interest is the drag coefficient C_d , defined as

$$C_d = 2F_d / \rho V^2 A$$

F_d is the drag force, ρ is the density, V is the velocity, and A is the frontal reference area. Because the objective is to compare a stock and a modified bumper, both cases are set up as similarly as possible. Any change in C_d or lift coefficient C_l should then be attributable only to geometry differences rather than solver settings.

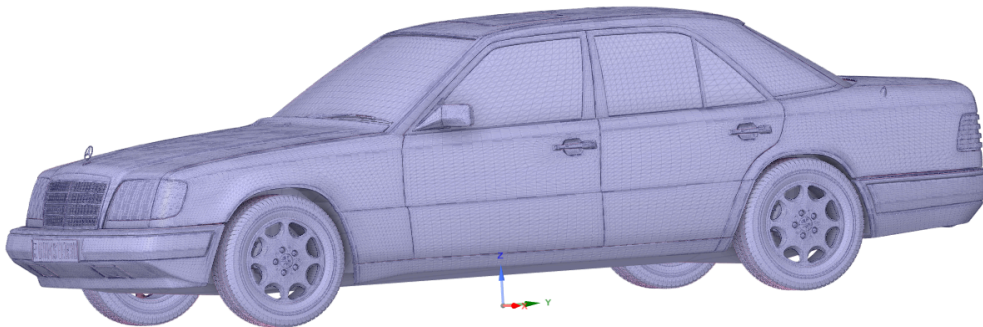


Figure #1 – Imported stock 1995 Mercedes-Benz E320 geometry (≈ 1 million facets) before shrinkwrapping and simplification.

II. Steps in ANSYS (Methodology)

Geometry Processing

The original vehicle model was imported as a detailed OBJ file of a 1995 Mercedes-Benz W124 E320, downloaded from Sketchfab [9]. The geometry was imported as a 1:10 scale, so the first step was to accurately scale the model to full size using known W124 dimensions. Once scaled, several preprocessing operations were required to convert the 3D artist-style mesh into a form suitable for CFD.

Because the OBJ model contained over one million facets, including very sharp edges, deep crevices, and highly detailed rims, the raw geometry could not be meshed directly in ANSYS. These features typically create non-manifold edges and prevent volume mesh generation. To overcome this, the following workflow was applied:

1. **Auto-Skinning:** The model was first processed using auto skin to generate a smooth, continuous surface over the entire vehicle. This step ensures the body behaves like a single watertight solid instead of a patchwork of unstable facets (Figure #10).
2. **Removal of unnecessary details:** Highly detailed features, especially the rims, were removed. These regions cause extreme skewness during meshing and do not meaningfully affect external flow for this level of simulation.
3. **Shrink-Wrapping:** Multiple successive shrink-wrap passes were used to create a uniformly closed surface. Each iteration closed holes, bridged gaps between facets, and removed topological defects. This produced a watertight approximation of the actual car silhouette while preserving all macro-aerodynamic geometry.
4. **Edge Smoothing:** Sharp edges were softened so that the shrink-wrapped surfaces could be repeatedly auto-skinned without errors. The process continued until the model could successfully convert into a valid solid body.
5. **Simplified Rim Replacement:** After the chassis was stable, cylindrical wheel geometries were created and Boolean-added to the car body. This preserved correct wheel positioning without reintroducing high-frequency geometry.
6. **Measuring Reference Frontal Area:** To compute an accurate drag coefficient, the frontal area had to be measured directly from the model. The car was cut at the centerline to expose an infinitely thin mid-plane. The frontal profile area of this center slice was extracted ($\approx 1.853 \text{ m}^2$). Since the slice reflected the whole vehicle width, the actual frontal area was computed as: ($A=1.85/2=0.92 \text{ m}^2$). This value was later used as the reference area in Fluent.
7. **Half-Car Model Creation:** After the area measurement, the car was reassembled and then intentionally cut again along the center plane. The passenger-side half of the vehicle was kept for the simulation. This was done to reduce computational cost by $\sim 50\%$, simplify meshing, and take advantage of symmetry, a common practice for CFD.
8. **Enclosure Creation:** The half-car geometry was placed inside a 3D enclosure sized according to recommended automotive wind-tunnel proportions. The earlier length measurement ($\approx 4.8083 \text{ m}$) was used to scale the domain appropriately (e.g., 3 car lengths upstream and 5 downstream).
9. **Boolean Subtraction:** The half-car was subtracted from the enclosure to create the fluid domain. All boundaries (inlet, outlet, symmetry, road, walls) were then named for Fluent.

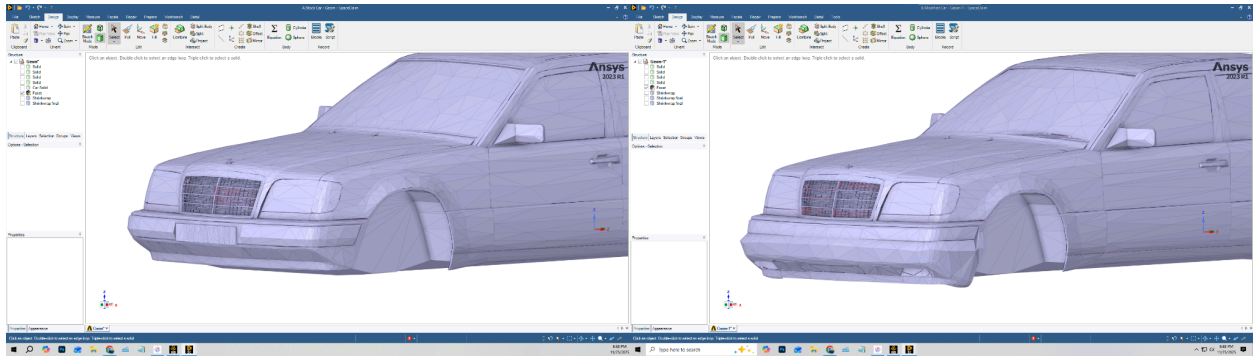


Figure #2 – Stock vs Modified – Front three-quarter view

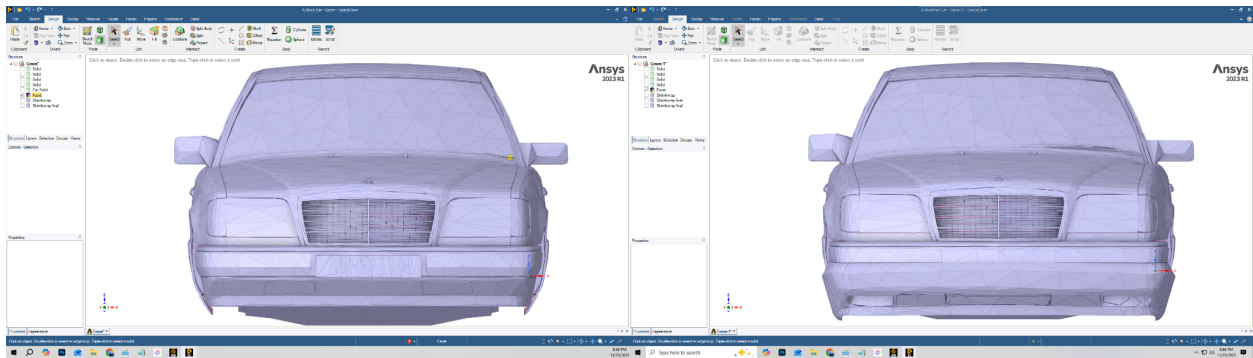


Figure #3 – Stock vs Modified – Front orthographic

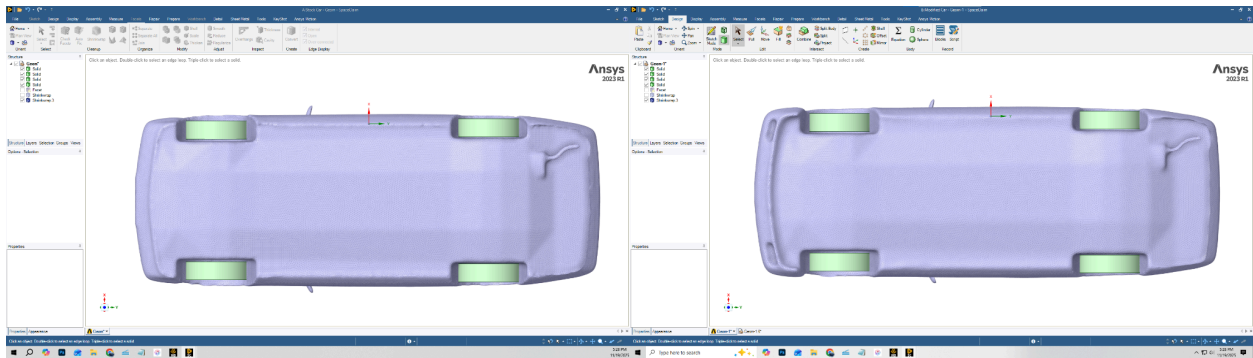


Figure #4 – Stock vs Modified – Bottom View showing solid cylinders for wheels

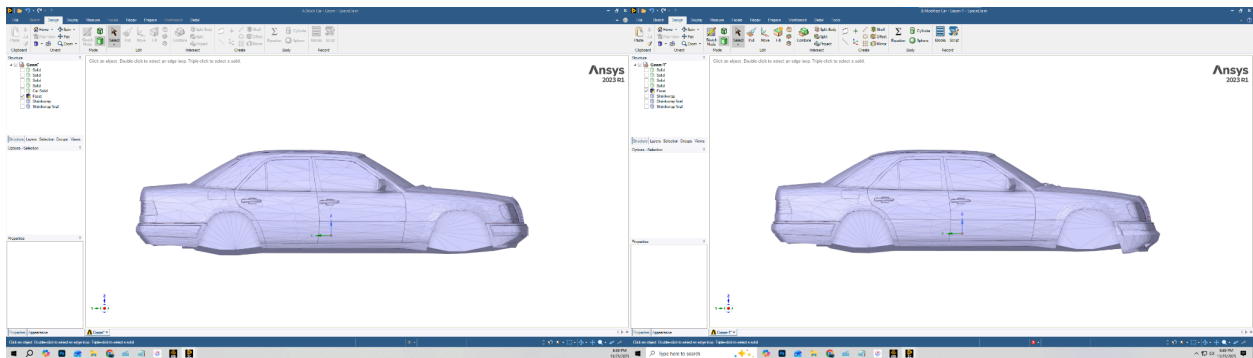


Figure #5 – Stock vs Modified – Side profile

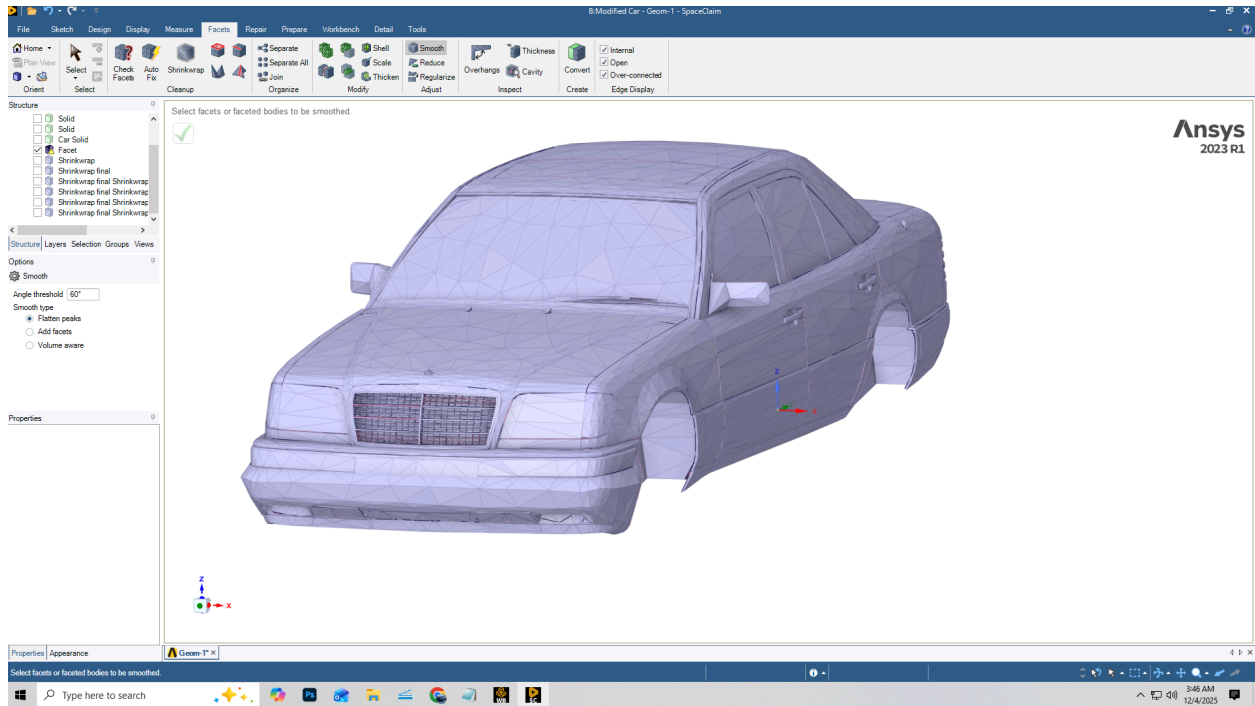


Figure #6 – Modified Bumper – Front Quarter View

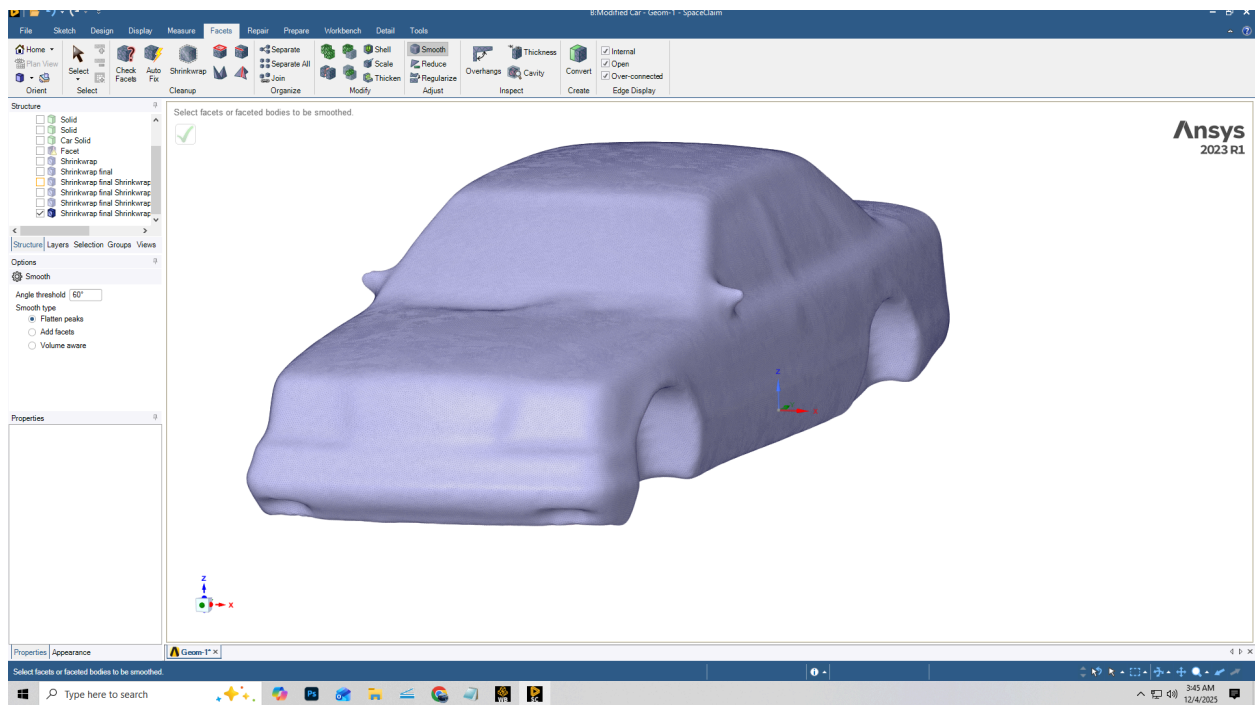


Figure #7 – Shrink-wrapped and smoothed exterior surface of the modified W124 after faceting cleanup.

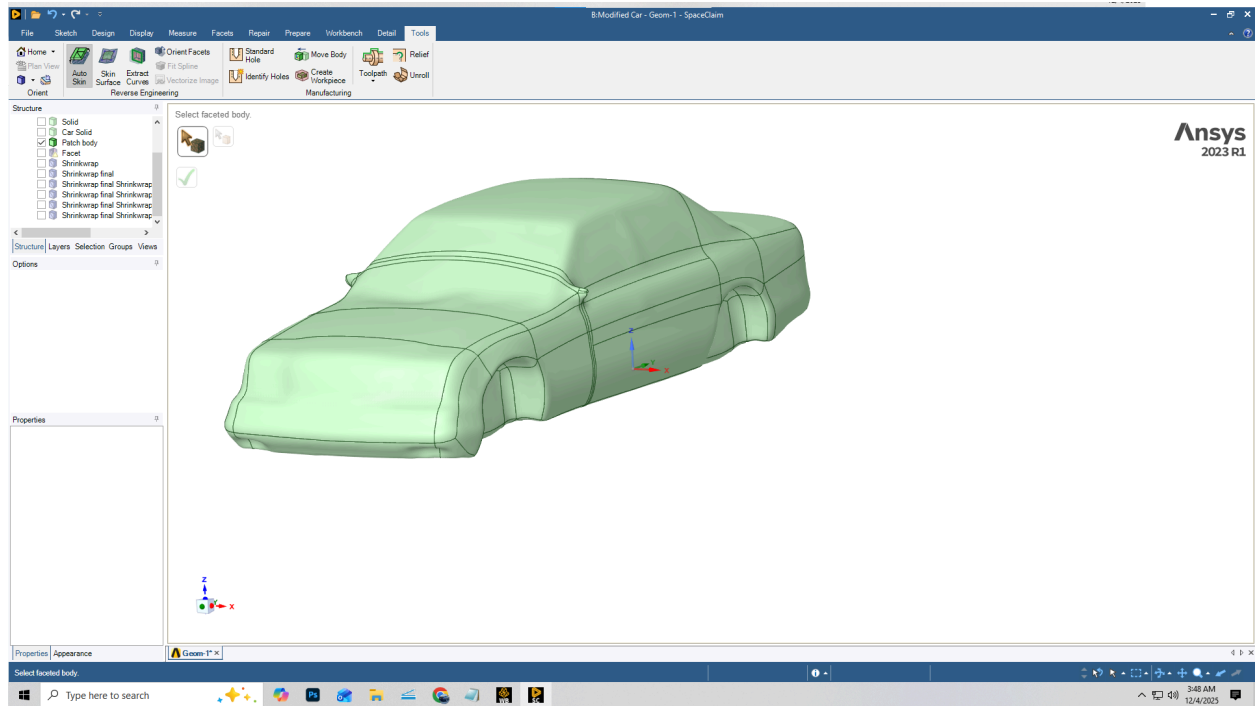


Figure #8 – Autoskin-generated solid body created from the shrink-wrapped surface.

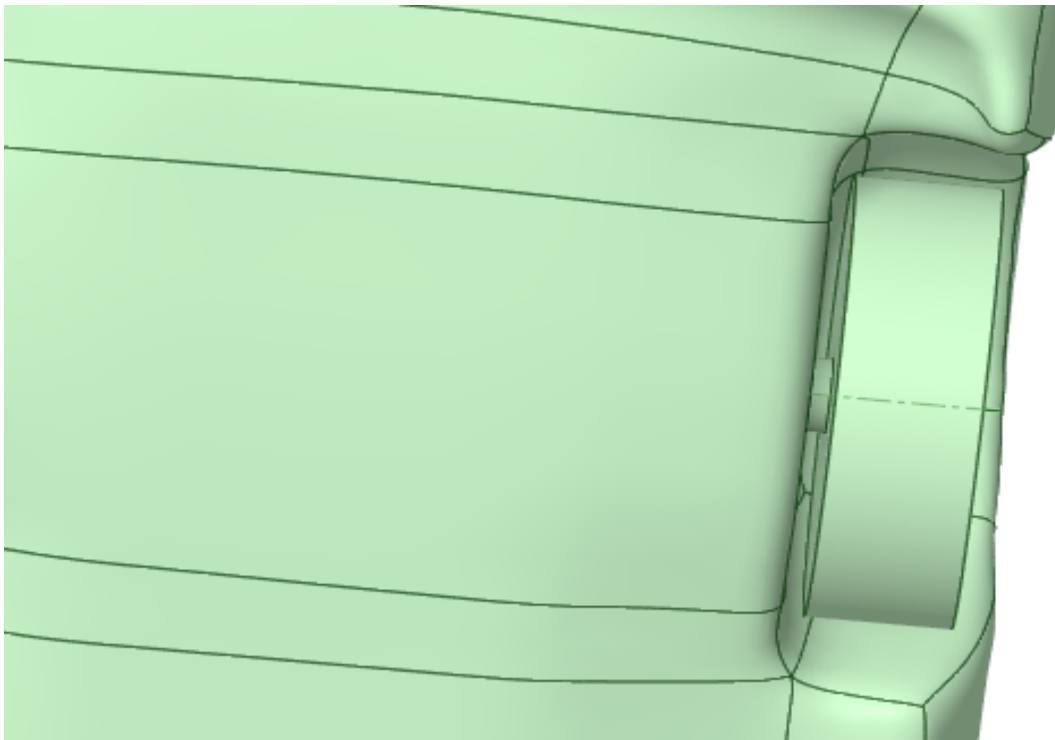
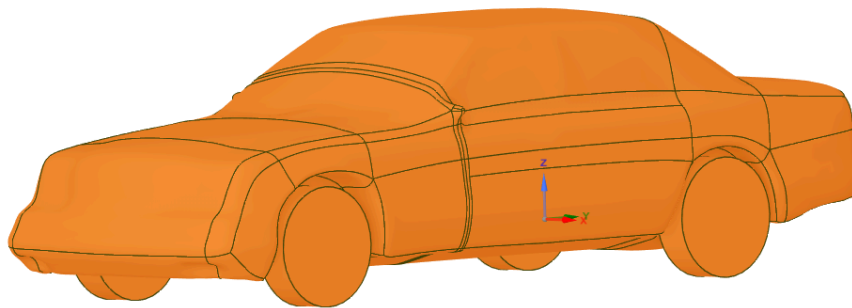


Figure #9 – Close-up of wheel-well fit showing proper Boolean attachment of the cylinders.



Area	27.8546m²
X Overall	1.7673m
Y Overall	4.8083m
Z Overall	1.3986m

Figure #10 – Completed modified-car solid model with simplified wheels and smoothed body surfaces.

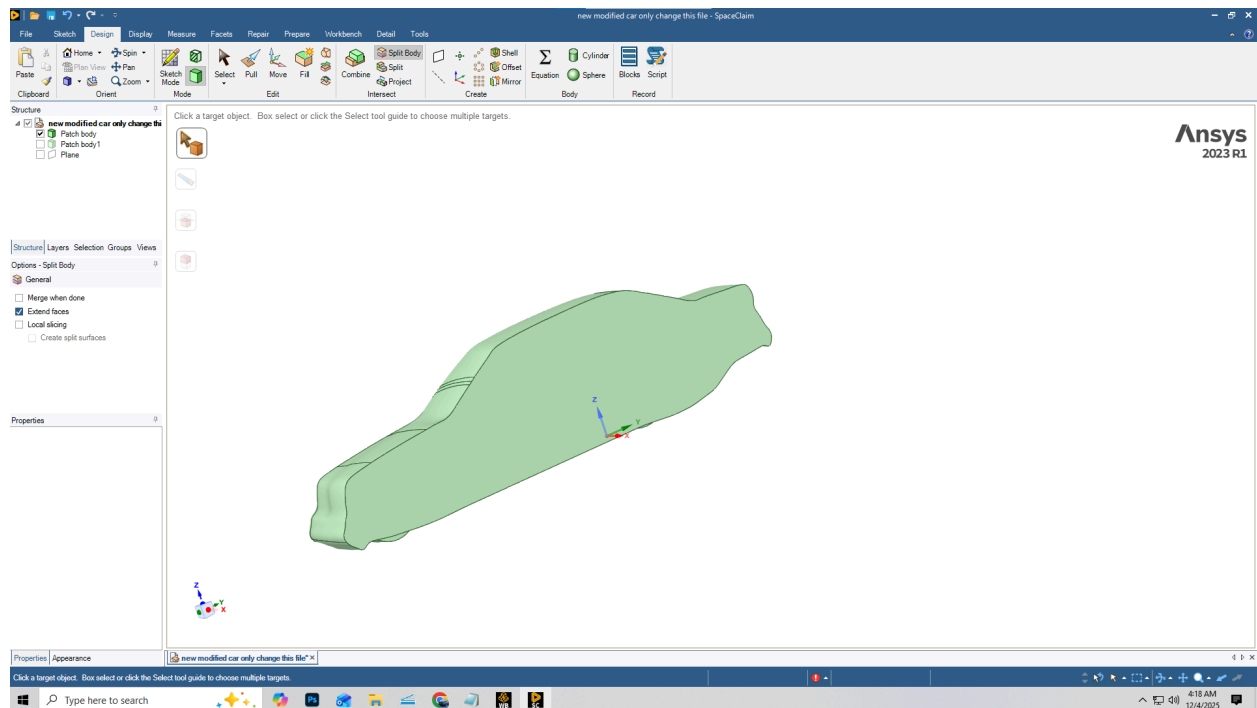


Figure #11 – Passenger-side half of the car retained for simulation after symmetry cut.

Enclosure Creation

Using SpaceClaim, a wind-tunnel-style enclosure was created around the half-car model. The upstream, downstream, and lateral extents were sized based on common automotive CFD practice (on the order of several car lengths upstream and downstream) to allow the flow to develop and the wake to form without being clipped by the boundaries.

The half-car solid was then subtracted from this enclosure to create the fluid domain. Boundary zones were named for use in Fluent: inlet, outlet, symmetry, road, walls, and car. The road and walls represent the stationary test section, the symmetry plane lies on the vehicle centerline, and the inlet/outlet defines the main flow direction. The enclosure was kept large enough in height and width so that blockage effects on the flow around the vehicle remained small [4].

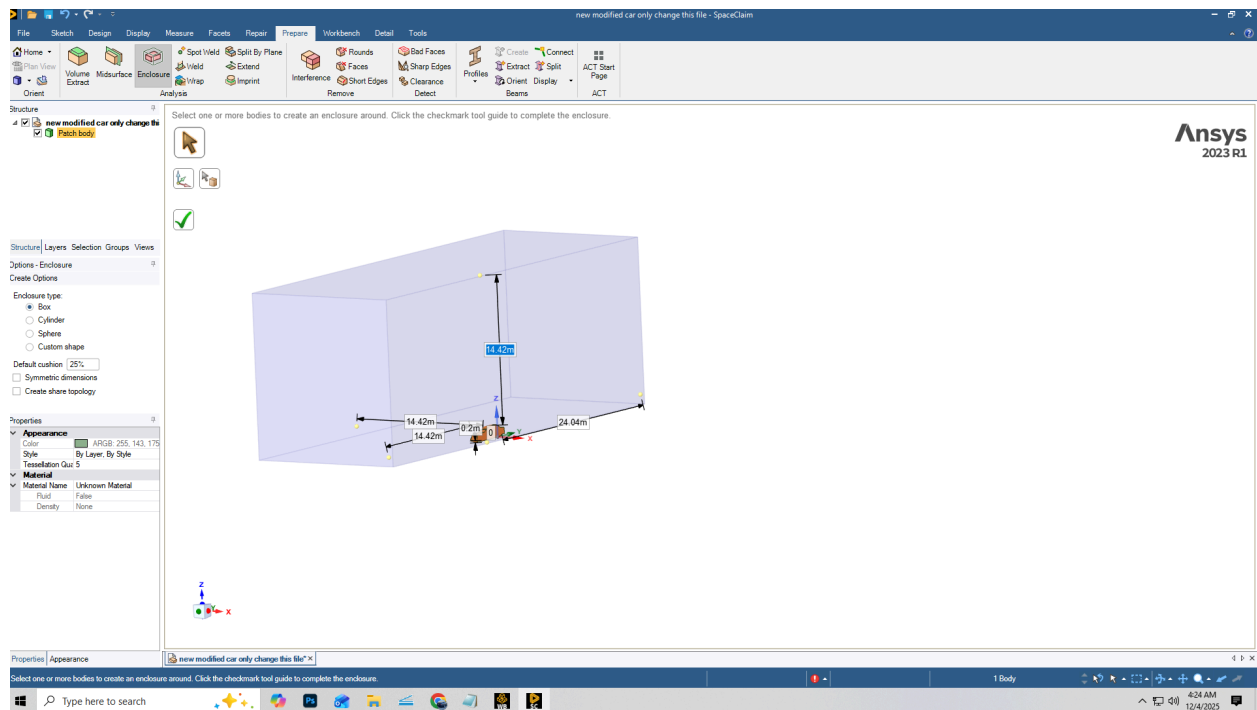


Figure #12 – Flow enclosure sized using rule-of-thumb length multipliers around the half-car.

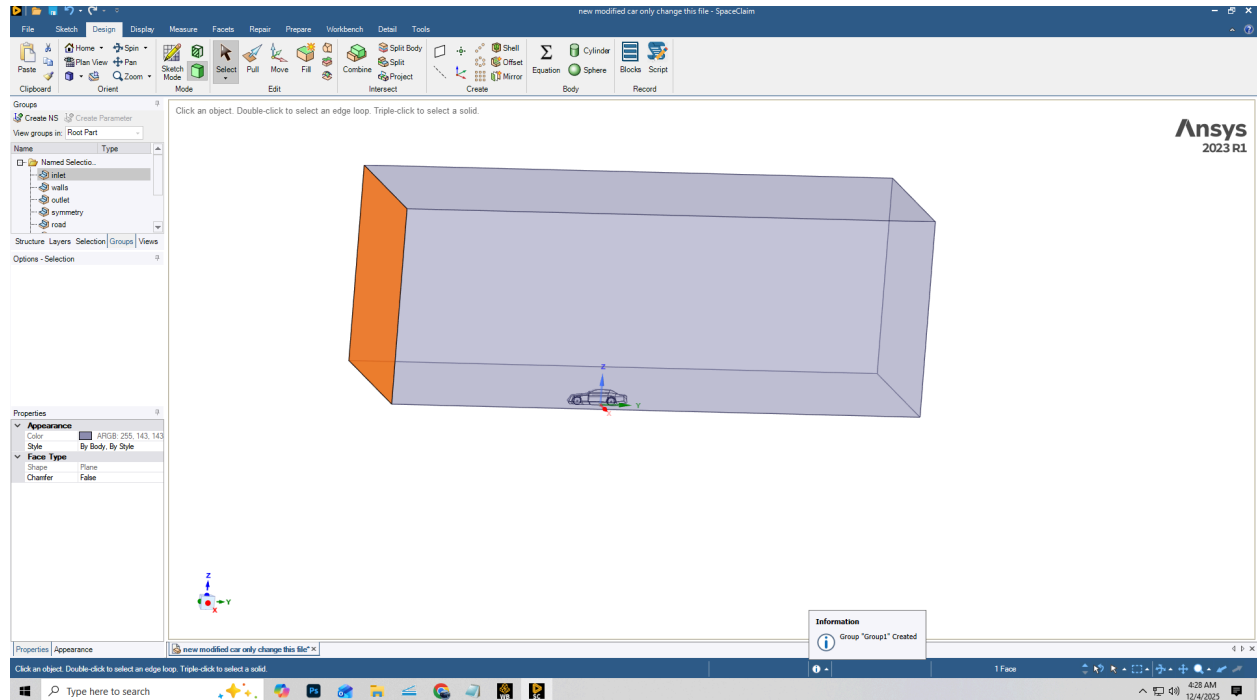


Figure #13 – Fluid domain created by subtracting the solid car from the enclosure volume.

Meshing

Meshing was performed using the Fluent Meshing watertight geometry workflow. The geometry was identified as a single solid body embedded inside one fluid region. A base mesh size was selected to keep the total cell count under approximately 100,000 cells per case, which is modest but acceptable for a comparative study.

Local refinement was introduced around the vehicle using surface sizing and proximity settings. An inflation layer was added near the road and car surfaces to resolve the boundary layer better. The first layer height was set to 0.02 m with an expansion ratio of 1.2, giving several prism layers before transitioning into the unstructured volume mesh (Figure #14).

For the stock bumper case, the final mesh contained 99,044 cells with a minimum orthogonal quality of about 0.14 and a maximum aspect ratio of approximately 141. For the modified bumper case, the mesh contained 95,496 cells with a minimum orthogonal quality of about 0.15 and a higher maximum aspect ratio (~265), reflecting the slightly different local geometry around the new bumper. These quality metrics are typical for shrink-wrapped automotive shapes and are sufficient for qualitative and comparative analysis, although they are not fine enough for high-accuracy prediction of absolute drag.

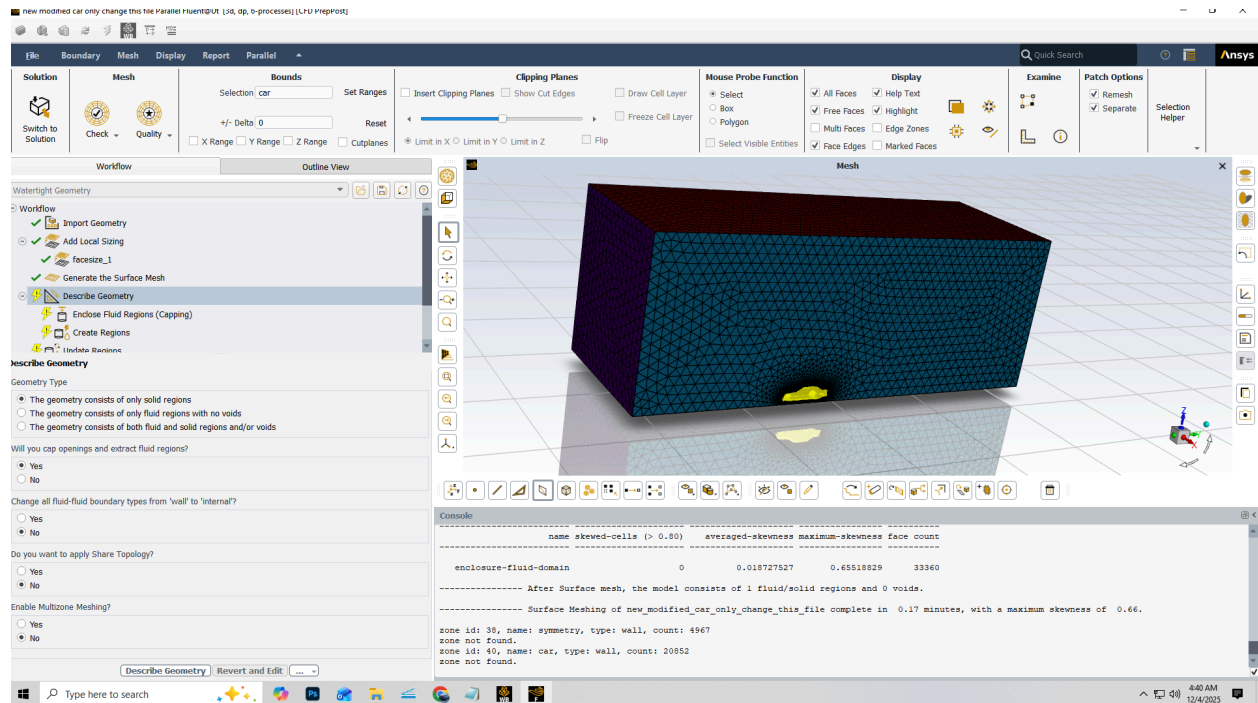


Figure #14 – Surface mesh generated on the car and enclosure with local refinements applied.

Solver Setup

The meshes were exported to Fluent, where both cases shared the same physical and numerical settings. The solver was configured as a steady, pressure-based, three-dimensional RANS simulation of incompressible air. The standard $k-\epsilon$ turbulence model with scalable wall functions was selected [3]. This model is robust on coarser meshes and is widely used in external aerodynamics [2], [5].

A uniform velocity inlet of 20 m/s was applied at the upstream boundary, normal to the inlet face. Turbulence intensity was set to 5% with a viscosity ratio of 10, representing a mildly turbulent approach flow. The outlet boundary was defined as a pressure outlet with zero gauge pressure. The symmetry plane enforced zero normal velocity and gradients across the vehicle centerline. The road and car surfaces were modeled as no-slip walls. All other enclosure faces were also treated as stationary walls (Figure #15). Representative reference values used by Fluent are summarized below (Table #1).

Parameter	Value
Area	~0.92 m ²
Length	4.8083 m
Density	1.225 kg/m ³
Velocity	20 m/s

Table #1 – Reference Values

Convergence

Both cases were run for 500 iterations.

Turbulence, velocities, and momentum all converged; continuity stabilized but did not meet the strict residual target, which is common for external flows with wakes and recirculation.

Although most images in this report illustrate the modified bumper model, both the stock and modified W124 geometries were processed using the same workflow.

Each model underwent identical operations, including: shrink-wrapping, smoothing, auto-skinning, solid generation, wheel replacement, enclosure creation, named selections, mesh sizing, inflation, and solver setup.

By maintaining consistent preprocessing and simulation parameters, any differences in aerodynamic coefficients arise solely from geometric changes rather than differences in numerical setup.

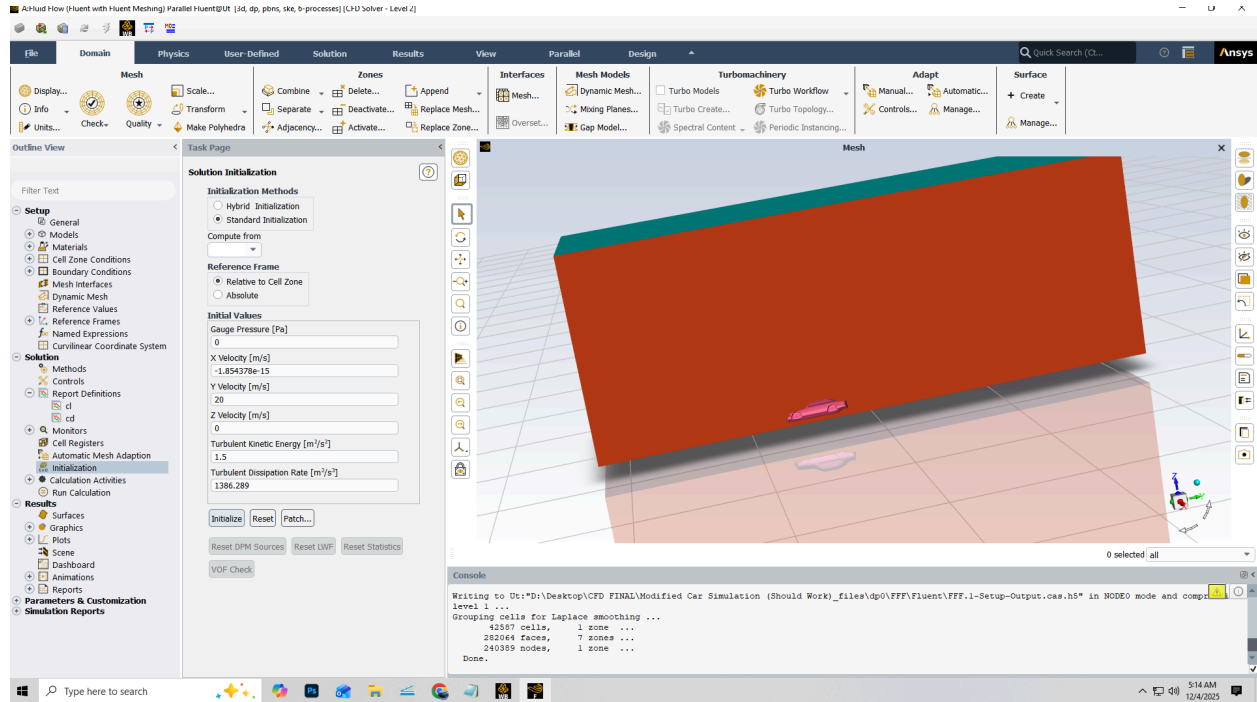


Figure #15 – Boundary condition assignment in ANSYS Fluent showing the enclosure surfaces.

III. Results & Discussion

Case	Cd	Cl
Stock bumper	0.29271	−0.02968
Modified bumper	0.29199	−0.04659

Table #2 – Drag and lift coefficients.

Interpretation

The modified bumper reduced drag by 0.00072, which is about a 0.25% difference. This is extremely small and falls within the expected numerical variation for moderate-resolution CFD meshes on shrink-wrapped vehicle geometry. However, the result is still useful because it shows that the modified bumper does not increase drag relative to the stock W124, which is already known to be a well-optimized sedan aerodynamically [1], [6].

Because the stock and modified cases used identical mesh settings, boundary conditions, turbulence modeling, and solver controls, any differences in drag and lift can be attributed primarily to geometry changes rather than numerical setup. The slightly more negative lift coefficient for the modified bumper indicates a small increase in net downward force, but the magnitude is modest and should be interpreted cautiously, given the mesh resolution and modeling assumptions [4].

From a physics standpoint, most of a three-box sedan's drag comes from pressure losses in the wake behind the rear window, C-pillar, and trunk, plus underbody flow and wheel wakes, rather than only the front bumper [6], [7]. The front bumper helps shape the stagnation region and guide flow around and under the car, but it is not the dominant contributor to overall drag. In this project, the mesh density was on the order of 10^5 cells, the geometry had been smoothed and shrink-wrapped, and a standard $k-\epsilon$ RANS model with wall functions was used as a robust, low-cost turbulence model [2], [3], [5]. Each of these choices introduces some numerical uncertainty in the exact C_d value.

Given this setup, variations on the order of ± 0.001 in C_d are expected between similar meshes, slightly different shrink-wraps, or minor changes in local curvature. The observed 0.25% drag reduction is therefore consistent with both the limitations of the model and the relatively subtle nature of the bumper modification.

Do Bumper Redesigns Matter in General?

In industry, larger front-end and underbody redesigns absolutely matter: changing grille openings, adding air dams, deflectors, underbody panels, and diffusers, and reshaping rear-end geometry can significantly reduce drag and lift when they are designed as part of a complete aero package [6], [8]. Wind-tunnel and CFD studies have shown that such devices and rear-end treatments primarily work by modifying the wake structure, raising base pressure, and managing underbody flow, which can yield meaningful reductions in drag and improvements in stability [6]–[8].

However, minor front-bumper styling tweaks on their own, especially when the underbody is not fully modeled, and the ground and wheels are stationary, tend to produce only local changes in pressure and flow separation with negligible effect on the overall drag coefficient. The results from this project are consistent with that expectation. The modified bumper slightly reshapes local flow and pressure around the front of the car, but within this modeling framework, it does not generate a dramatic global reduction in drag. Instead, it behaves like a visually updated component that is aerodynamically “safe”: it maintains the W124's already well-optimized drag characteristics rather than degrading them.

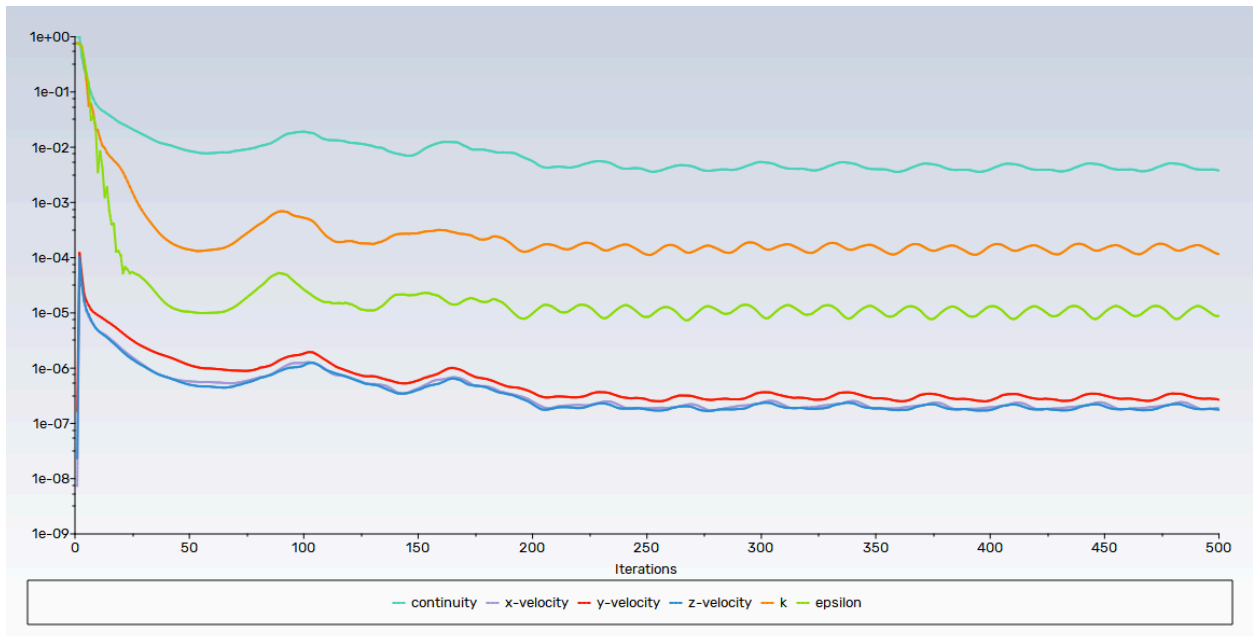


Figure #16 – Residual history for continuity, momentum, k, and ϵ , showing stable reduction over 500 iterations.

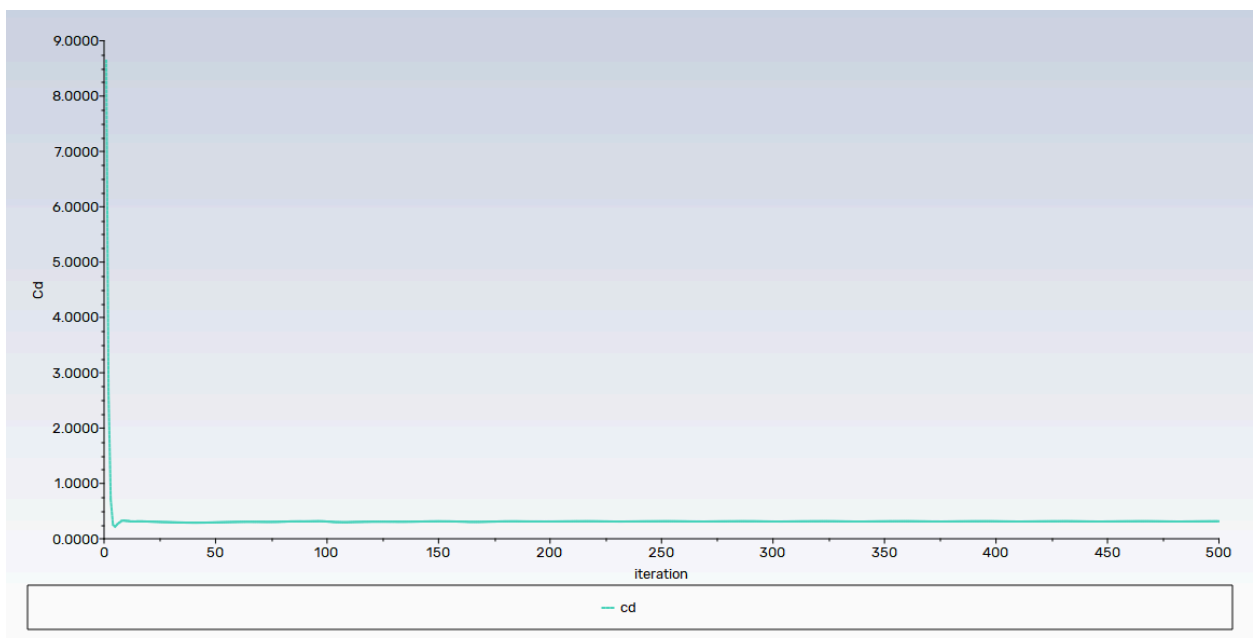


Figure #17 – Drag coefficient convergence history for the modified bumper case.

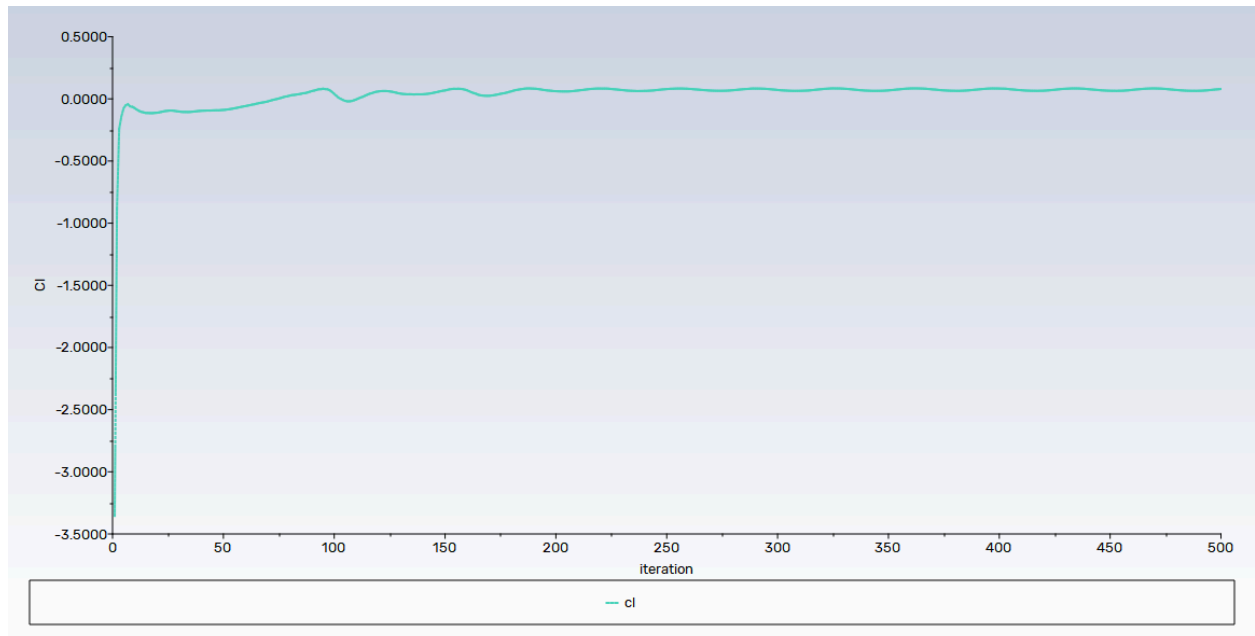


Figure #18 – Lift coefficient convergence history for the modified bumper case.

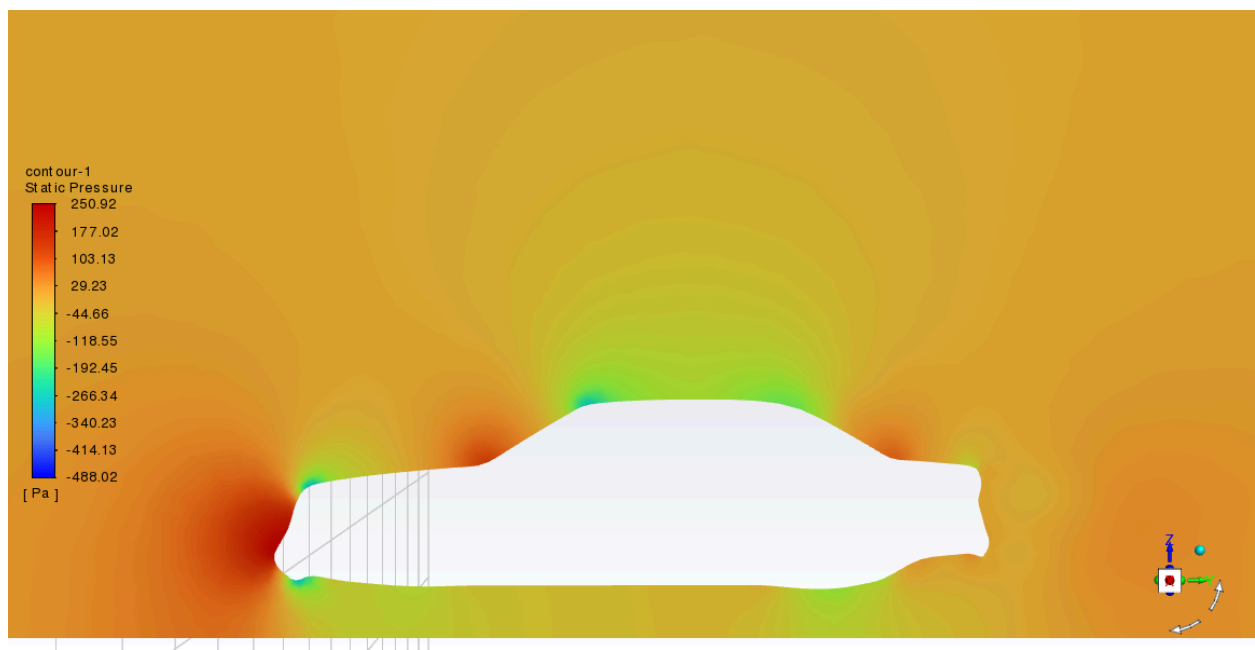


Figure #19 – Pressure contour on central symmetry plane showing stagnation region and wake pressure recovery.

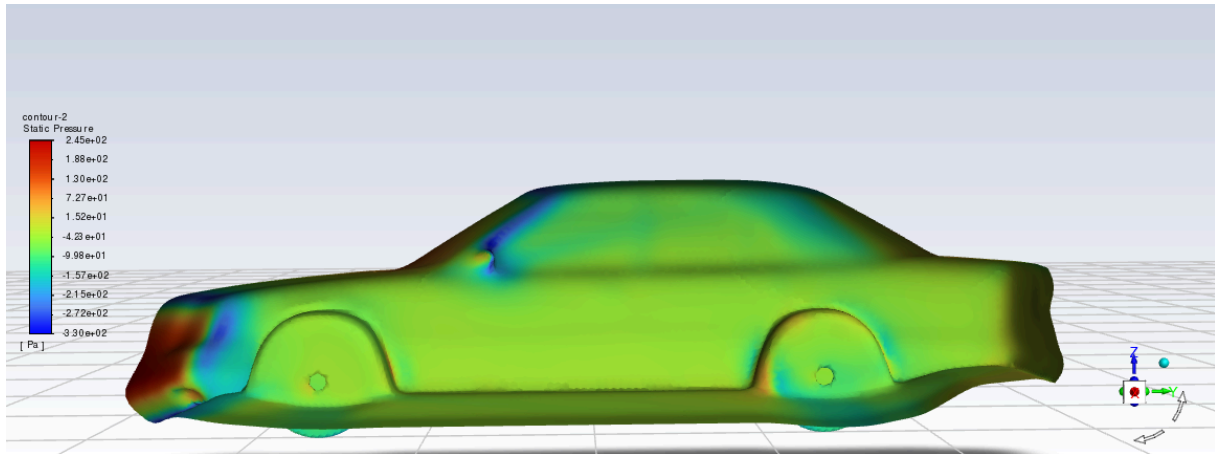


Figure #20 – Static pressure distribution across the modified vehicle exterior.

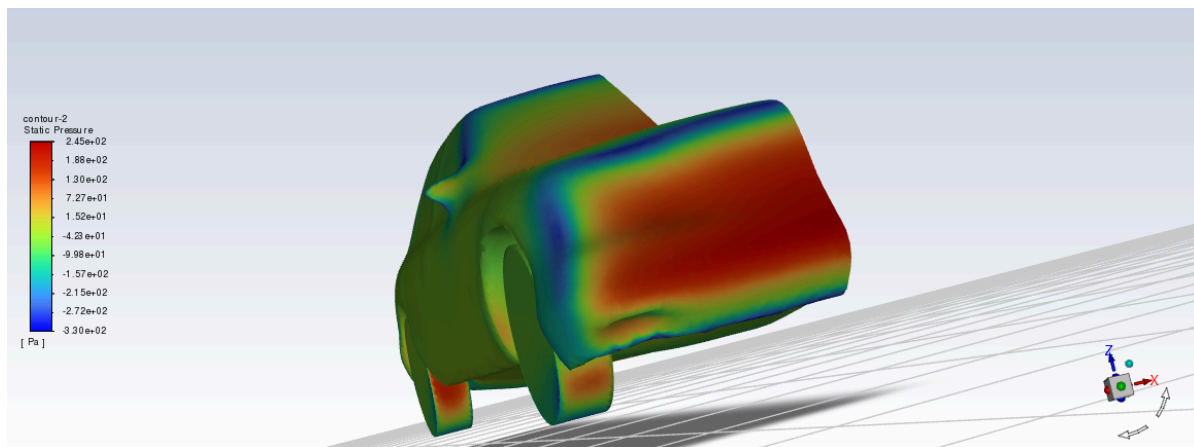


Figure #21 – Surface pressure on the front portion illustrating wake formation and low-pressure recirculation.

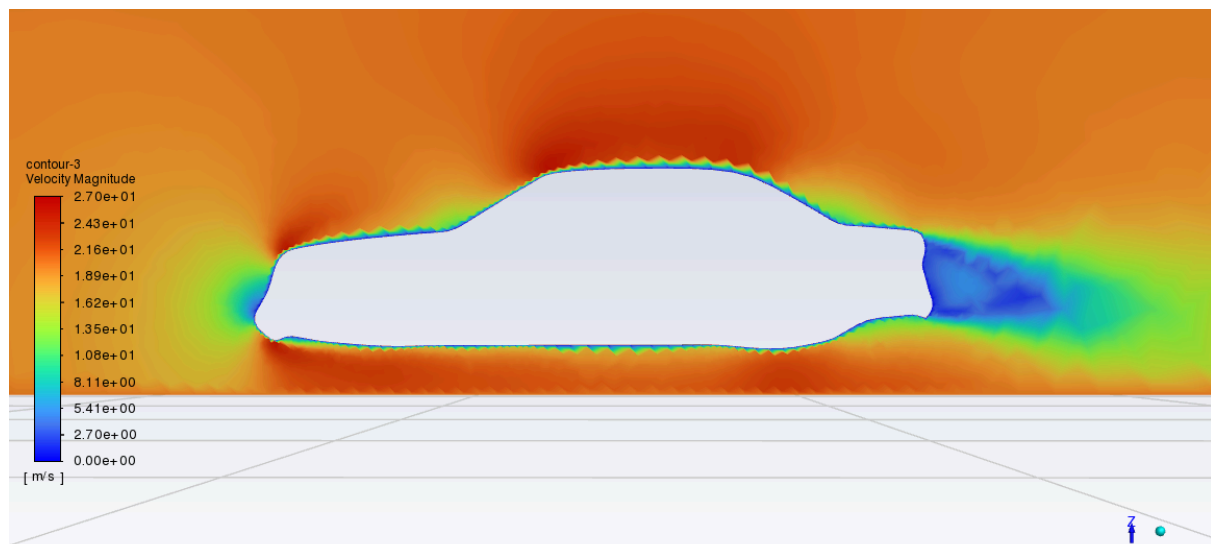


Figure #22 – Velocity magnitude contour on the symmetry plane showing acceleration over the roof and underbody.

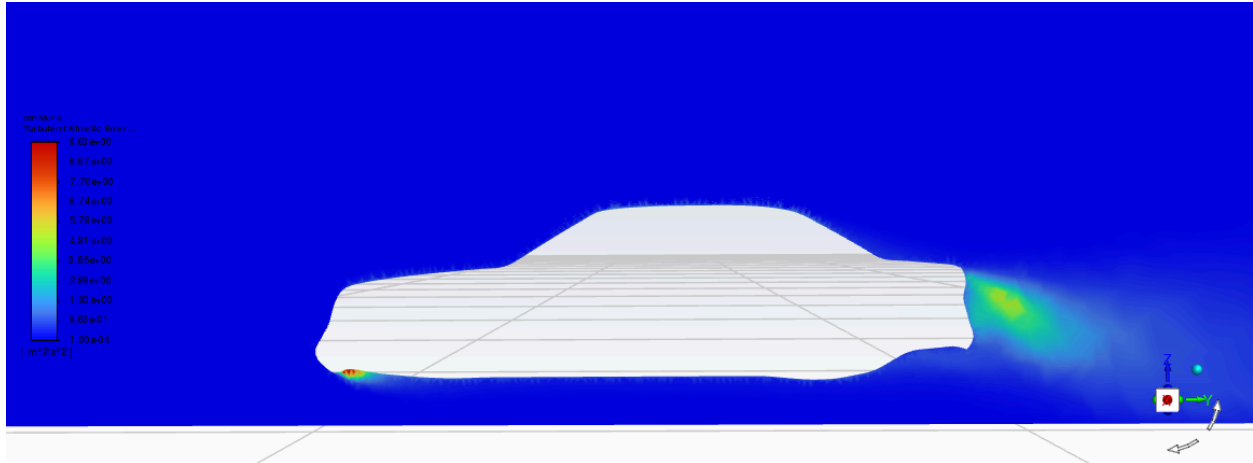


Figure #23 – Turbulent kinetic energy field on symmetry plane highlighting regions of separation and wake growth.

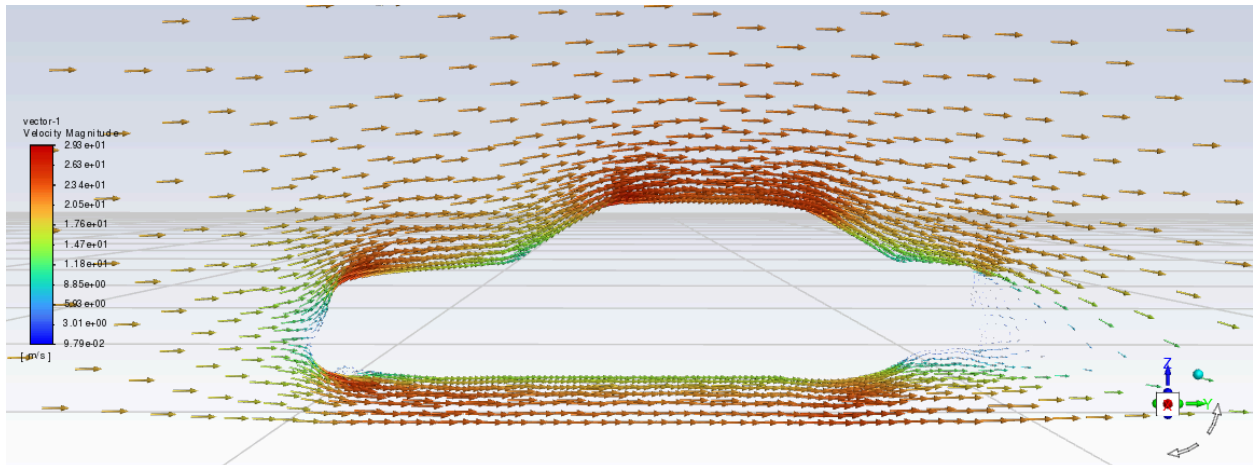


Figure #24 – Velocity vector field showing acceleration over the roof and underbody, and wake formation behind.

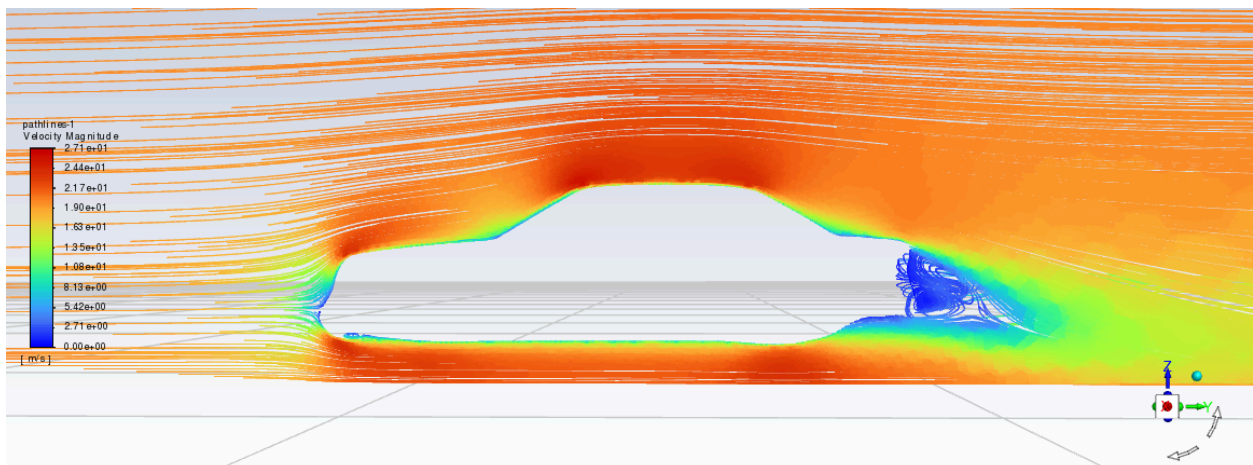


Figure #25 – Pathlines showing airflow attachment over the hood and roof, and separation behind the trunk.

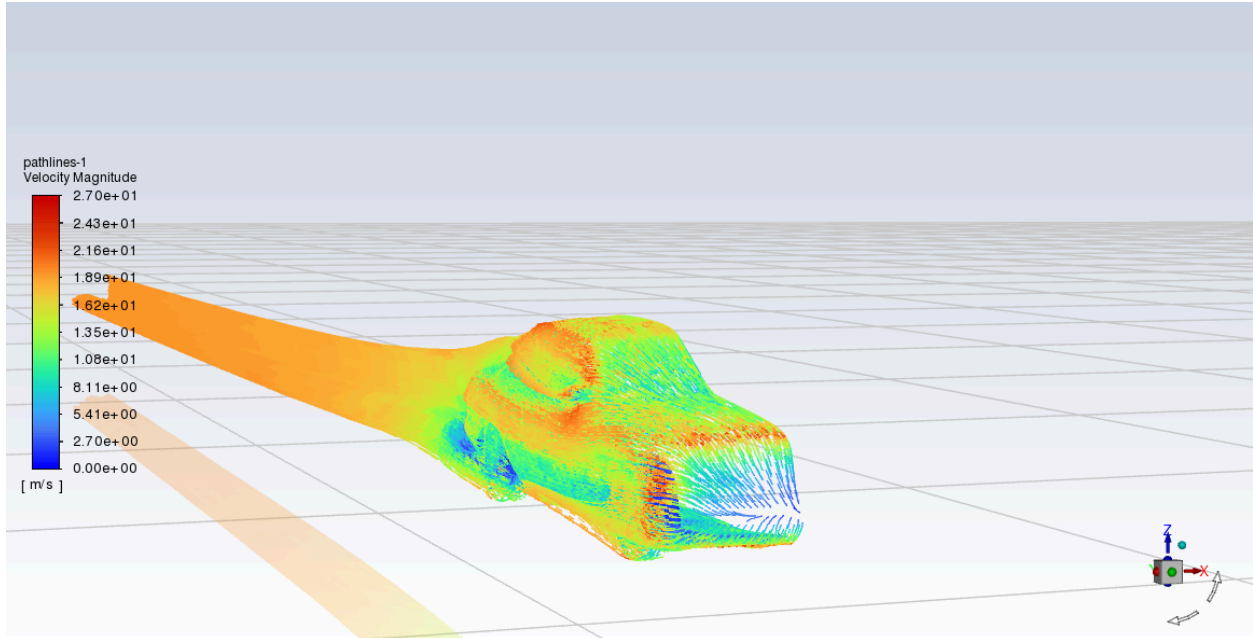


Figure #26 – 3D pathlines visualizing wake structure and stream behavior downstream of the car.

Forces - Direction Vector (0 1 0)						
Zone	Forces [N] Pressure	Viscous	Total	Coefficients Pressure	Viscous	Total
car	60.385439	5.9223086	66.307747	0.26590913	0.026079067	0.2919882
Net	60.385439	5.9223086	66.307747	0.26590913	0.026079067	0.2919882

Table #3 – Modified car: computed drag coefficient and force breakdown (pressure and viscous components).

Forces - Direction Vector (0 0 1)						
Zone	Forces [N] Pressure	Viscous	Total	Coefficients Pressure	Viscous	Total
car	-10.986706	0.40697164	-10.579734	-0.048380295	0.0017921121	-0.046588183
Net	-10.986706	0.40697164	-10.579734	-0.048380295	0.0017921121	-0.046588183

Table #4 – Modified car: computed lift coefficient and vertical force distribution.

Forces - Direction Vector (0 1 0)						
Zone	Forces [N] Pressure	Viscous	Total	Coefficients Pressure	Viscous	Total
car	59.337955	6.8756222	66.213577	0.26231531	0.030395065	0.29271037
Net	59.337955	6.8756222	66.213577	0.26231531	0.030395065	0.29271037

Table #5 – Stock car: computed drag coefficient and force breakdown (pressure and viscous components).

Forces - Direction Vector (0 0 1)						
Zone	Forces [N] Pressure	Viscous	Total	Coefficients Pressure	Viscous	Total
car	-7.038426	0.32486976	-6.7135562	-0.031114772	0.0014361518	-0.02967862
Net	-7.038426	0.32486976	-6.7135562	-0.031114772	0.0014361518	-0.02967862

Table #6 – Modified car: computed lift coefficient and vertical force distribution.

IV. Conclusion

This project used ANSYS Fluent to perform a comparative CFD study of a 1995 Mercedes-Benz E320 with a stock front bumper and a modified bumper design. A highly detailed 3D model was cleaned up through shrink-wrapping, smoothing, auto skin, and simplified wheel construction, then cut into a half-car configuration and placed inside a wind-tunnel-style enclosure. Both geometries were meshed using the same watertight workflow and solved with a steady RANS formulation using the standard $k-\epsilon$ turbulence model.

The modified bumper produced a drag coefficient of 0.29199 compared to 0.29271 for the stock bumper, with a slightly more negative lift coefficient. The relative drag difference of roughly 0.25% is extremely small and likely within the numerical uncertainty associated with the mesh and modeling approach. In practical terms, the redesign appears aerodynamically neutral: it does not significantly hurt the car's drag performance, but it also does not deliver a major improvement.

Despite these limitations, the project demonstrates how CFD can be used to visualize flow structures, explore “what-if” design changes, and quantify their impact on key performance metrics. The work also highlights the importance of clean geometry, consistent preprocessing across cases, and realistic expectations when using moderate mesh densities and simplified physics.

Future work could focus on refining the simulation fidelity. This would include increasing the mesh size into the half-million to multi-million cell range, using a full-car model including underbody flow, resolving the boundary layer with smaller first-cell heights, switching to the $k-\omega$ SST turbulence model, modeling a moving ground and rotating wheels, and ideally comparing results to experimental data or published drag coefficients for similar vehicles. Overall, the simulation proves that CFD is a powerful tool for visualizing airflow behavior and evaluating design decisions, even when starting with imperfect or simplified geometry.

V. References

- [1] Mercedes-Benz of Scottsdale, “Aerodynamics: Artistry and Science in Motion,” 2023. [Online]. Available: <https://www.mbscottsdale.com/blog/mercedes-benz-aerodynamics-artistry-and-science-in-motion/amp/>
- [2] J. D. Anderson, Fundamentals of Aerodynamics, 6th ed. New York, NY, USA: McGraw-Hill, 2017.
- [3] ANSYS Inc., ANSYS Fluent Theory Guide, Release 2023 R1, 2023.
- [4] H. Schlichting and K. Gersten, Boundary Layer Theory, 9th ed. Berlin, Germany: Springer, 2016.
- [5] H. K. Versteeg and W. Malalasekera, An Introduction to Computational Fluid Dynamics: The Finite Volume Method, 2nd ed. Harlow, U.K.: Pearson, 2007.
- [6] W. H. Hucho, “Aerodynamics of road vehicles,” Annu. Rev. Fluid Mech., vol. 25, pp. 485–537, 1993.
- [7] S. R. Ahmed, G. Ramm, and G. Faltn, “Some salient features of the time-averaged ground vehicle wake,” SAE Tech. Paper 840300, 1984.
- [8] N. Salehin, N. Z. Saimoon, M. M. Ahmed, and M. Masud, “Passive flow control devices for road vehicles: A comprehensive review,” 2025.
- [9] Nieve5677, “Mercedes-Benz E-class (Mk2) (W124),” Sketchfab, 2023. [Online]. Available: <https://skfb.ly/pvLvX>