Design Journal Summary
FEEG6013 Group Design Project

15
Aerodynamics and model development of formula student car

The development of a legal and robust aerodynamic package for a Formula Student car suitable for wind tunnel testing

ID Number: 31942938
Name: James Clay

Primary Supervisor: Dr Zhiwei Hu
Co-Supervisors: Dr David Marshall

Submitted on: 09/05/2024

Summary of individual contribution

For this project, I was the Computational Fluid Dynamics (CFD) lead and team leader. My primary roles were to develop a CFD simulation of our full-scale Formula Student car for use in design iteration, produce a new sidepod design with tested cooling capacity and to validate the results of our wind tunnel test. I provided support to less experienced CFD users in the team and altered the simulation when needed. I sourced a MATLAB lap time simulator and performed all testing with it, collaborated with our team lead to solidify a clear design philosophy around our car and collaborated with other CAD teams to reinforce cohesion throughout the car's design. I took a role in the assembly of our wind tunnel model. During our wind tunnel testing, I was the primary member computer controlling the models' movements and did all the post-testing data analysis and CFD validation. Finally, throughout the project I assumed leadership roles, frequently assisting others and running meetings on multiple occasions.

Roles, activities and outputs

Initial planning and organisation

At the project's inception, I spearheaded the creation of a collaborative SharePoint space to facilitate team communication, and I took the lead in scheduling two regular meetings to accommodate the availability of each team member. Additionally, I introduced a skills matrix [DJE 1, pg 2], drawing inspiration from exemplary reports, to promote transparency and informed task delegation by allowing each member to identify their interests, strengths, and weaknesses.

Identifying key development areas crucial to enhancing both the aerodynamic package and the wind tunnel model was paramount. Consequently, I crafted a comprehensive project plan outlining aims, objectives, and tasks to steer our project forward. Initially, the plan ambitiously aimed to design and test a completely new aerodynamic package. However, it soon became apparent that our team lacked the requisite aerodynamic expertise and resources for such an endeavour. Thus, I adjusted the plan to focus on designing a new front wing and mid-car while concurrently improving the wind tunnel model.

To guide future aerodynamic development decisions, I formulated a flow field concept delineating our intentions and targeted flow structures for the final aerodynamic package. Leveraging my experience in SUFST's aerodynamics subgroup and my background in formula car aerodynamics from my individual project (IP), I created the flow field concept, as illustrated in Figure 1. Further elaboration on the flow concept is provided in the group report.

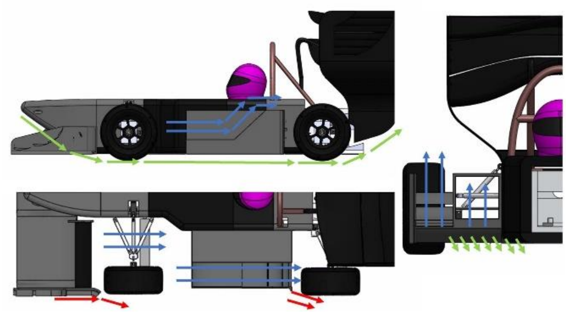

Figure 1 Flow Field Concept

During these early project stages, as the sole member with relevant experience, I assumed the role of mentor to other group members. My primary task involved imparting knowledge about front wing design to fellow aerodynamicists in the group. I shared insights gained from previous work and directed them towards additional resources, such as real-world Formula One designs, to expand their knowledge base.

My contributions during the project's foundational stages were indispensable. Without them, the project lacked structure. While others could have managed administrative tasks like setting up the group SharePoint, I believe the project would have faced significant delays due to a lack of understanding about single-seater race cars.

CFD Setup

The computational workflow inherited from last year's group was unusable due to many errors in meshing and solver settings causing solutions to fail. This meant we required a new CFD workspace to gather our data.

The setup of this was done by me exclusively. This was because we chose to continue to use STAR-CCM+ as I was the only person with experience using the High-Performance Computers at the University, which would be required for the level of mesh detail used in this project. and this was the software I was familiar with. [DJE 2, pg 2]

The initial setup would be conducted using last year's final design. The data would be validated against that which was provided in last year's report as no full data set had been provided from their CFD.

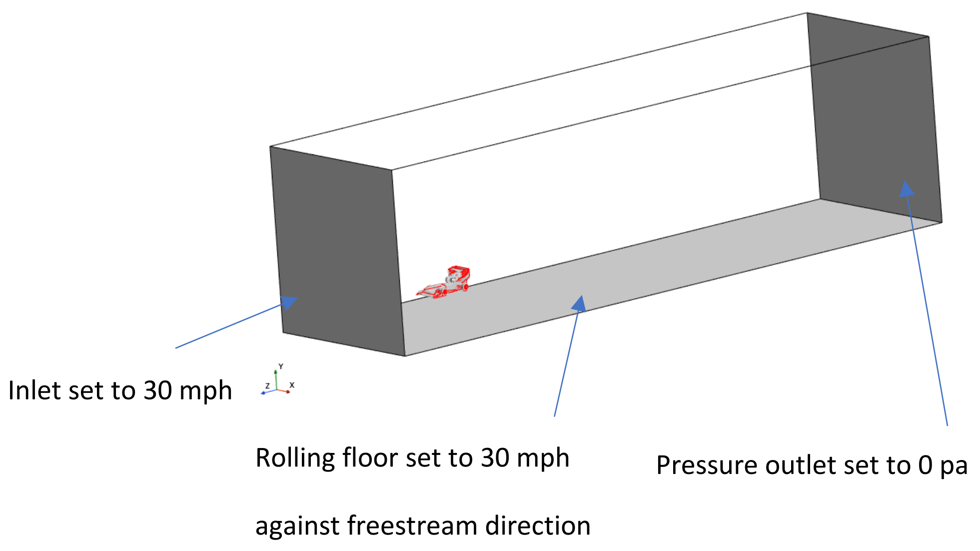

Figure 2 Domain setup

I setup up the boundary conditions for a straight-line half-car simulation. This is done to reduce computational time as the car is symmetrical and half of the domain can be modelled using a symmetry plane. I set the inlet velocity to be 30 mph as this is approximately the average speed of formula student cars across a lap distance. The ground was set with a boundary condition moving at 30 mph opposite the freestream direction. The wheels were setup with a boundary condition where they are rotating around their axle at 58.3 rad/s which is how fast a wheel of diameter 450 mm would be rotating if the car was travelling at that speed with no slip. Figure 2 shows these boundary conditions which will be used throughout the design process.

The next step was meshing which aimed for an accurate baseline representation through a mesh independence study, enabling observation of flow interactions, vortices, and separation points. Ensuring correct modelling of the viscous sub-layer involved splitting the CAD into constituent surfaces for enhanced mesh refinement control. Surface mesh splits were categorized into aero-components, chassis, wheels, and suspension. Mesh parameters between components were globally linked, allowing for scaling during mesh independence testing, while near-wall mesh parameters remained fixed, ensuring consistency in boundary layer mesh design. Figure 3 outlines the split surface parts.

The volumetric mesh was divided into fixed volumes for density refinement, with high-resolution regions designated for aerodynamically critical areas and vortex-prone regions. This is shown in figure 4. Coarser refinement areas were allocated where reduced detail sufficed, notably around aerodynamic components, as depicted in figure 6. Wake refinement regions, representing less crucial flow structures, were meshed less densely than finer regions but denser than the overall domain, providing a smoother transition from the car's vicinity to the broader mesh, as shown in figure 5. Initial prism layer settings were informed by the previous year's model, refined through iterative adjustments and optimization.

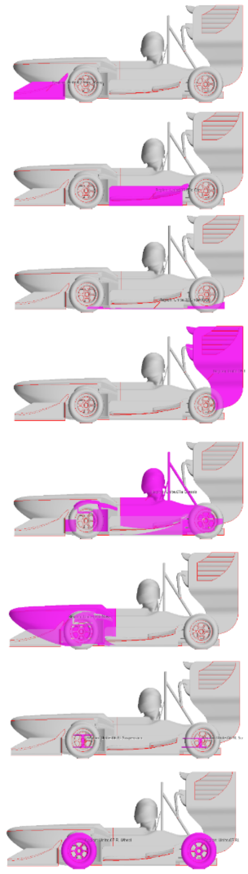

Figure 3 Diagram of the different surface controls

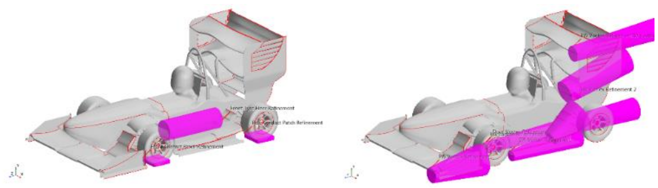

Figure 4 Fine vortex refinement

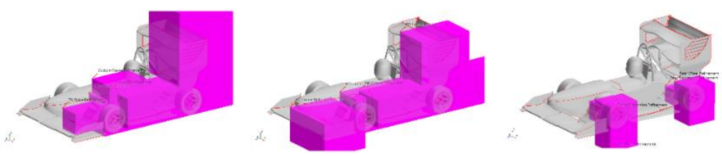

Figure 6 Close car refinement

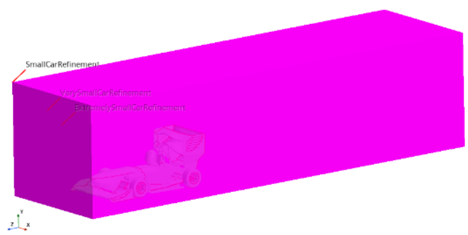

Figure 5 Coarse wake refinement

Now the solution was ready to mesh. The base size of the mesh generation was set to 0.03 m as this is what last year's design used. This generated a mesh that was 53 million cells in size. This had a wall time of approximately 16 hours when using 120 cores of the HPC. The simulation was set to run for 4000 iterations before completing however, upon looking at the residual outputs, it was clear that the solution had converged far before 3000 iterations as is shown in figure 17 and so I decided that this was the new standard iteration count. This reduced the wall count from 16 hours down to 12, greatly saving on both time and energy throughout the project.

Iteration	Continuity	X-momentum	Y-momentum	Z-momentum	Iteration	Continuity	X-momentum	Y-momentum	Z-momentum
2991	5.347972e-05	2.050976e-03	2.177476e-03	7.518071e-05	3991	5.527831e-05	1.745880e-03	1.824007e-03	7.176948e-05
2992	5.483726e-05	2.089806e-03	2.208562e-03	7.720528e-05	3992	5.664314e-05	1.770366e-03	1.855058e-03	7.474606e-05
2993	5.651962e-05	2.142995e-03	2.241127e-03	8.009922e-05	3993	5.753280e-05	1.796063e-03	1.894003e-03	7.672409e-05
2994	5.853937e-05	2.209977e-03	2.279117e-03	8.387636e-05	3994	5.753644e-05	1.814150e-03	1.915523e-03	7.576989e-05
2995	6.086439e-05	2.290002e-03	2.326727e-03	8.850877e-05	3995	5.737296e-05	1.827942e-03	1.922442e-03	7.358947e-05
2996	6.339355e-05	2.381954e-03	2.384663e-03	9.387929e-05	3996	5.765937e-05	1.842416e-03	1.927093e-03	7.201194e-05
2997	6.607049e-05	2.482962e-03	2.452846e-03	9.982547e-05	3997	5.836802e-05	1.856086e-03	1.920010e-03	7.098864e-05
2998	6.873714e-05	2.590748e-03	2.528334e-03	1.061235e-04	3998	5.922215e-05	1.868325e-03	1.909622e-03	7.041513e-05
2999	7.140690e-05	2.700987e-03	2.608117e-03	1.125827e-04	3999	6.005871e-05	1.882789e-03	1.908649e-03	7.024477e-05
3000	7.371495e-05	2.804108e-03	2.684729e-03	1.188779e-04	4000	6.052233e-05	1.900073e-03	1.921011e-03	7.038199e-05

Figure 7 Residuals from CFD

Last year's group also provided no post processing script so I had to create one from scratch. The script created was capable of outputting slices for Coefficient of Pressure (Cp), Total Coefficient of Pressure (CpT), Velocity and Vorticity. The script was created using java and I learned how to do this with help from tutorials on the STAR-CCM+ forums, YouTube videos and from resources from SUFST.

Now that the initial meshing was complete, I had to conduct a mesh independence study to gain confidence in the mesh density. The process of varying the mesh density was very simple, all that was required was editing the base size that was linked to all the volume mesh controllers. The results from this study showed that drag converged at around 35-40 million cells in the mesh while lift converged at around 50 million. For this reason, I decided to keep the base size at 0.03 m as the 50+ million cell mesh would offer much better data validity at a small cost to wall time.

The main issue that the values that the solution converged to values that were 8% and 28% away from the reference values of lift and drag respectively from last year. As a result, a solution using STAG 7 was created as I possessed a full, comprehensive data set for that geometry which was also already validated [DJE 3, pg 1]. The results showed good promise [DJE 3, pg3]. The error in lift appeared to be from problems adjusting the tyres between models as they were the main difference in lift values between the dataset. As can be seen, the error was much lower for this simulation than for that of the 2023 car and it was for this reason that I was confident in the CFD and was happy to continue.

The final part of the CFD process was modelling the radiators inside of the sidepod. To do this, a second region was made in CFD and it was made as a porous region. The radiator was modelled as a simple cuboid with the dimensions of the radiator mentioned in the cooling section. Ideally, the parameters for the radiator porosity would have been gathered experimentally, however there was not enough time or resources for this so the values used were the commonly accepted values for car and motorbike radiators.[1] This was also where I learned the method of how to model this condition in CFD as it was not something I was familiar with.

Cooling Modelling

My role for the design of the car had been to develop a new mid car section to aid with the cooling of the batteries while still generating high performance. An opensource MATLAB script was found which models this type of cooling design, and it was taken and adapted to fit our car. [DJE 3, pg 2] It uses the NTU method of heat transfer to model the heat lost to the air as a function of flow rate, airspeed through the radiator, ambient temperature, water inlet temperature, and radiator size.

The first piece of information I needed to gather was the velocity of the air entering the inlet. From CFD, it was found that the velocity of the air entering the cooling system was approximately 85% of that of the freestream velocity and this did not vary as a freestream velocity changed. The heat transfer coefficient was assumed to be approximately the same as the one provided by the author. This provided the following plot of cooling capacity.

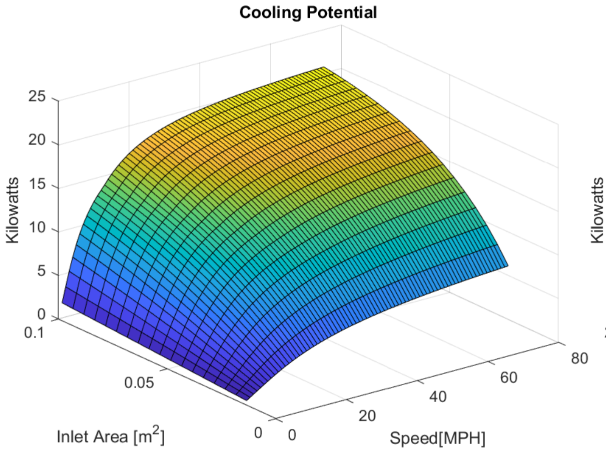

Figure 8 Cooling capacity plot

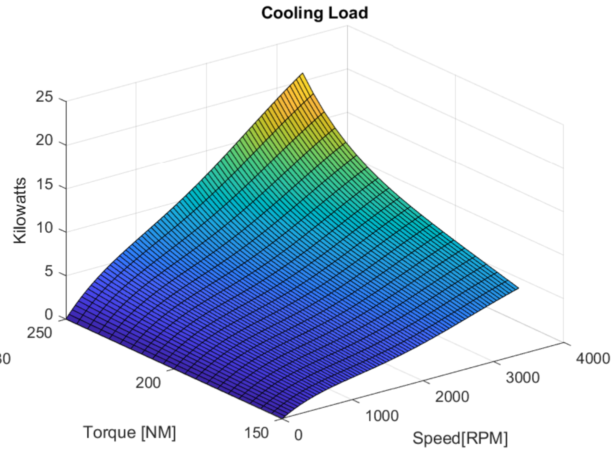

Figure 9 Cooling load plot

The other plot that the script created was the cooling load generated by the motor and battery unit. This was done with a generous assumption that all the inefficiencies in the motor and motor controller are cooling load. A value for the heat added into the system was found based on the current torque and rpm of the motor. There was also a built-in factor of safety of 1.5 to account for the many unpredictable variables and this variable was kept at this value.

From this it was clear that the cooling load would be somewhere in the vicinity of 10 kW. This meant that from the cooling plot it could be seen what the ideal radiator size would be. I decided a size of 0.05 m was ideal as this gave a cooling of 10 kW once the car was above 20 mph which is very low speed for this type of car.

However, upon browsing for radiators, I found a radiator from that was quoted as being suitable for BEV formula student. The radiator measures 195 mm x 178 mm which is slightly above what was wanted but it was decided that for now it would be a good initial test that and it could change if the data suggested it was needed to.

Now that I knew what the parameters of the cooling system were, I needed to test whether the cooling design would be sufficient. Initially, I just used average velocity to see whether the cooling would suffice for a lap distance, however it was pointed out to me in the interim presentation that I had overlooked the fact that this only works for a linear relationship [DJE 4, pg 1]. The relationship in this model is an exponential one and therefore I needed to implement a new strategy.

The way I resolved this was by feeding the velocity and motor speed data from lap time simulations (see next section) into the script and plotting how this would change the heating of the car over a full endurance race distance at an ambient temperature of 35 °C. The temperature of the batteries on the final lap of the race is shown in figure 10. The goal of this project was to keep battery temperature under 50 °C and this cooling design achieves this so I was content with the design parameters that had been obtained from this simulation.

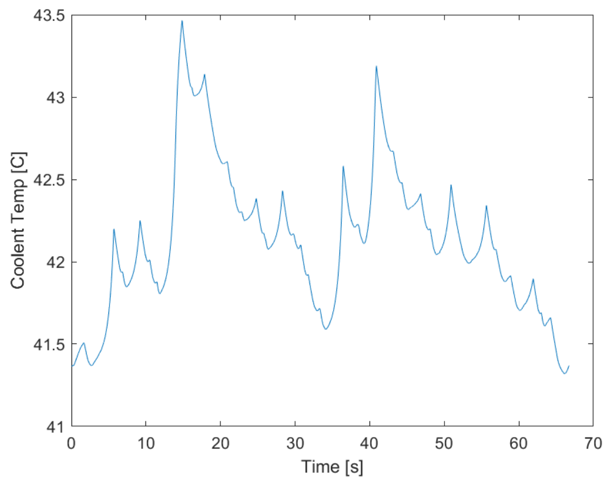

Figure 10 Plot of battery temperature over the final lap of endurance race.

Lap Time Simulation

To aid in both the analysis of the aerodynamic performance data as well as the cooling modelling, a lap time simulator needed to be used. I decided to continue using an open source lap time simulator called OpenLAP which the group used last year [DJE 1, pg 3]. Over the course of the project, this lap simulator was used as a quick way to quantify performance enhancements.

Once again, last year's simulation was not provided so I had to learn how to use the software myself and create a new simulation for both the car and track. The simulator is designed for use with an ICE powered car not a BEV. However, the car data was inputted by assuming a 100% efficient gearbox with only 1 gear (BEVs do not require gearboxes) and the torque curve was simply replaced with one that would be standard for a BEV as the creation of our own drivetrain is out of the scope of this project so approximate guesses of a power output had to be used.

The track dataset was hand created by myself. I did this by finding the track map from the 2023 UK Formula Student event and, using pixel measurements, converted the map picture into a set of straights and corners which the software could interoperate. The track did not need to be completely accurate as the data is only being used to present the difference in the performance of our designs and only needed to capture the essence of a Formula Student track especially since the track layout drastically changes between events between countries.

The output from the software is shown in figure 11. The software clearly does struggle with some pure longitudinal acceleration scenarios due to the instant torque nature of the powertrain however the lap time output was 69.091 s which is within the range of lap times from the 2023 autocross event so I was satisfied with the model.

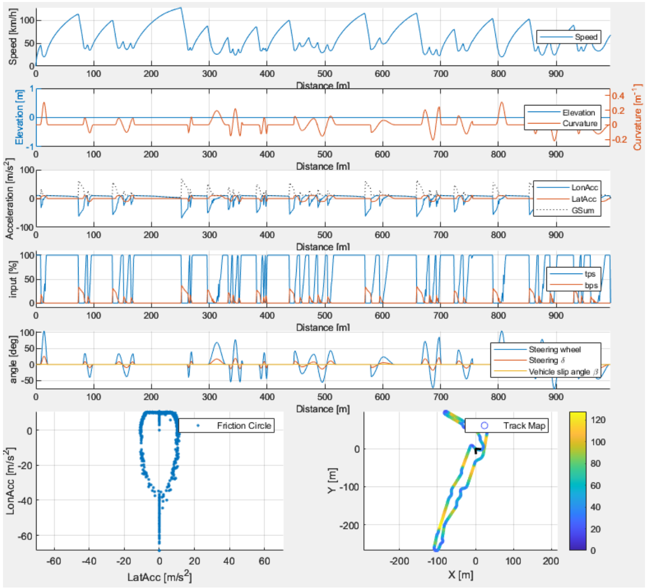

Figure 11 OpenLAP output

Sidepod Design

This first iteration of my sidepod design was very simplistic, and it did not perform well in CFD. There were 2 main goals to be used in this design: firstly, I made the underbody curved as this would create a venturi tunnel which would constrict the flow, creating a region of low pressure under the car, creating downforce. The strakes were added underneath to try and straighten the flow during cornering to encourage it to travel through the smaller cross section.

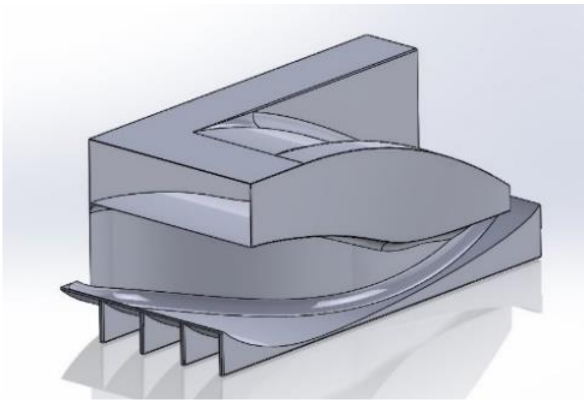

Figure 12 Sidepod Design 1

The other idea I had was to feed high energy air downstream of this component. This would help with rear tyre squirt and keep the turbulent wake of the tyre away from the diffuser, reducing its efficiency. This goal was aimed to be achieved through many geometry decisions. Firstly, the undercut between the inlet and the floor that travelled along the length of the sidepod would encourage flow to follow along using the Coanda effect and would be fed towards the diffuser. It would also have a secondary effect of pushing some air outwards, increasing outwash. Another similar concept is the sloped “slide”-like design on the top surface. What this does is takes the near freestream flow from on top of the chassis and feeds it towards the diffuser. The final thing is that by constricting the exhaust of the sidepod, the air would be forced to accelerate out of the sidepod towards the rear of the car.

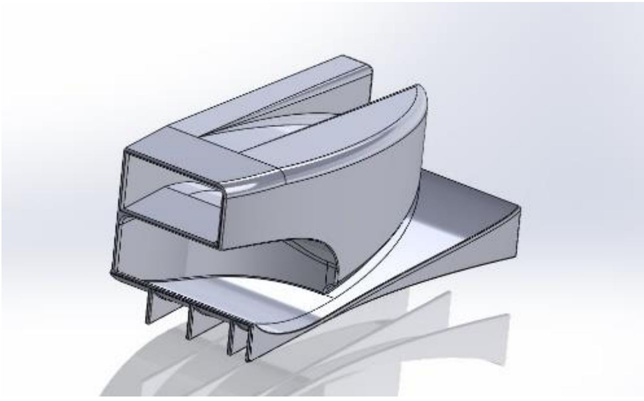

Figure 13 Sidepod design 2

I then tried to make a more refined version of the previous design. It aimed to gain performance by trending further towards the design of Formula 1 sidepods. A top down C_PT plot of a slice of the sidepod can be seen in figure 14.

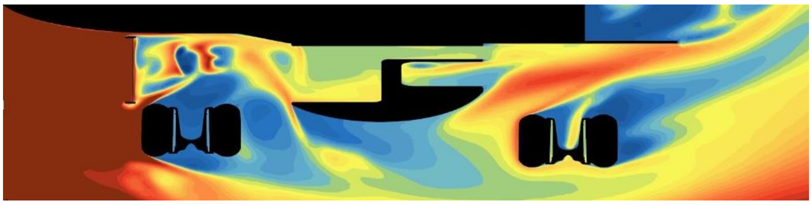

Figure 14 top down total pressure coefficient plot of design 2

The undercut had an effect in pushing air outwards. The idea of guiding high energy flow between the tyre and the chassis also worked as the low-pressure region is being kept away from the diffuser but it could be better. Below in figure is a side slice of total pressure.

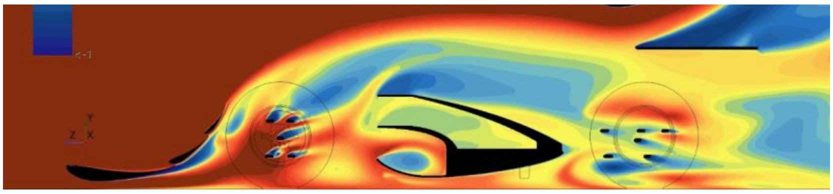

Figure 15 Side-on total pressure plot of design 2

Figure 15 shows that there was a lot of separation on the top of the sidepod this is bad because it means there is more drag as well as the flow heading to the rear of the car down the “slide” being lower pressure which is not good for diffuser performance.

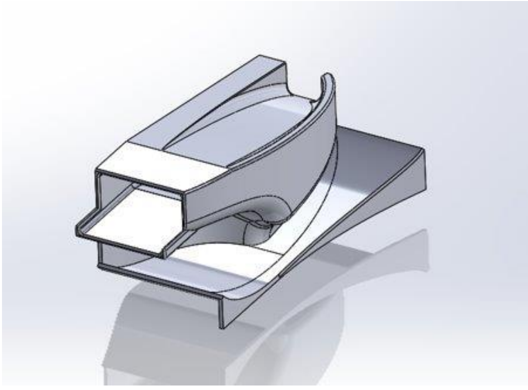

Figure 16 Sidepod design 3

The changes I made to this design included adding the extension in front of the inlet to try and reduce separation on the top. The strakes on the bottom were removed to explore a lateral expansion idea. Finally, the “slide” was widened. This was done so that the outer wall would be thinner and could possibly introduce a spilling effect over the top to generate more downforce.

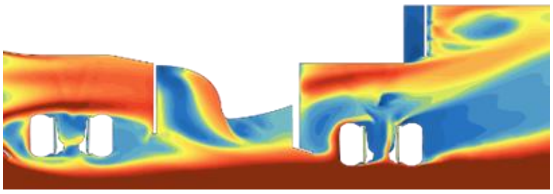

Figure 17 Top down total pressure plot of design 3

Figure 17 shows that the changes had been successful in managing the rear tire squirt as the tyre wake is separated much more from the diffuser external flow.

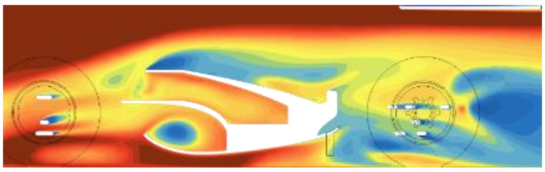

Figure 18 Side on total pressure plot of design 3

Figure 18 shows how the changes has reduced the amount of separation. Both the top surface and above floor separation has been improved.

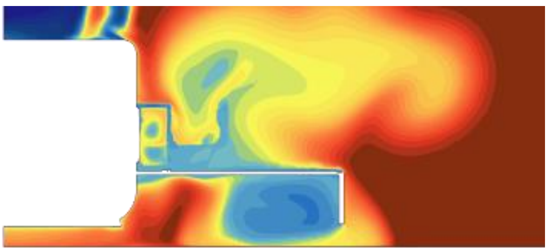

Figure 19 Front on total pressure plot of design 3

Finally, figure 19 shows that the removal of the strakes has allowed for a large low-pressure region to form under the car which is good and optimal for performance gains. These performance results are promising however, the difference in downforce is still larger than would be liked. However, there aren't many obvious areas where this could be resolved with the current concept and so I needed to explore a new design path. [DJE 3, pg 4]

Instead of taking inspiration from Formula One aerodynamics, where designs are heavily restricted, I decided for the next concept I would take inspiration from IndyCar and other Formula Student cars. [DJE 4, pg 2] These were considered because IndyCar is a very low drag concept due to the fact they run on ovals and Formula Student is the discipline this car is designed for (however with 11/19 DNFs last year, they still may not be the best inspiration).

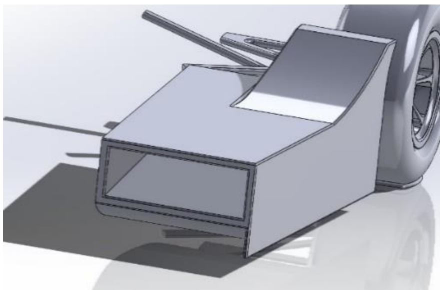

Figure 20 Sidepod design 4

This design was very simplistic. It did not have any overbody aerodynamics and the main source of downforce production would be the underbody venturi tunnel. The wheel guard would also produce some small upwash which would help slightly and by reducing the airflow to the rear wheel, it would reduce the lift the tyre generates.

This design already performed better than the previous ground effect design [DJE 4, pg 2]. Unfortunately, due to the simplicity of this design there also wasn't much room to change things.

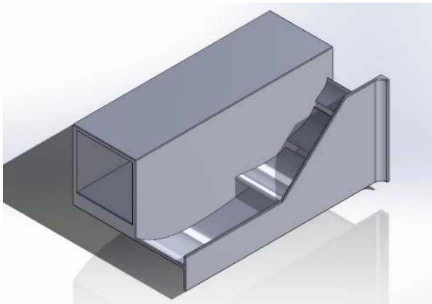

Figure 21 Sidepod design 5

This design was inspired by a lot of other high end formula student cars. It keeps the existing ground effect design but now it has aerofoils outboard of the sidepod inlet to create more downforce while still guarding the rear wheel by producing upwash.

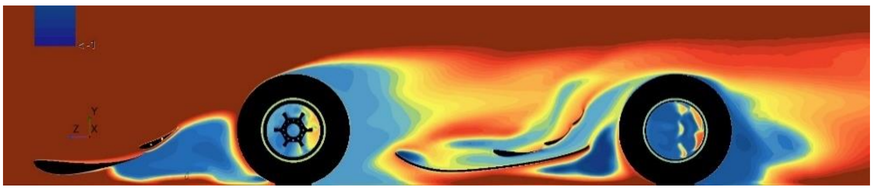

Figure 22 Side on total pressure plot of design 5

This CFD slice shows that the design was working almost as hoped. The only point of concern is the point of separation at the leading edge of the floor. There is a lot of low pressure produced beneath the mid car and the aerofoils are producing good amounts of lift with no separation and also directing flow away from the rear wheels. The performance was very good and this is now the design to beat. [DJE 5, pg 2]

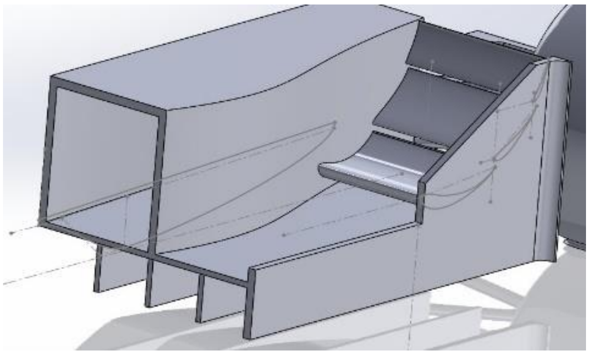

Figure 23 Sidepod design 6

This design attempted a few things: it added strakes underneath as the flow was moving laterally a lot, The leading floor edge was flattened a bit to try to decrease separation, the exhaust of the sidepod is reduced in area to accelerate the flow towards the diffuser as well as allowing for a wider span of the aerofoils. Unfortunately, this did not produce the results expected and increased drag and reduced downforce. [DJE 5, pg 2] This was the final design I could create before it was time to manufacture the sidepods for the wind tunnel test.

Wind Tunnel Test

During the wind tunnel testing, I had the role of the computer technician for most of the time. This was a role taught to me and A. Sidhu by Dr David Marshall, giving us the ability to move the car and change front and rear ride heights. In addition, Dr Marshall taught me how to create and edit ride height maps and run scripts, giving me a good understanding of the system by the end of the test. I was also in charge of adjusting the front wing angle in between runs during our front wing angle tests.

I communicated with Dr David about our run plan and he helped me choose what to do with our test given some of the restriction we had with our final model. This gave me a clear understanding of what the run plan was, and I communicated this effectively to the rest of the team to maximise our time and ensure the full run plan was completed. From this point onwards I took a leadership role in planning our timings for the day to keep testing running smoothly. This was a success, and all planned running and flow visualisation was completed by the end of day three.

Due to my understanding of the force balance recording plan, system, and data, it became my role to analyse the data collected. Dr David Marshall provided an Excel sheet of examples of how the data could be analysed [DJE 6, pg 5]. This was used to process all the run data for use as CFD Validation data where applicable. I provided members of the team CFD validation simulations replicating the wind tunnel run conditions for them to run overnight on Iridis, splitting the load. All data was correctly postprocessed and plotted against each other. Validation data also had percentage error calculations completed.

Validation data concluded sub optimal accuracy however the trends pointed to there being a likely systematic error. This still meant we could be trusting of our CFD so long as we applied the needed correlation factors. The likely cause to these errors were due to 2 things: the surface roughness of 3D printed parts as well as the simplification of the model in CFD.

Miscellaneous Achievements

Throughout this project, I also undertook many tasks that were in addition to my primary objectives. I mentored members of the group throughout the project on the topics of race car design. This included assisting A. Ahlam and Y.H. Tang with the development of the front wing design as they were struggling to keep up to schedule [DJE 5, pg 3-4]. I liaised with I. Lim over the design of new bodywork to find a compromise between aerodynamic efficiency and overall practicality [DJE 5, pg 5]. I assisted A. Sidhu and I. Lim in the week before the wind tunnel test by helping to assemble the wind tunnel model as well as fitting the scale model's radiators [DJE 6, pg 3]. I also created a new rear wing design however I did not have time to develop it enough to find any performance gains [DJE 6, pg 2].

Key achievements

In the project, I was the team leader, CFD lead and solely responsible for the sidepod development, which was integral in allowing the team to iterate and develop our final design proposal. Through these roles, I achieved the following key achievements:

- Created a project and initial design plan. And organised group members accordingly.
- Developed a successful CFD workspace, including the successful modelling of radiator aerodynamics.
- Performed mesh independence studies and CFD validation testing.
- Created a cooling model which could be used to accurately measure whether our cooling design would be able to meet our design objectives.
- Acquisition, adaptation and use of a lap time simulation model.
- Created 6 iterations of sidepod design each with design changes influenced by a combination of theoretical knowledge and CFD data.
- Oversaw most of the aerodynamic development of the car as the rest of the group struggled with the aerodynamic ideas required.
- Planned and oversaw our wind tunnel testing program.
- Handled and analysed all of the output data from the wind tunnel test.
- Wrote a large section of the group report with my content being approximately 40% of the total page count.

Critical review

General Project Review

Overall, the project can be split into three distinct outputs; the computational workflow, the aerodynamic design, and the wind tunnel model, with there being pros and cons to each. The aim of the project was to successfully develop improvements to an aerodynamic package that was designed last year while also improving the reliability and robustness of the wind tunnel model.

The aerodynamic development was successful in that it sufficiently cooled the batteries of the car whilst retaining 95% of the performance from last year and theoretically lapping within 0.2 s of last year's car. The problem was that there was still so much room for aerodynamic development. The project suffered from a lack of aerodynamic ability. Other than myself, the project consisted of I. Lim and A. Sidhu who were on the mechanical engineering course and subsequently had little to no aerodynamic knowledge; A. Ahlam who despite being on the aerospace program, did not choose to take the race car aerodynamics module in semester 1 and did not fill me with much confidence in her ability; Y.H. Tang who did take the race car aerodynamics module but failed it. Once that information was passed on to me in a group meeting after results day, I informed both supervisors in our next group meeting that I felt like I was having to bare the load of 2 or 3 people's worth of aerodynamic work and my concern's surrounding that. They understood my concerns but also knew it was now too late in the development cycle to resolve any issues if there was even any way to resolve it. [DJE 5, pg 1]

The CFD model was successful in aiding our design development. However, it would have been much better if we could have had more chances to conduct validation. This would have perhaps been possible if we didn't have to start from almost scratch but ultimately, the CFD needed to be ready early in the design process so that it could be used to iterate through our designs so it suffered from a lack of testing.

Ultimately, a point of reflection is that I struggled to take full control as team leader and failed to identify weaknesses in the group early enough to resolve these issues. I tried as hard as I could to pass on my knowledge to other members and I even combined both A. Ahlam and Y.H. Tang onto the front wing together early in the design process to try produce some positive output which did work as the front wing was successful however it did require a large amount of input from myself to even make sure it was legal within the regulations. I did feel a large responsibility as team leader and most experienced to try and fill in wherever possible and I should have gotten involved earlier in the design process but I was already struggling with the amount of work I was undertaking at that point in the project. Work was very sporadic from all members including myself and this will largely be in part from balancing other university commitments with the project but it did make coordinating things harder. I also personally struggled with motivation caused by mental health issues however, these were very hard for me to resolve due to lack of medical treatment available.

Innovation

Our team has undoubtedly demonstrated innovation for the entirety of the project. We have developed novel features for the front wing such as the adjustable flaps, as well as a brand-new bodywork and a sidepod design which is developed much further than a lot of Formula Student teams allowing our design to perform much better than others in an endurance event. The suspension system is also sophisticated and able to function properly when needed. We also experimented with different orientations of 3D printing to gain insight into the printer's behaviour to minimise failed prints in the future.

Nonetheless, being quick-witted and thinking outside of the box also helped advance our progress, since unforeseen problems arise and ad hoc solutions are applied to mitigate the issue. For instance, the team faced a problem where the bodywork did not sit perfectly while accommodating the load cell. Myself and A. Sidhu created quickly created a plan and hastily put it into action by modifying and adapting the bodywork geometry, or else the wind tunnel experiment would not have successfully taken place in time. [DJE 6, pg 4]

Communication

Communication was perhaps the weakest point of this project. The group did not bond extremely well and some members were much quieter than others during meetings. However, communication did persist well over text and was regular from all group members. There were regular meetings every eek however attendance fluctuated mostly due to illnesses but sometimes absences were unannounced.

The problems were evident when it came to the understanding of the project, this is in part due to ability previously mentioned however I do also feel in hindsight that I could have done a better job at explaining the design objectives to the group members as this became a problem in both the presentations we gave throughout the year where I was extremely confident in the design but some members were less so.

Sustainability

Throughout the project there were aims to increase its sustainability. Firstly, the car continued with the concept of a BEV as the global effort continues towards the development of alternative fuels to fossil fuels. Secondly, there were efforts in the computational design stage to reduce the amount of computational power used as to not use more electricity. Finally, we used PETG plastic which is fully recyclable and at the end of it's lifecycle can be converted back into 3D printer filament to be used once again.

References

[1] Anon. "How to simulate radiator, fan, and duct in STAR-CCM+ and design suggestions". siemens.com. Available https://community.sw.siemens.com/s/article/How-to-simulate-radiator-fan-and-duct-in-STAR-CCM-and-design-suggestions [Accessed 28 04 2024]