Elway Fleming

Lead Aerodynamicist

Aerodynamics Case Study

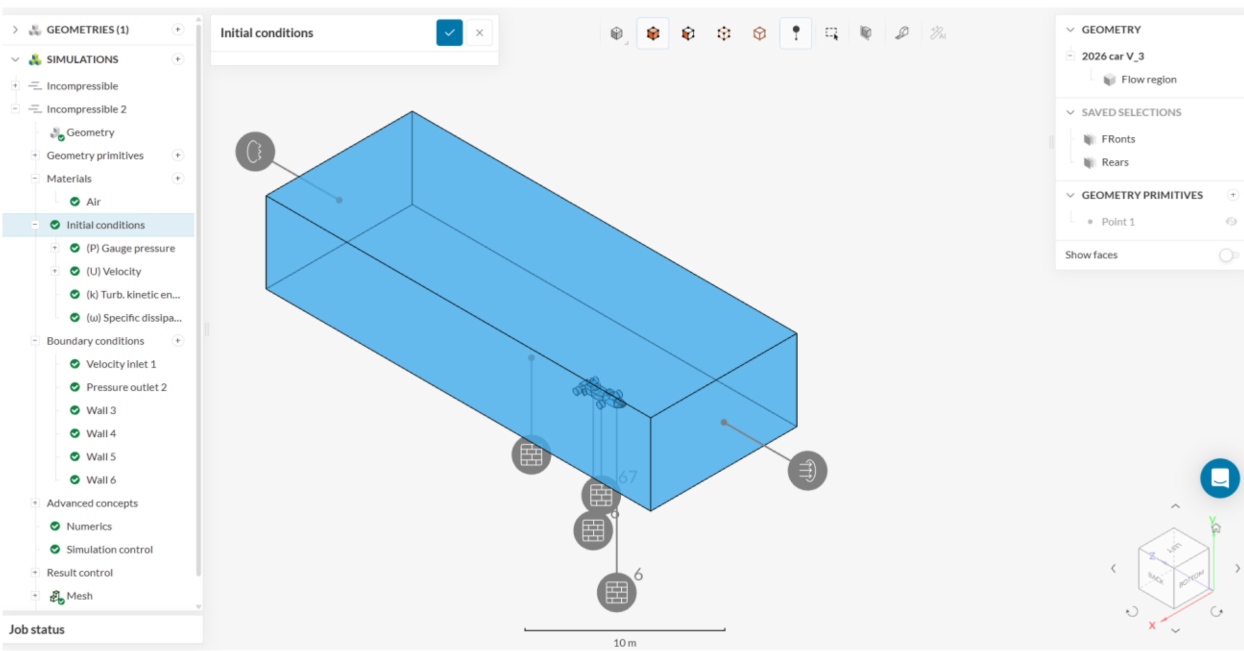
Red Raider Racing, 2026 FSAE EV Car

Texas Tech University

Computational Fluid Dynamics (CFD) is a powerful tool which allows us to analyze the aerodynamic effects on our designs, and can be used to guide our design process before real-world testing of our prototypes take place. In this project, we aim to gather valuable data and flow visualization on a 1:1 model of our 2026 FSAE EV car. We will be using SimScale, a cloud based CFD program to conduct these evaluations. We will be importing a simplified model of this years car, with a driver model in the cockpit. To help reduce simulation time and complexity, the cockpit will be filled in below the top of the body panels, and elements such as the suspension tower will be approximated using simple shapes such as blocks. This drastically cuts down on simulation time, as we are not concerned about how air flows through the individual springs and shocks, but rather general trends on the overall car. In this simulation, we are looking to obtain the following information:

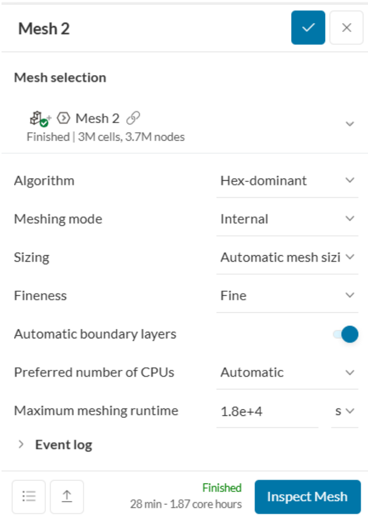
- Force Values (Lift, Drag, Yaw)
- Coefficients of Lift and Drag
- Center of Pressure
- Areas of High Turbulence
- Flow Visualization

In order to find these values, we are going to conduct a full car CFD. Normally it is applicable to cut the car in half to reduce computation load and time. However, due to the unsymmetrical design of our car, it is more accurate to run the full car and accept the time and resource penalty. We will be running an incompressible, steady-state simulation, as our car does not plan to exceed Mach 0.3 where shockwaves become a factor. For this simulation we will be running the car directly into the airstream as a baseline, although running the car at an angle can be a valuable tool to predict how airflow changed as the car enters, takes, and exits a corner. Our velocity will be at 13.411 m/s (roughly 30mph), as this is around the maximum speed achieved by formula cars during the endurance and autocross events. Although our car can go faster in a straight line, the track is designed to be technical enough to limit vehicle speed for safety, thus we will be using this speed to optimize our car around for the best performance. We will be running this simulation with a moving road, and spinning tires for the most accurate results. As our car is open-wheeled by nature, the turbulent flow around our exposed tires accounts for a large percentage of our drag, and understanding the flows at play will be crucial when it comes time to design our undertray for next year. We will be using the standard air density and viscous settings for air provided by SimScale. This makes our initial setup as such:

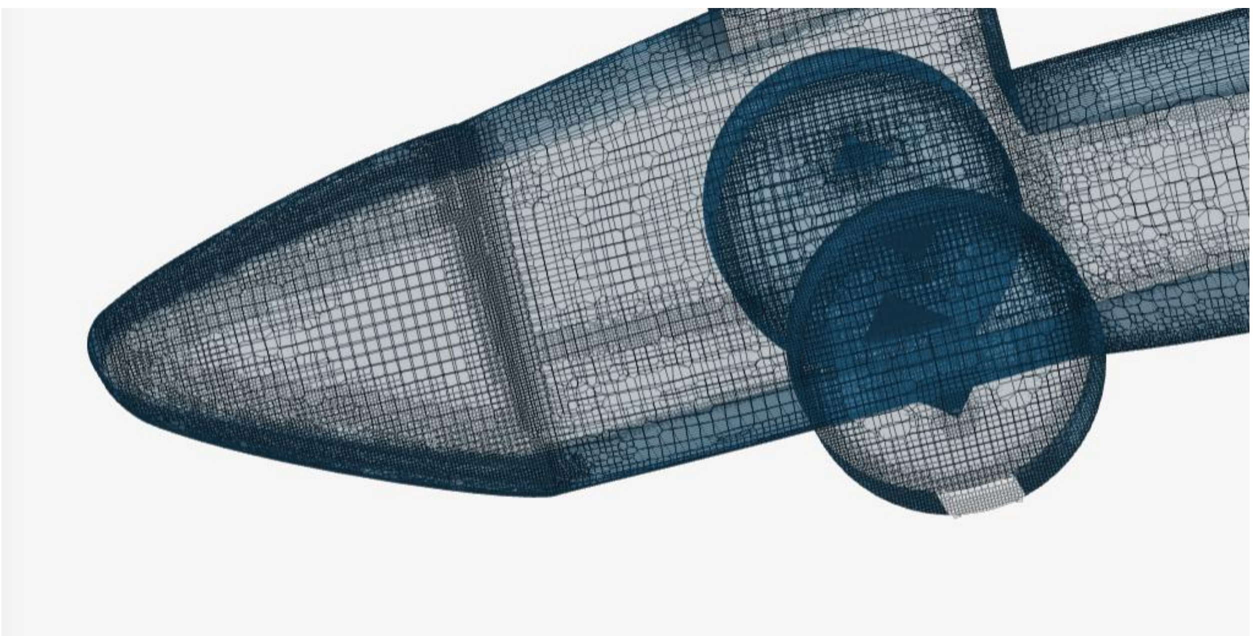


Now that we have our model imported, and our initial conditions set, we must setup our mesh. This is where historically our team has had problems with. Getting a good mesh is crucial to having a good result, as tiny imperfections in the mesh can change the geometry of your car, give false or inaccurate results, or even prevent the simulation from converging to an answer.

After several trial and error runs, we dialed our mesh settings down to the following:



These mesh settings give us a tight mesh, with little to no errors, and which tightly hugs our cars initial geometry:

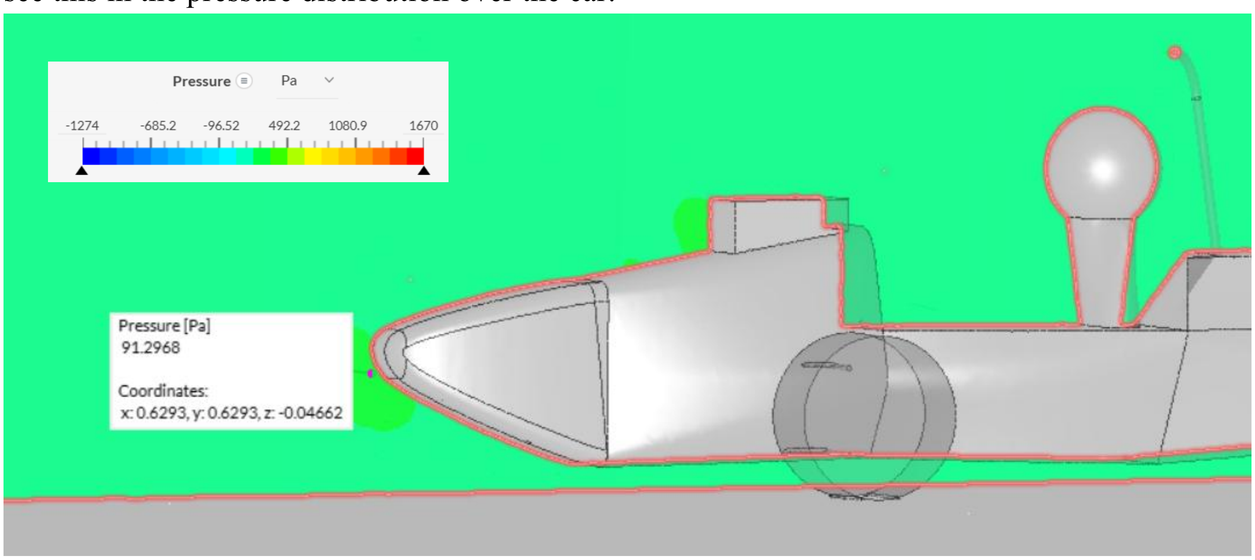


After we verify that our mesh is without any large faults, we can run the simulation. Our Simulation ran in 35 mins, consuming 18.54 core hours. From this simulation we were able to obtain the following data:

- Forces:
  - Lift: 11.30N
  - Drag: 56.26N
  - Yaw: -3.72N
- Coefficents:
  - Lift: 0.149
  - Drag: 0.753
    - Note: Frontal Area of 0.696 m^2 calculated from SolidWorks

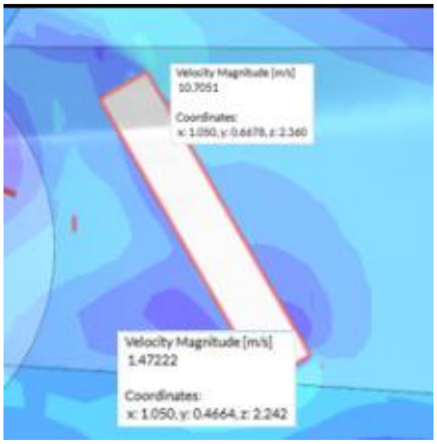
By using our moment values, and our coordinate system we can calculate where our center of pressure is, at z=1.366. We are going to work this as a two-dimensional problem now, as our x (yaw) forces are not in our interest for determining the percentage of force over each axle. If we simplify our car down to a two dimensional problem, now we have a simple statics problem where our lift, 11.3N, is concentrated at a point, 1.366, on a number line. We have two supports (tires), one at 1.02 and one at 2.67 (the z values for the center of our wheels). We can then calculate the load distribution, and find that 8.93N is concentrated over our front axle, and 2.37N over the rear. This nets us a balance of 79.03% over the front.

This heavy concentration over the front axle is coming from our nosecone, which was originally designed to house a front wing, which was later removed due to budget cuts. We can see this in the pressure distribution over the car:



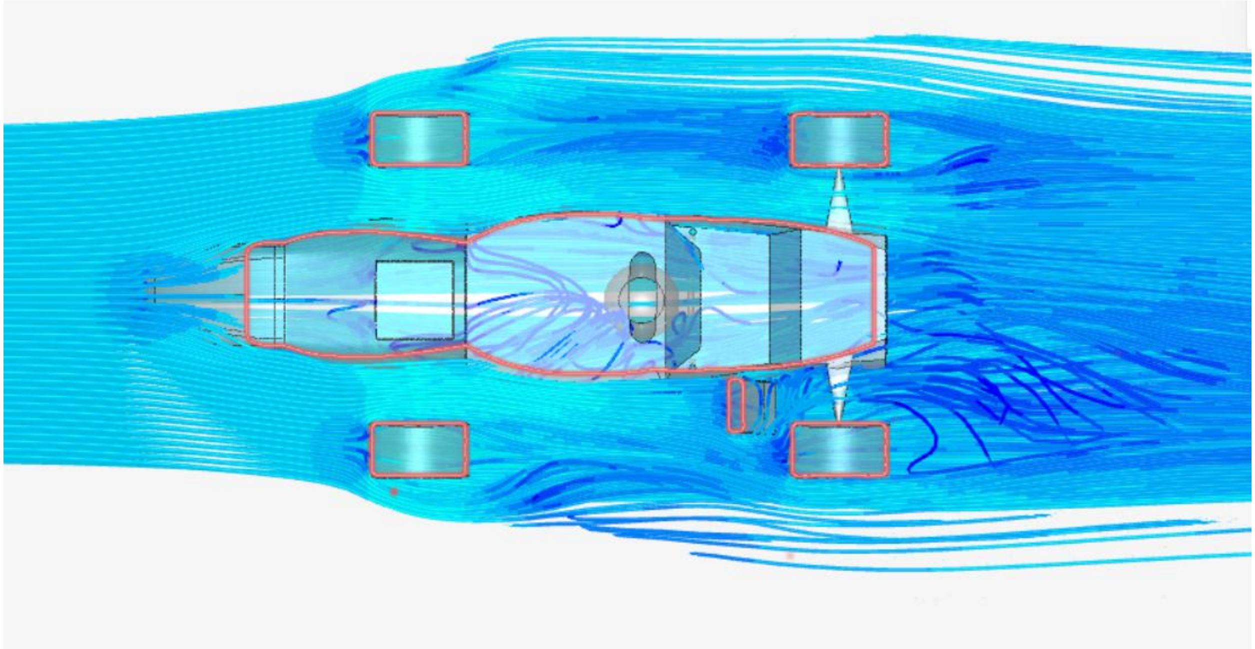
Given that the surrounding airflow has a pressure of 18.35 Pa, this patch of high pressure airflow underneath the long nosecone is providing the lions share of the lift which our car produces, and will directly contribute to understeer. These findings give me validation for the need of a front wing for next year's to either directly counter this lift, or an undertray with a highly tuned center of pressure to bring balance back to the car at high speeds.

By switching our views from pressure over to velocity, we can find our next concern. Airflow over our radiator is being impacted by the wake from our front tires, netting in an uneven velocity distribution over the radiator's bottom quarter.



This is a problem as our cooling system is already extremely minimized due to weight and fluid type restrictions. The use of a rear mounted fan to supplement this stagnant airflow on the bottom quarter may be necessary to prevent overheating on extended runs, such as those experienced during the endurance section of competition.

The final concern which we wish to address in this study is how the turbulent airflow coming off of the tires effects the car. We can use the partical trace feature to visualize how airflow is affected.



The introduction of the radiator on one side of the car appears to help direct front tire wake further around the rear tires. By having this asymmetrical setup we can compare the undisturbed airflow vs the airflow where the radiator sits. The airflow re-joining aft of the radiator is being rapidly funneled inside the rear tire. The entire setup seems to be very rapidly devolving into a mess of turbulence and vortices. A sidepod which would allow airflow to smoothly rejoin the surrounding flow would most likely improve our drag coefficient in a meaningful way. Another takeaway is how the airflow inside of the front tires behave. This simulation can help us locate where to get the best airflow for an undertray, and where to start funneling turbulent air into outwash. The use of bargeboards would be an essential integration of any form of undertray/underbody downforce.

As we prepare for the SAE International FSAE competition in Brooklyn, MI this summer, the CFD results have been crucial in our understanding of the core aerodynamic principles at play. The next step would be to compare our simulations to real world data. We plan on doing this by comparing flow over the car using yarn to the simulations observed flow. We can also use pressure patches places around the car to see localized pressure data, and compare that to SimScale results to further enhance our understanding of how CFD differs from the real world, and where we should actually be looking for correlations.

I would like to take this opportunity to personally thank the people over at SimScale for helping drive us. This is my first year as an Aerodynamics Lead, and my Sophomore year here at Texas Tech. Being able to take these complex topics and behaviors and present them in such a user friendly manner has greatly helped me understand and plan for next year. I look forward to writing another case study on the design of our underbody, and correlation to real world results here very soon!

-Elway Fleming