

VEHICLE ANALYSIS

CONTENTS

Technical Summary	2
Sketches	3
CFD Modelling	5
CAD Model	5
Cut Plots: Pressure and Velocity	6
Surface Plot: Pressure	6
Flow Trajectories	7
Calculations	8
Estimate C_d	8
CFD Values	8
Wind Tunnel	9
Discrepancy	10
Comparison	10
Next Steps	11

TECHNICAL SUMMARY

The car I chose to model is a heavily-modified Lamborghini Huracan - I started out by sketching the outline of a Huracan, before modifying its bodywork extensively. I then imported the four elevation-sketches I made (top, side, front, back) into SolidWorks as canvases to work from. Drawing the same line on two of these faces, I could then project the line between the two. Repeating this for every line on the car, I then lofted between these to create the final shapes before knitting them into a solid body.

I have made slight modifications since the previous iteration: I filleted all the corners for a more rounded overall shape. Short of rebuilding the whole car to have a lower frontal area and a less vertical rear, I hoped this would be the next best thing to reduce the drag coefficient - the ground clearance was already fairly low.

What follows is a documentation of the whole process, from sketches to CFD, and a comparison of CFD data against wind tunnel data. The final conclusion of my CFD analysis was that this car would need to exert a force of **581 N** to overcome drag at 70 mph. As power $P = Fv$, at a speed of 31.3 m s^{-1} this is a power of **18.2 kW**.

SKETCHES



Early concept sketches for my idea - I did these using a drawing app on a tablet.

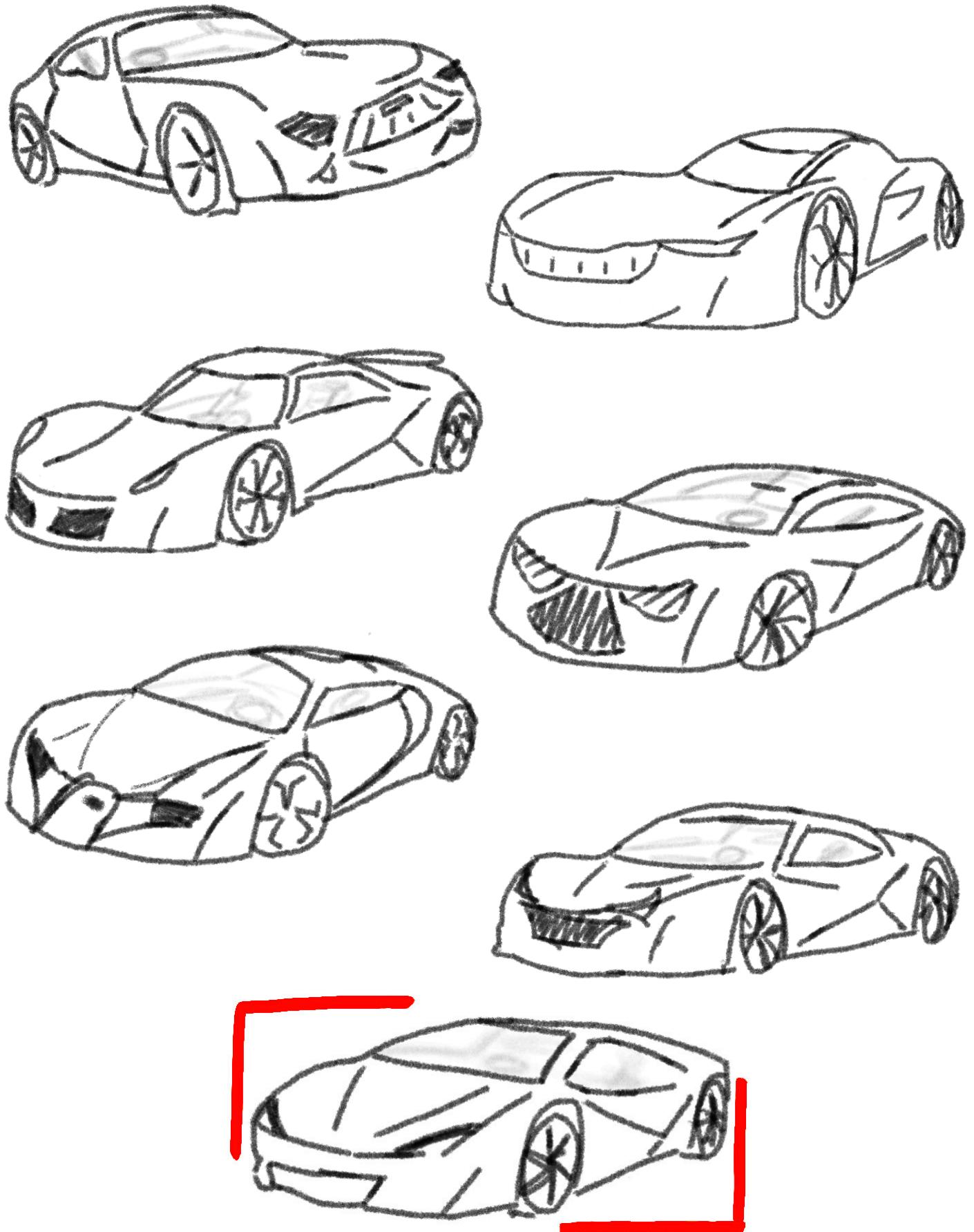
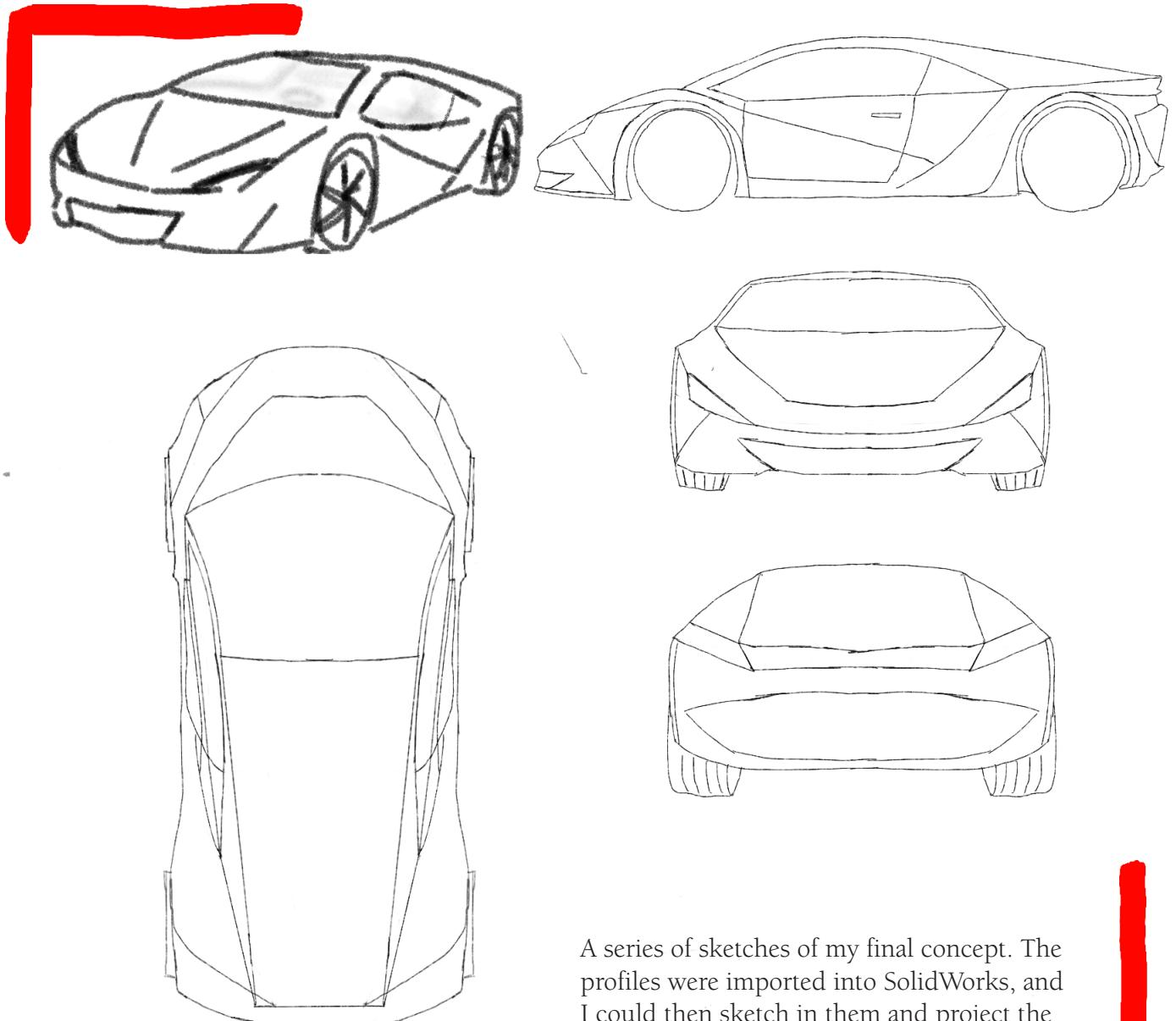
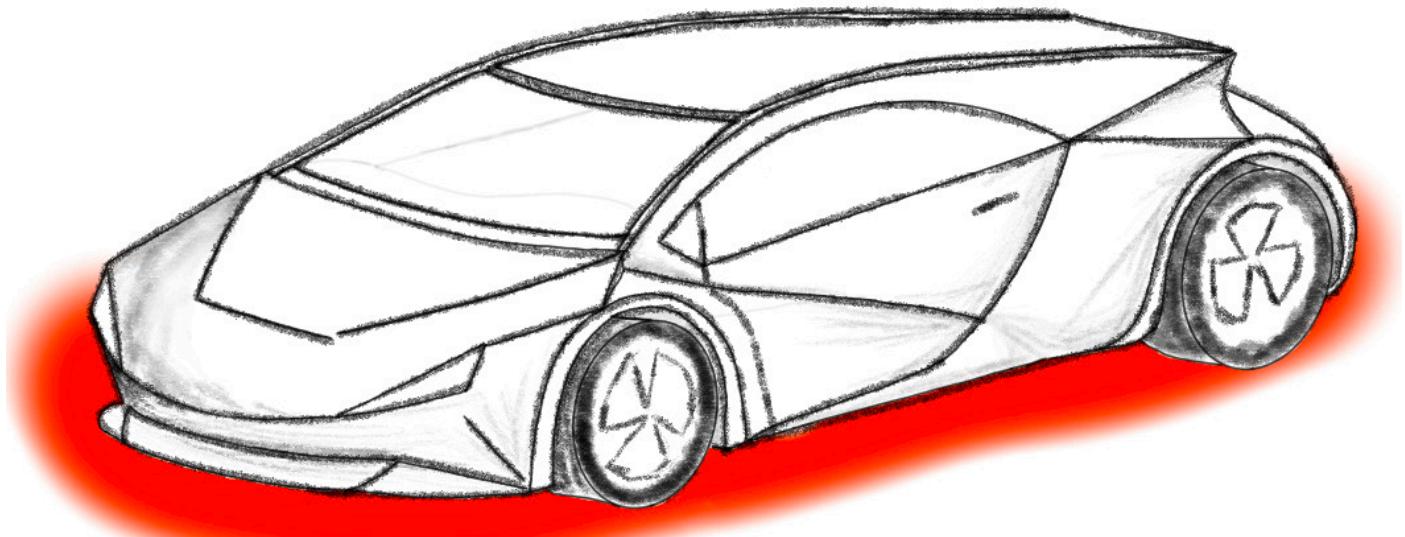


Figure 1: Initial sketches.



A series of sketches of my final concept. The profiles were imported into SolidWorks, and I could then sketch in them and project the curves into 3D space before lofting between them.



Figures 2 & 3: Final concept sketches.

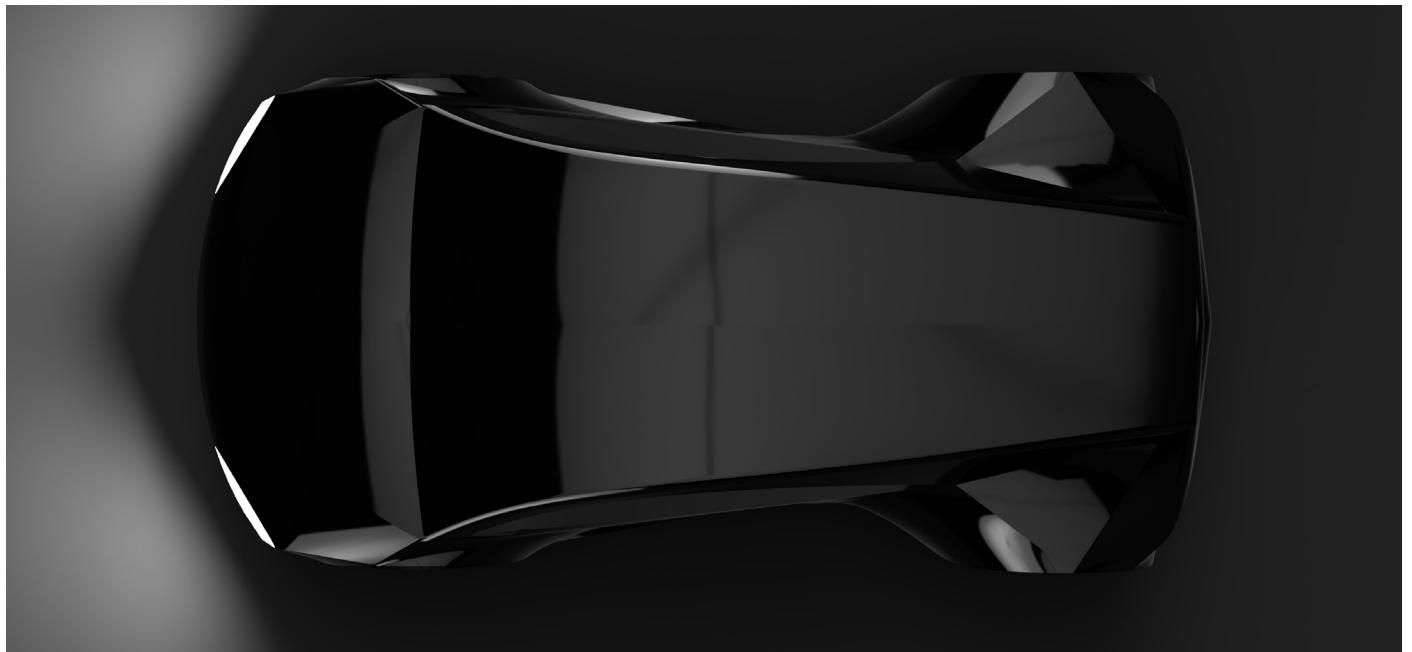
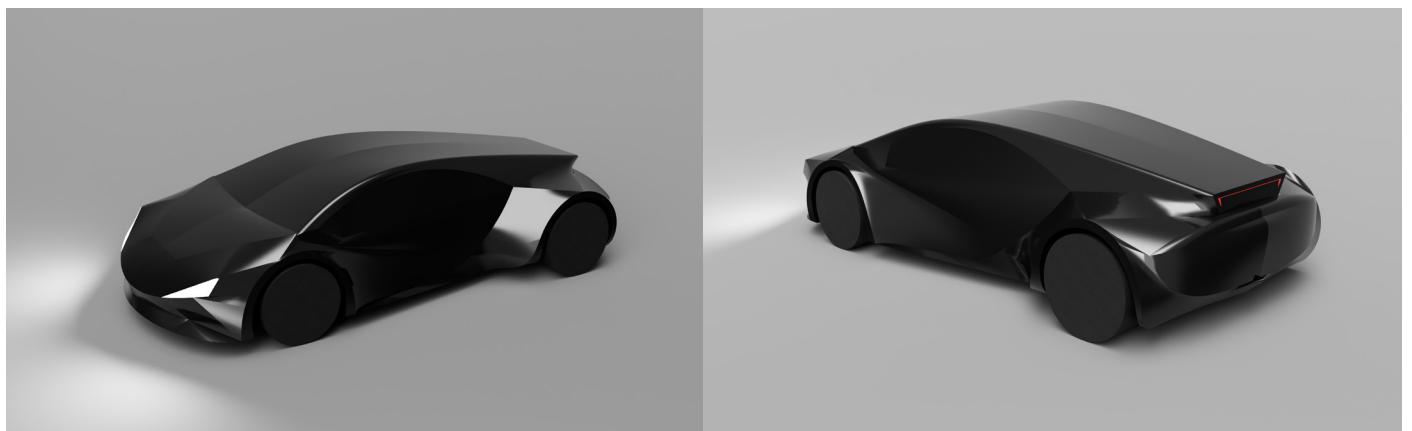
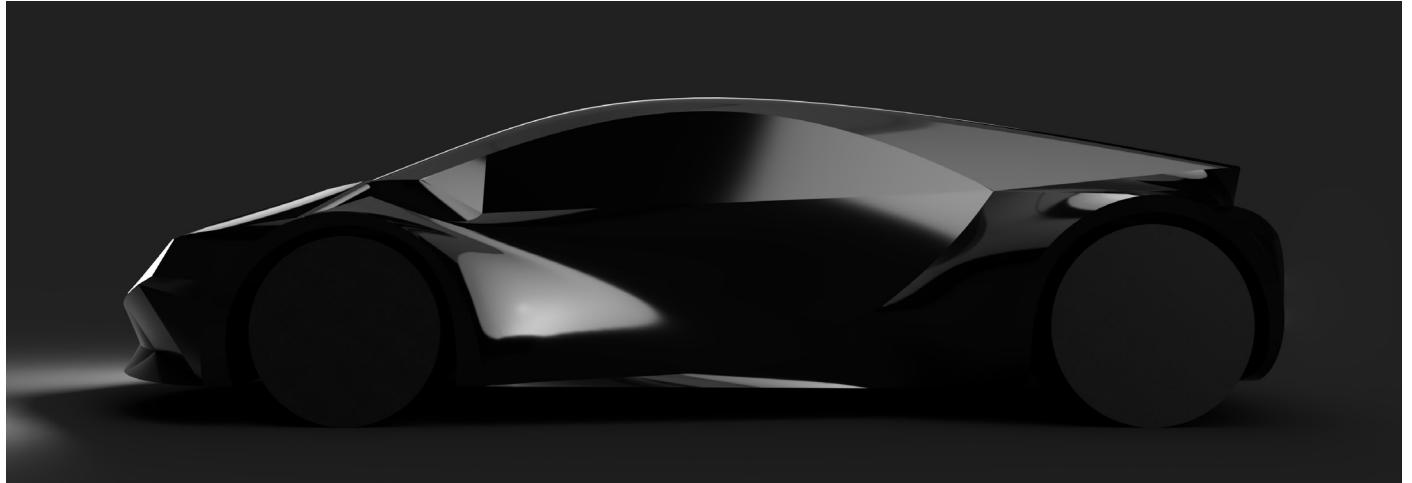
CFD MODELLING



My CFD modelling went as follows:

CAD Model

I enjoy CAD modelling and rendering, so I decided to import my SolidWorks model into Fusion to render it.



Figures 4-7: Final renders.

Cut Plots: Pressure and Velocity

Once I had run the flow simulation, I generated these cut plots, for pressure (top) and velocity.

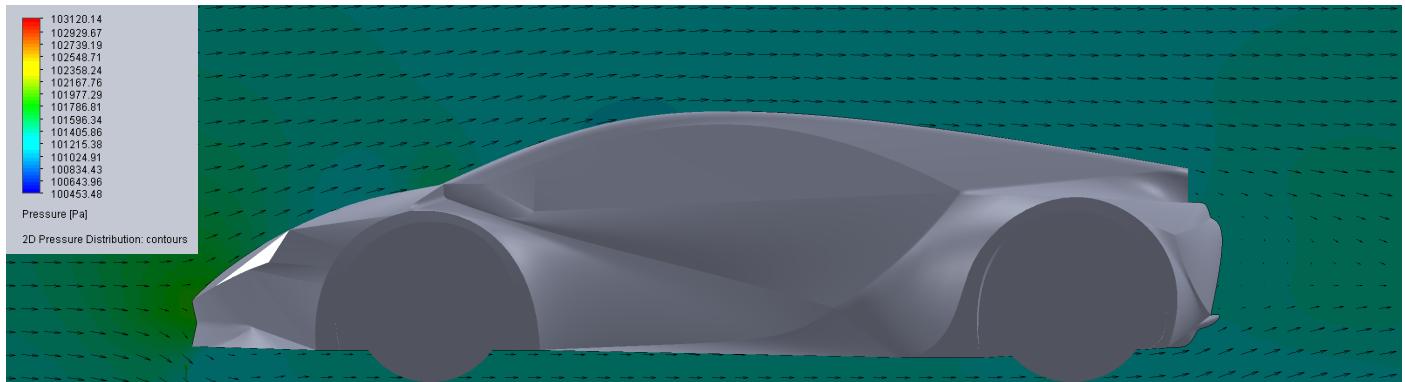


Figure 8: A cut plot showing **pressure** along the car, in Pa.

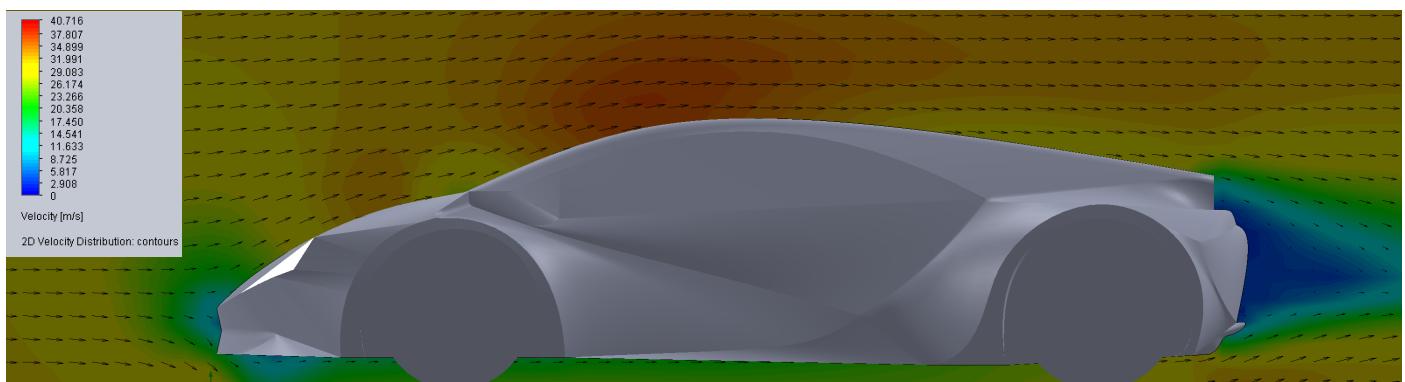


Figure 9: A cut plot showing wind **velocity** along the car, in $m s^{-1}$. There is a significant ‘dead zone,’ contributing to drag, behind the car.

Note: The hardware I am using (ICL Library computers) do not have the necessary graphics to process streamlines. Instead, I have used Vectors, which shows similar information. Vectors in both are for velocity.

Surface Plot: Pressure

The surface pressure plot shows that the pressure on the car does not get particularly high in any areas. The significant pressure drop above the rear wheels is worrying - it suggests flow delamination.

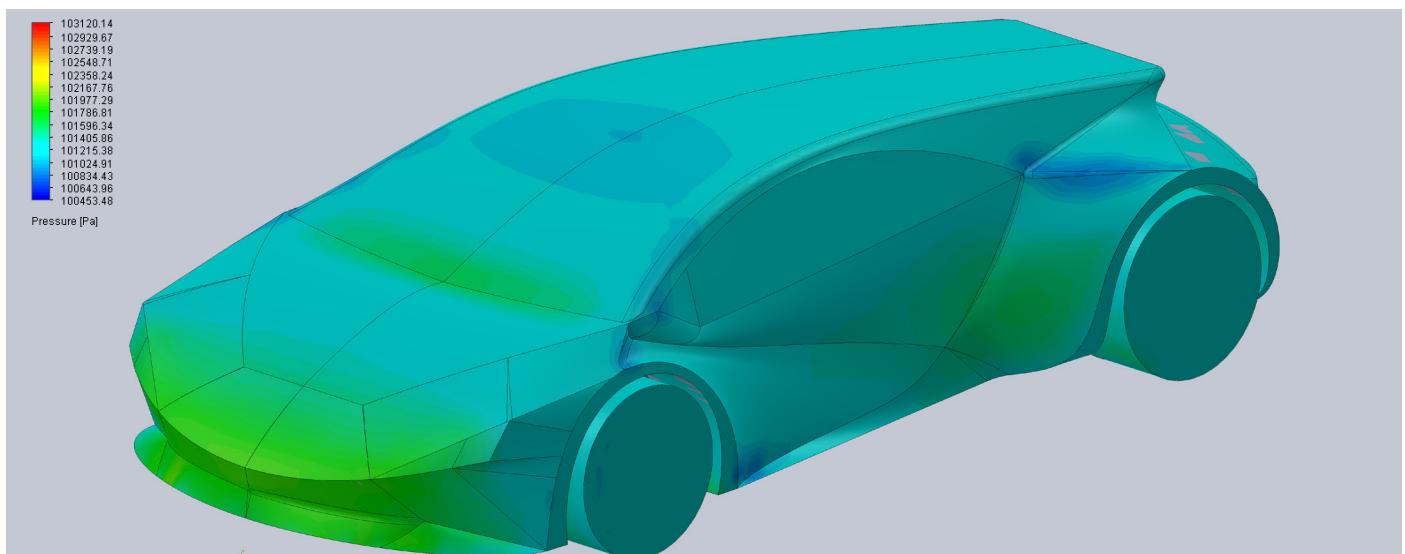


Figure 10: A surface plot showing **pressure distribution** over the front of the car, in Pa.

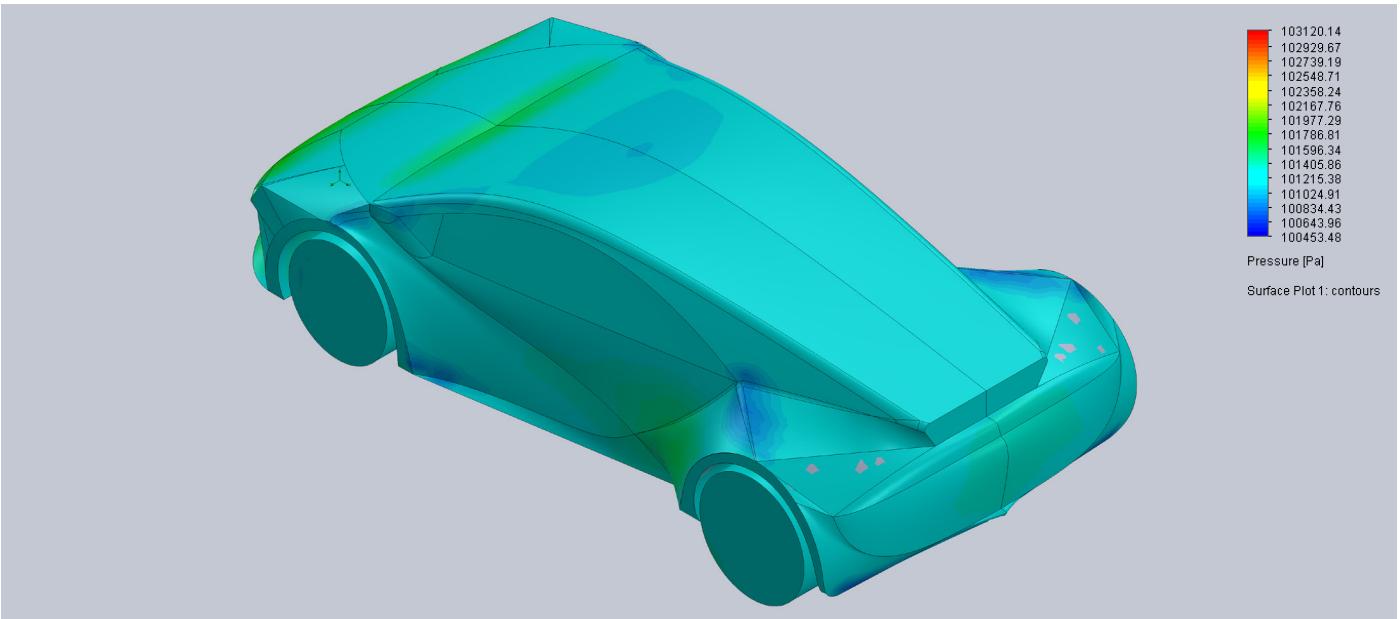


Figure 11: A surface plot showing **pressure distribution** over the rear of the car, in Pa .

Flow Trajectories

These flow trajectories were then generated from the CFD data:

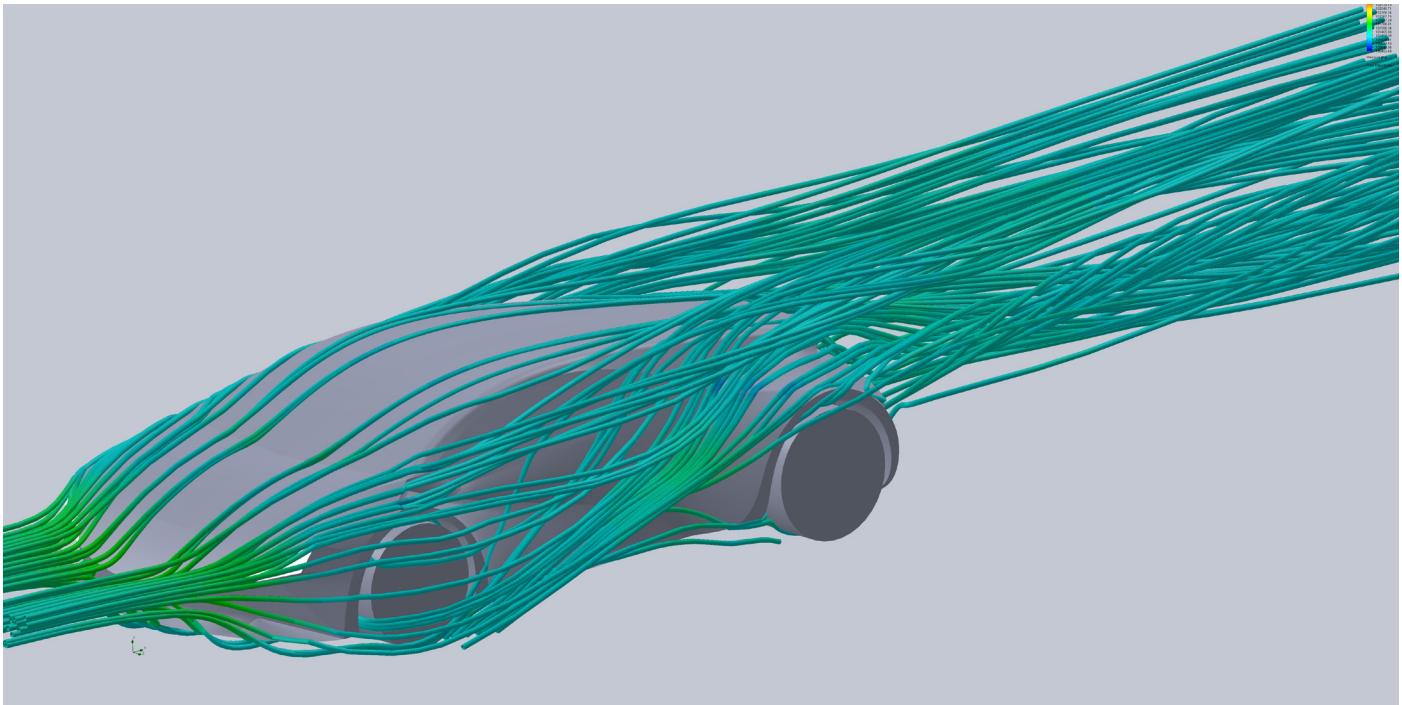


Figure 12: **Flow trajectories** over the car. Pipes are coloured by **velocity**, in m s^{-1} .

This is far more turbulent than hoped. The downforce can easily be visualised (all the streams are moving upwards behind the car), but there are several vortices behind the car (better visible on Fig. 13. below). This may well be because of how far the rear wheel arches stick out: they force air over the car, rather than around it, so the air must fall to fill the low pressure area behind the car.

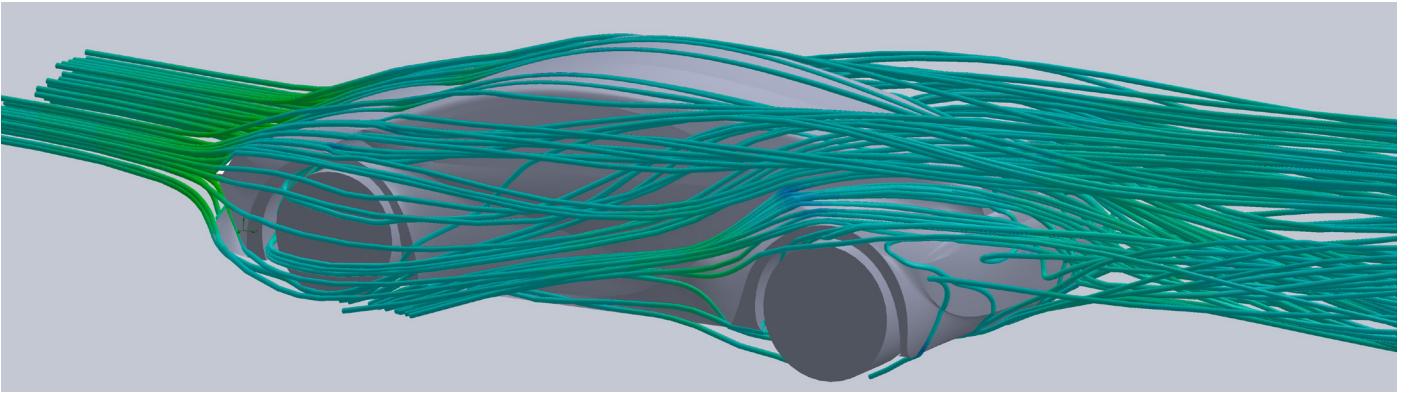


Figure 13: **Flow trajectories** over the rear. Pipes are coloured by **velocity**, in $m s^{-1}$. Note the messy streams.

CALCULATIONS

At this stage, I had accurate values for drag, lift and drag coefficient C_d from the CFD analysis. Before I look at them, I will calculate an estimate for C_d using the estimate tool.

Estimate C_d

From the estimator tool:

Front end (top): A-2 - partially rounded (2); Windshield: B-3 - curved at top and bottom (3); Roof: C-1 - narrows at back (2); Lower read end: D-3 - rear wheel arches stick out (3); Front end (side): E-3 - curves down to a near-vertical nose (3); Windshield peak: F-1 - curved (1); Rear roof: G-3 - No boot, squared roof back (4); Cowl/fender cross section: H-2 - Hood flush with vertical-sided fenders (4); total = 22.

$$C_{d \text{ (estimate)}} = 0.16 + 0.0095 * \sum N_i$$

$$C_{d \text{ (estimate)}} = 0.16 + 0.0095 * 22$$

$$C_{d \text{ (estimate)}} = 0.37$$

$C_d = 2F / \rho v^2 A$
Frontal area $A = 2.2 m^2$, wind velocity $v = 31.3 m s^{-1}$,
air density $\rho = 1.2 kg m^{-3}$

$$\therefore F_{\text{(expected)}} = 481.3 N$$

CFD Values

The CFD tells us that the true values are:

Goal Name	Unit	Value	Avg. Value	Min. Value	Max. Value
Drag	N	581.0634968	582.2517478	579.9837058	585.1400085
Lift	N	377.64448	375.7902538	371.2317882	381.255219
Drag Coefficient	(n/a)	0.446731752	0.4476453	0.44590159	0.449865846

The drag coefficient is very high: this is probably because the car is very angular and has a nearly vertical rear - since the last submission, and in response to feedback, I have rounded many of the corners, but this has had little effect. In further development I would reduce the frontal area of the car to reduce C_d .

While the $C_{d \text{ (estimate)}}$ seems to be within reasonable bounds, it gives an enormously different drag force F - this is probably because the estimator assumes a relatively smooth, rounded car, and my design is very angular.

$D = 581.06 N$, and as $P=Fv$, this means that the power needed to maintain $v = 31.3 m s^{-1}$ would = $31.3 m s^{-1} * 581 N \approx 18.2 kW$.

WIND TUNNEL

The data found from a wind tunnel analysis of an object are not the same as those found from a CFD analysis. Here I will compare the CFD and wind tunnel datasets from another supercar. I chose this model of those available as it is the most similar in shape to mine.

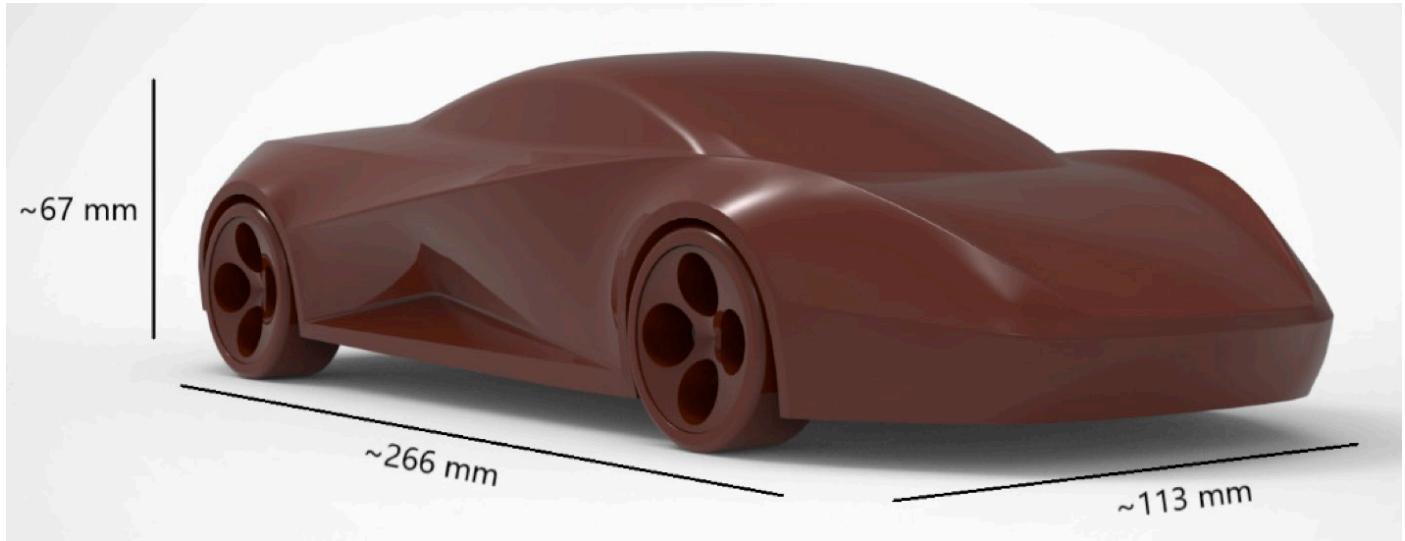


Figure 14: the SolidWorks model of the car tested in CFD.

Below are the results for the CFD-derived values compared to those from the wind-tunnel testing. They are sign-adjusted, i.e. positive is the same direction for all values.

	Unit	CFD Value	Wind Tunnel Value
Wind Velocity	m s^{-1}	30.833	30.552
Lift Force	N	0.983	- -
Drag Force	N	0.966	1.316
C_d	(n/a)	0.272	0.37

As can be seen, very similar wind speed values ($v = 30.692 \pm 0.5\%$) give noticeably different coefficients of drag ($C_d = 0.321 \pm 15\%$).

Both used a model a smaller scale, so the error is not due to this - both have a frontal area of 0.0061 m^2 .

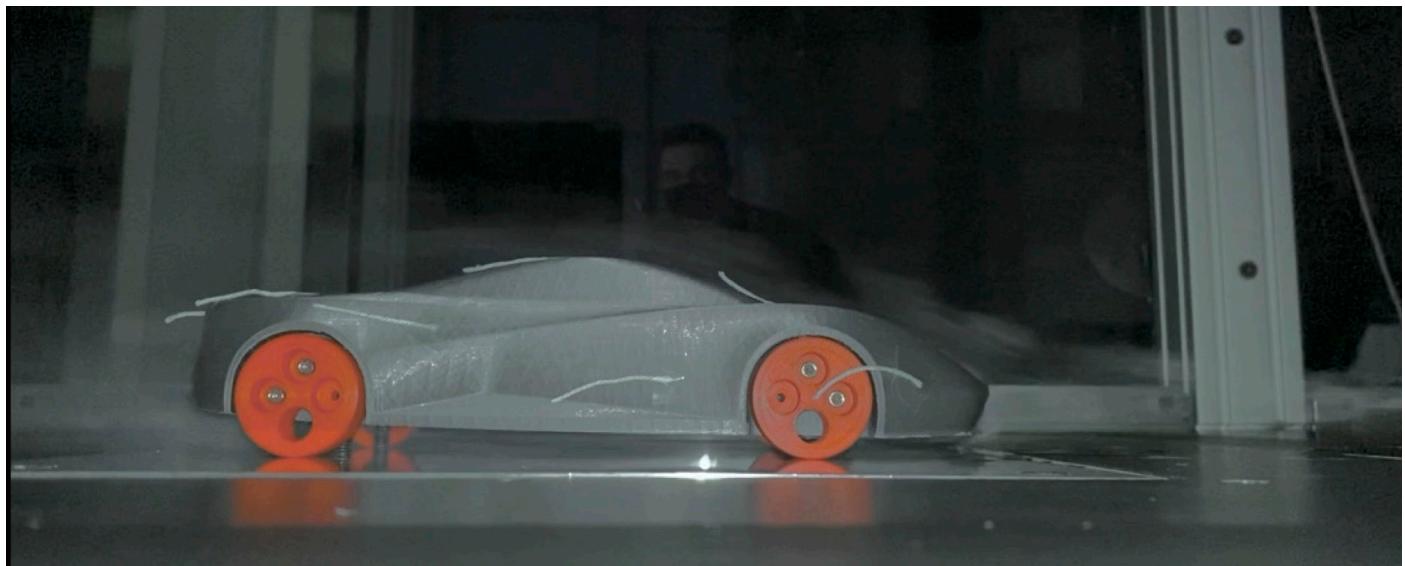


Figure 15: the 3D printed model in the wind tunnel.

Discrepancy

This discrepancy is significant - but where is it coming from? The simple answer is 'Drag.' The drag coefficient is higher in the wind tunnel values because the drag force is higher. But this is (obviously) only a partial answer.

There are a few things that could be causing this increased drag:

- Model Accuracy:
A 3D printed model is, because of the nature of 3D printing, less accurate than the CAD model. Even in the low-quality image above, layer lines are clearly visible, and these will have a significant effect on the aerodynamic performance of the car, increasing drag.
- Streamers
The model features several streamers to help visualise airflow. The car is relatively small, so these streamers may increase drag. Assuming each of 14 streamers (7 on each side) is a piece of yarn with a 2 mm diameter, the total area of $14 \times \pi r^2 = 14 \times \pi \times (1 \times 10^{-3})^2 = 4.4 \times 10^{-5} \text{ m}^2$, they represent 0.7% of the total frontal area of the car. This is significant - and if we took into account the less aerodynamic surface of the yarn compared to the plastic (not just the frontal area), this difference would become even larger.
- CFD
It is likely that the most significant difference here is neither of the above. It is another issue, one that is much harder to test. The CFD results are always referred to as a 'model' and the wind tunnel results as 'data' with good reason: SolidWorks is good, but it isn't *that* good. CFD simulations are based on the Navier-Stokes equations, which are solved iteratively, not numerically - they can only ever be an approximation.

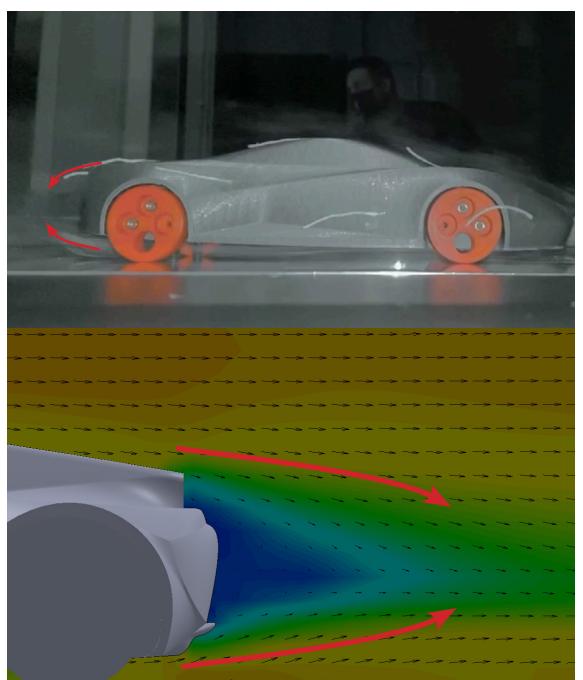
Comparison

When compared to my car, the example has a lower C_d and a corresponding lower drag force. Adjusted for the 1/18 scale of the model car, my car has a drag force of $581 \text{ N} \times 1/18 = 32.7 \text{ N}$ - 32x higher than the example car. The frontal area of my car is 2.169 m^2 , 1.2x the value of the example. This should not increase F by 32x.

As can be seen in the wind tunnel video (screenshot right), there is a small area behind the example car in which there is no airflow - an area of low pressure. This area is also present in my CFD analysis (the blue area on the cut plot for velocity, below right), but it is much larger than on the example model - this increased size will cause increased drag.

I think my car preformed poorly mainly because it is so angular (even though I've smoothed many edges) and because of its vertical rear. There may also be imperfections in the model, which would impact results, but this is unlikely as it passed the SolidWorks Geometry Check. I think, therefore, that my car may perform better in a wind tunnel: a car like this having such a high C_d is unusual, and a 3D printed model may have less sharp corners - a wind tunnel would also eliminate the errors caused by the approximate nature of CFD.

Figures 16 & 17: smoke around the wind tunnel model; detail of the rear of my car in a cut plot.



NEXT STEPS

The aerodynamic performance of my design is far from ideal - it's clear that the design needs work. I will sketch modifications over the renders and analyse the effects they will have using the drag estimator tool.

My modifications will:

- Make the rear end smoother
- Make the sides of the car less indented
- Flatten the back of the roof
- Remove the sharp feature above the rear wheel arch - this caused a sharp pressure decrease (see below)

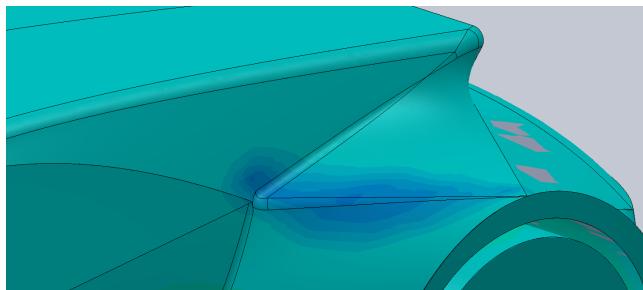
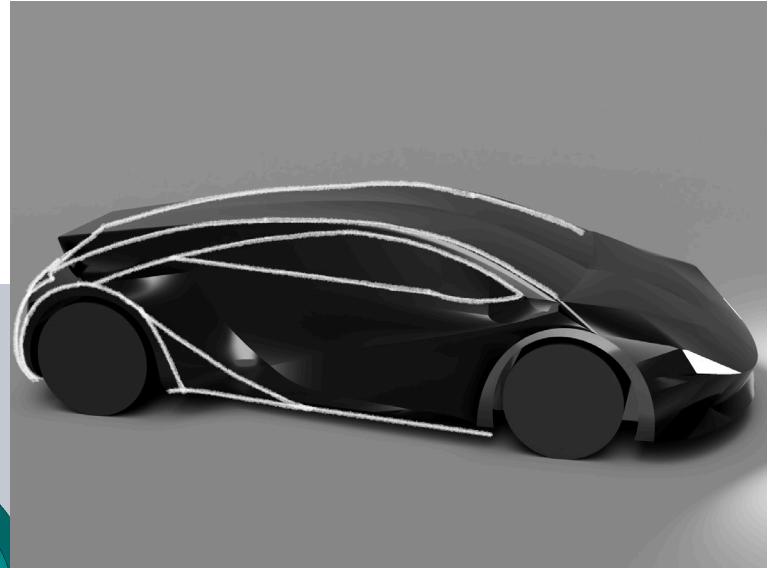


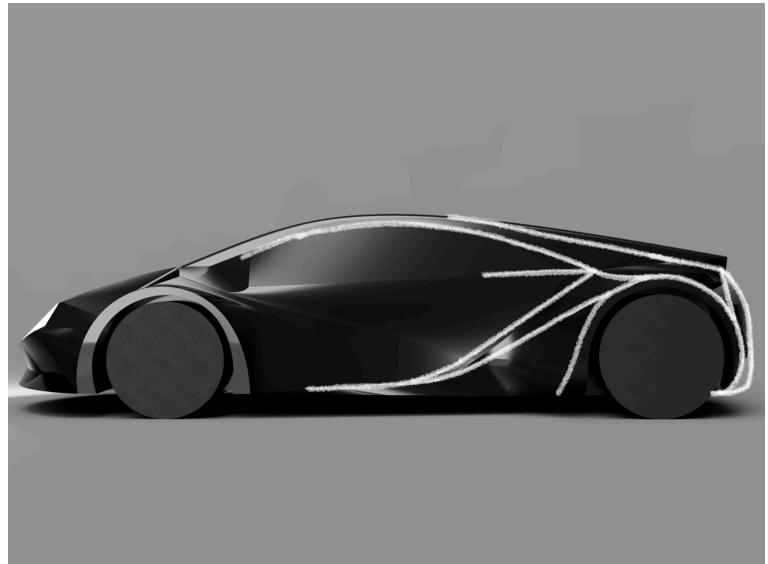
Figure 18: sharp feature causing low pressure



I used a tablet to sketch over my renders to show the new shape.

From the estimator tool:

Front end (top): A-2 - partially rounded	2
Windshield: B-3 - curved	3
Roof: C-1 - narrows at back	1
Read end: D-1 - sides are smooth	1
Front end: E-3 - curves down	3
Windshield: F-1 - curved	1
Rear roof: G-1 - Curved down	1
Cross section: H-1 - Curved	1
	Total = 13

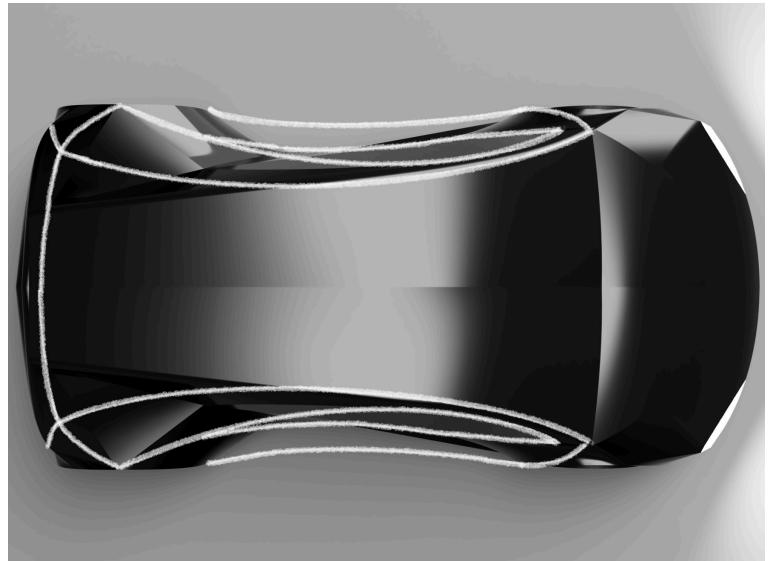


$$C_d \text{ (estimate)} = 0.16 + 0.0095 * \sum N_i$$

$$C_d \text{ (estimate)} = 0.16 + 0.0095 * 13$$

$$C_d \text{ (estimate)} = 0.284$$

This is a large improvement (-0.9) on the previous iteration - even allowing for this only being an estimate, this brings the car in line with the example discussed above.



Figures 19-21: my model with sketched modifications (white), making the car more aerodynamic to reduce drag.



Modelled on SolidWorks 2021 · Rendered on Fusion 360

Dyson School of Design Engineering · Thermofluids: Energy and Design