MAE 5230 CFD Design Report

Roman Trujillo

Daniyar Kushaliev
Maximilian Mendoza

Abstract

Different shapes of an electric car were explored to reduce the car's drag. We chose to modify the back, the front, and the roof of the car to reduce the drag. The final design of our car is similar to a mini-van. Even though the low-pressure region behind our design is larger than the initial car design, the reduction in drag from keeping flow as attached to the car as possible outweighs the increase in drag from the larger low-pressure region behind the car. Further refinement of our design, including a less pronounced nose and a smaller car width, further reduced our drag force, and our final drag decreased by $\sim 32\%$ of the initial design drag force. We believe that, in order to increase the range of an electric car, incoming flow must be as attached to the car for as much as possible.

1 Introduction

This project examines the effects of drag on an electric car and explores ways to minimize it. With carbon emissions from humans reaching new heights, it is important to address these factors in order to mitigate climate change. The transportation sector consumes the most energy in the United States by far, with estimates being around 28% of all energy consumption. This displays the issue with gas-driven cars with regards to sustainability. On the other hand, "Electric Vehicle Outlook 2019" predicts that 30% of all cars in the world will be electric by 2040, and predicts that EV sales will be \$10 million by 2025, \$28 million by 2030, and \$56 million by 2040[1]. This trend clearly shows how important and relevant electric cars are becoming in the modern landscape and provides strong motivation for their exploration and optimization.

The background and motivation leads well into the overall design goals of this simulation. Mainly, they are to reduce the drag force on the electric car via three differing methods. Reducing the drag is important to elongate the distance EV cars can travel, and it could even lessen the gas needed to travel for hybrid options. In general, reducing the drag on the car increases its travel distance between charges, which can make the electric option even more competitive with gas car alternatives. The three altering methods of reducing the drag force will be discussed more thoroughly in the design section of the report.

Looking at the overall design goals, it is also important to recognize the constraints and limited scope of this specific simulation. The task is to take a pre-planned CAD design into ANSYS Fluent in order to evaluate the changes of the design on the drag force. The design constraints themselves are as follows: the length of the car be between 3 and 6 meters, height from the car floor to roof should be greater than or equal to 1.35 meters, width of the car should be between 1.5 and 2 meters, wheel base should be 3 meters, a track width greater than or equal to 1.3 meters, the wheel diameter and thickness are 0.45 meters and 0.175 meters respectively, height from the ground to car floor is to be 0.15 meters, and the model of the car is assumed to be symmetric in the width direction.

2 Mathematical Model

The model of inspection for the forces and velocities of flow around a car is the GEKO $k-\omega$ two equation turbulence model. This model involves six partial differential equations (one from mass conservation, three from momentum conservation in the three coordinate directions, and two from conservation equations for k and ω) and one algebraic equation relating to the turbulent stresses on the vehicle. There are seven unknown scalar functions to be solved for: $u(x, y, z)$, $v(x, y, z)$, $w(x, y, z)$, $P(x, y, z)$, $\mu(x, y, z)$, $k(x, y, z)$, and $\omega(x, y, z)$. This means that our system is closed and can be evaluated by the simulator. Some important assumptions that go into this governing model are that the fluid is incompressible and Newtonian and that the flow is stationary turbulent flow. The boundary conditions for the flow domain are as follows: $\vec{V}_{\text{inlet}} = u_0\widehat{\boldsymbol{i}} + 0\widehat{\boldsymbol{j}} + 0\hat{k}$, $P_{outlet} = P_{atm}$, with the no slip boundary condition $\vec{V} = 0\widehat{\boldsymbol{i}} + 0\widehat{\boldsymbol{j}} + 0\hat{k}$ being placed on the car body and the ground. Additionally, there will be a symmetry plane splitting the car body to simplify the problem, meaning that any derivatives of values crossing the plane will be zero.

[1] R.J. Barthelmie, "Transport", Lecture Series, Cornell University, 2022

$$\textit{Conservation of Mass (Continuity Equation): } \nabla \cdot \overrightarrow{\overrightarrow{V}} = 0$$

$$\textit{Conservation of Momentum: } \rho(\nabla \cdot \overrightarrow{\overrightarrow{V}})\overrightarrow{\overrightarrow{V}} = -\nabla \overline{p} + \nabla \cdot [(\mu + \mu_t)(\nabla \overrightarrow{\overrightarrow{V}} + \nabla \overrightarrow{\overrightarrow{V}}^T)]$$

$$\textit{Algebraic Relation Between } k \textit{ and } \omega\textit{: } \mu_t = \frac{\rho k}{\omega}$$

3 Numerical Solution Strategy

It is too complicated to solve the partial differential equations outlined in section 2 for the whole domain at once, so a numerical solution strategy must be implemented. First, the domain mesh will be split into subsections called cells. The solver will then take the boundary conditions to solve for cell center values by linearizing the conservation equations and then interpolate the cell center values to solve for values elsewhere in the domain. This process will be repeated iteratively, where the conservation of mass will be calculated at each iteration to analyze the quality of the solution until a specified threshold is reached. Additionally, an approximation must be adopted to efficiently solve for the effects of turbulence on the system. The turbulent fluctuations (u') will be separated from the time averaged velocity ($\bar{u}$), such that the following is true $u = \bar{u} + u'$. Applying this approximation to the conservation equations is what results in the k and ω terms arising from the nonlinear fluctuations.

The domain chosen for this simulation is 10 times the length of the car for all of the dimensions of the rectangular prism except for the front and back dimensions along the flow, which are boosted to 15 times the length of the car to assure detailed results along the flow path around the car. The boundary conditions for this model are free stream conditions for the top and side boundaries of the rectangle. The inlet flow side is set with a velocity, turbulent intensity, and turbulent viscosity ratio, with the latter two being used to determine the initial values of k and ω. Additionally ground and where the car surface meets the fluid air are wall boundary conditions with no slip applied, but due to the fact the simulation is in the car's reference frame, the ground must be given a velocity equal to that of the car to properly simulate the flow conditions.

3.1 Limitations of CFD Approach

There are a few of limitations to the CFD approach.

1. The RANS approach utilizes computational power more efficiently than other methods such as Large Eddy Simulation and Direct Numerical Simulation, but in taking the time average of the flow, the small scale fluctuations are lost. However, the results of the RANS approach usually provide enough information about the flow to guide design decisions.

2. The process by which the domain is separated into cells results in a discretization error arising. The effect of this error is mitigated by refining the mesh, though there is a practical limit to this solution set by the available computational power, so this error will always be present in one way or another.

3. Similarly, the linearization of the conservation equations to solve for the cell center values results in the presence of a linearization error. This error is driven to zero by increasing the number of iterations. Again, there is a hard limit on this solution due to the increased computational power required by the increased number of iterations.

4 Design

4.1 Initial Design

The initial design of the electric car is shown below, following the guidelines of Alain Alarco[2].

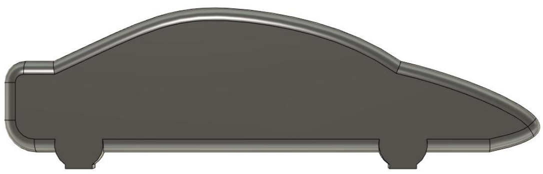
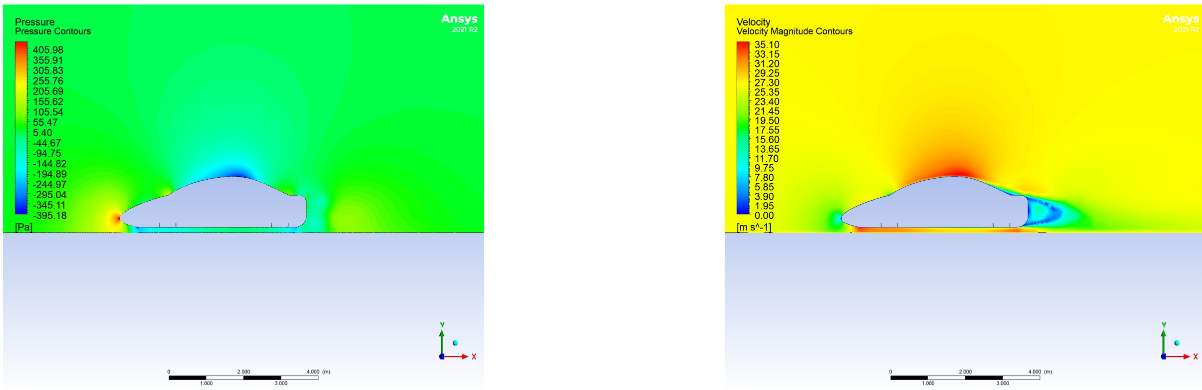


Figure 1: Initial car body design

As shown in the numerical results (figure 2b), a large re-circulation region is developed behind the car. The lower pressure behind the car (figure 2a) creates significant pressure drag force. What is also interesting is the region of separation that occurs after the top of the car roof. The separation region is of lower pressure than the incoming flow, thus, the separation region also adds to the total drag pressure felt by the car.

Instead of looking at the drag coefficient, we chose to instead look at the drag force. Fluent calculates coefficients using reference values for area, length, velocity, etc. Since we changed the car design, the reference values for each design would be different. Using the drag forces will give us a better understanding of how the various designs affect the flow. Out initial car design drag force is 176.51 N.

[2]https://confluence.cornell.edu/pages/viewpage.action?pageId=395473552



(a) Initial design pressure contours

(b) Basic design velocity contours

Figure 2: Initial design numerical results

For verification and validation, one of the first steps to take (the "low-hanging fruit") is to calculate the total mass flux of the interior. This way, at least, the total mass of the problem has been numerically balanced. Afterwards, check momentum balance. Fluent cannot check the momentum balance of the interior, and for interest of time, we did not check momentum balance manually. Another step to take is to increase the mesh (again, we did not do this for interest of time). For this, the number of cells near the car body would have to be increased. For turbulent flows, the boundary layer near walls is very thin. Thus, the mesh near walls must be small enough and numerous enough that boundary layer effects are, perhaps not fully, but largely solved. If results agree with the previous coarser mesh, then the solution is robust enough to be useful.

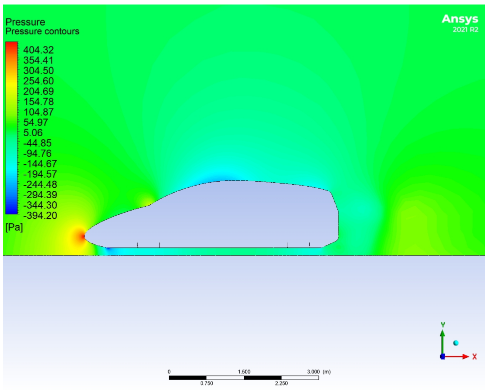
4.2 Design Exploration

To decrease the drag, various designs were explored to study their effects on flow and, ultimately, how much the drag would decrease. One of the designs focused on the back of the car, another design focused on the front of the car (the "hood"), and a third design focused on the roof of the car. However, the ultimate goal of our car was to help with the transport of people. Thus, our eventual car design resembles a mini-van.

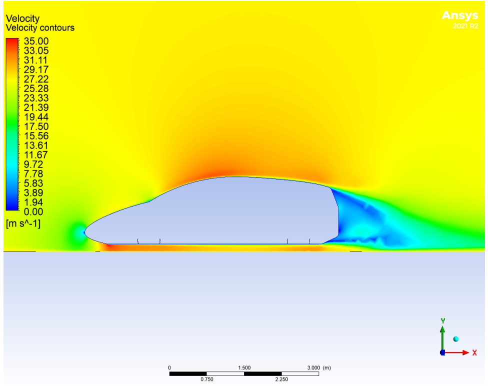
4.2.1 Back of Car

A fully aerodynamic car would have a gently-sloping back so that separation could not occur (a teardrop design), thus greatly reducing the pressure drag of the car. However, this design does not work realistically. If a family were to try and fit into the car, the teardrop design would mean the total length of the car would be much bigger than the allowed 6 m we were given. However, if the constraints were to be followed, then the car would not be able to fit even two people. The purpose of our design was to have as utilitarian of a car as possible.

With those constraints in mind, a design for the back of the car was made, following the general shape of a minivan. Results are shown below.



(a) Utilitarian car body pressure contours



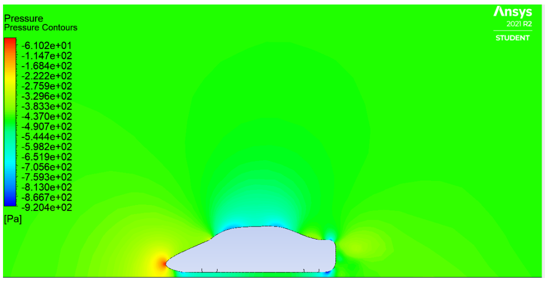
(b) Utilitarian car body velocity contours

Figure 3: Utilitarian car body numerical results

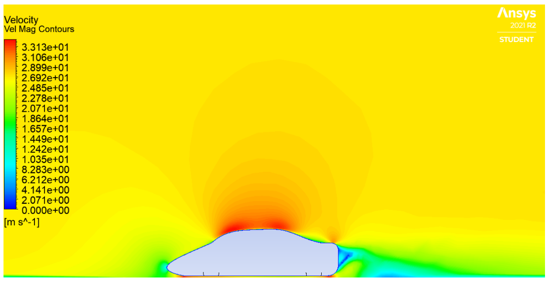
Because of the low amount of separation while the flow is still over the roof of the car, the total drag was 134.65 N. This is a great reduction compared to the original car design.

4.2.2 Slanted Hood

Another design explored is changing the front of the car. Again, the goal was to create as little flow separation as possible. With that in mind, a steeper hood was created to reduce the inertia effects of flow.



(a) Steeper hood pressure contours



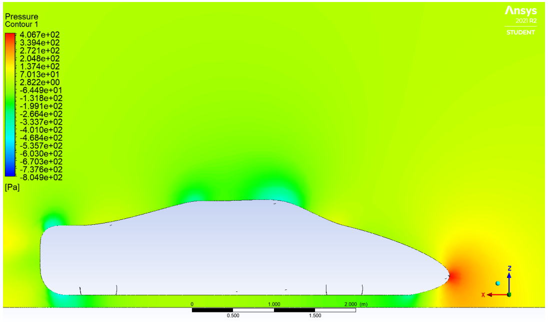
(b) Steeper hood velocity contours

Figure 4: Steeper hood numerical results

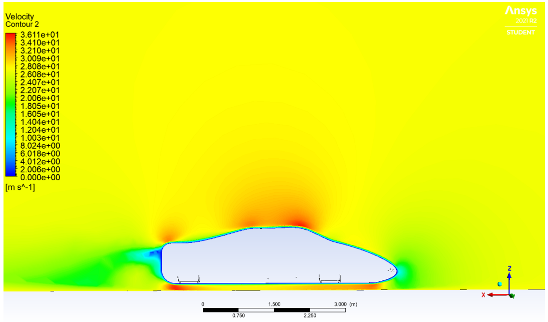
The drag force from this design is 146.61 N. We have had to change the roof of the car so that the car would be at the same height. That way, the drag force can be comparable to a car of the same height. In this instance, keeping the flow as attached to the car body as possible meant matching the "steepness" of the hood with the beginning part of the roof. The design may look unconventional, but numerically, the drag has decreased.

4.2.3 Flatter Roof

Again, going off of previous philosophy, a flatter roof of the car will also keep the flow attached to the body of the car.



(a) Flatter roof pressure contours



(b) Flatter roof velocity contours

Figure 5: Flatter roof numerical results

The drag force from this design is 169.00 N. One thing to note is that the roof design used is slightly illegal, as the car height is below limits. However, the philosophy is still the same. A much flatter roof than the original design will help keep flow attached to the car body.

5 Final Design and Conclusion

Eventually, all three designs were combined to create our final design. Upon further review, we refined the nose of the car. The drag force from our "refined" nose was very similar to the drag force from a more pronounced nose, so we did not feel the need to give results for the "unrefined" nose.

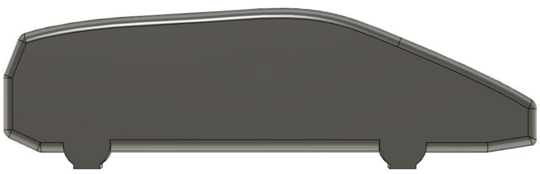
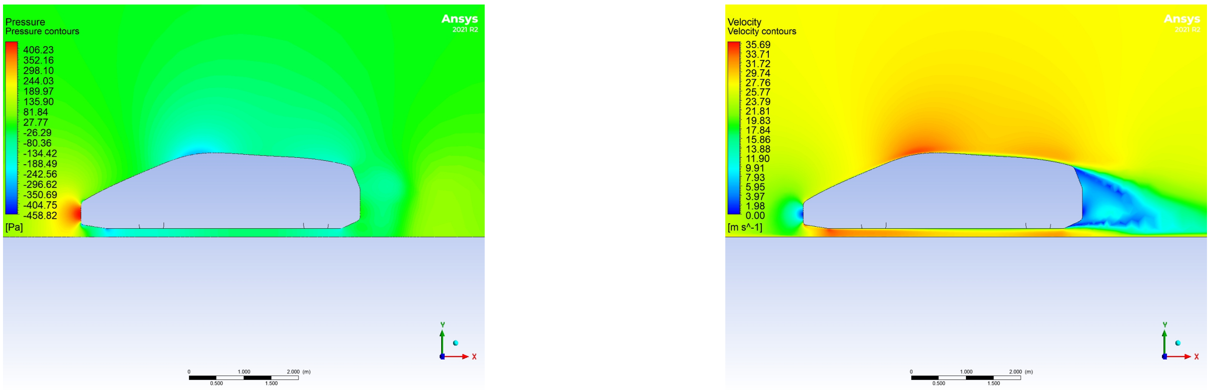


Figure 6: Our final car body design, resembling a mini-van.



(a) Final car body design pressure contours

(b) Final car body design pressure contours

Figure 7: Final car body design numerical results

The drag force is 135.63 N. The low-pressure and low-velocity region of the back of the car is bigger, however, the reduction in drag from keeping flow attached to the car outweighs the increased drag from the bigger low-pressure region at the back of the car.

Further refinement of the design was to decrease the width of the car, but still keep it within given specifications. The reduction did not significantly reduce the width of the car, as relatively non-obese people would fit inside it. The drag of the smaller car width was reduced to 120.13 N.

In the future, a more detailed work on the car body would be ideal. Eventually, taking inspiration from Bugatti, a "hollow" car shape could be tested. The car body, in this case, is not a solid mass that air must go around. Instead, there are various channels within the car body that force the flow to go through the car. Some could be used to cool various parts of the engine, but most of the channels would provide a way for incoming air to pass "through" the car, reducing the effective cross-section area of the car.

Although this project was not ground-breaking work, we still felt that important lessons about fluid dynamics were taught to us. We learned how powerful CFD can be, as well as the drawbacks of CFD in certain situations.

Appendix I: Comparison of Initial Design Results

The trends each of us obtained seem to be in alignment. The major differences between us were from the drag forces. Kushaliev obtained a drag force of 164.38 N, Mendoza's drag force is 182 N, and Trujillo's drag force is 183.16 N.

We were not sure exactly how this happened. However, when we changed the mesh of our simulations, the drag force on the car was ± 10 N. With that knowledge, we decided that "the" drag force on our initial design would be the average of our drag forces

$$\frac{182 + 164.38 + 183.16}{3} = 176.51 \quad [\mathrm{N}]$$

Appendix II: Team member contributions

Daniyar Kushaliev

Kushaliev's contribution was the exploration of designing the back of the car. He has also put together the final design of our car (see section 5). For the report, he has typed up sections 4 and 5.

Roman Trujillo

Trujillo's contribution was the exploration of the car roof. For the report, he has typed up sections 2 and 3.

Maximilian Mendoza

Mendoza's contribution was the exploration of the steeper hood. For the report, he has typed up sections 1, sections 2, and parts of section 3 (content that involves the domain and boundary conditions of the system).