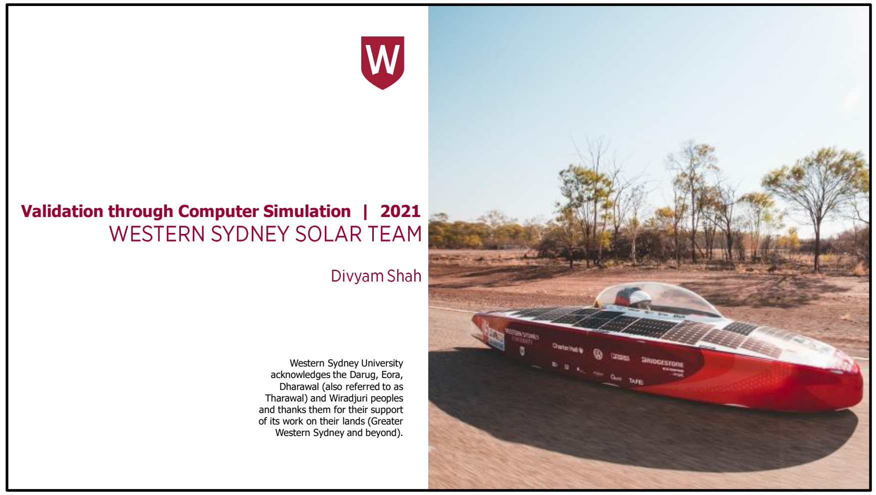
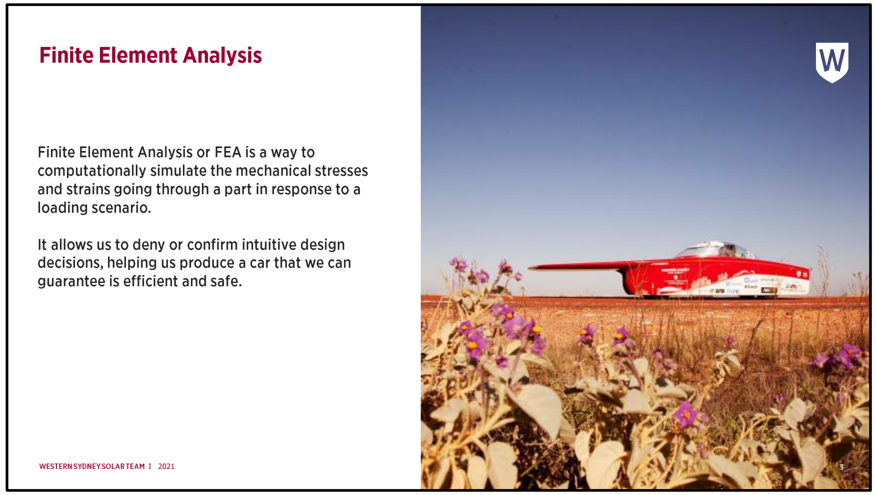
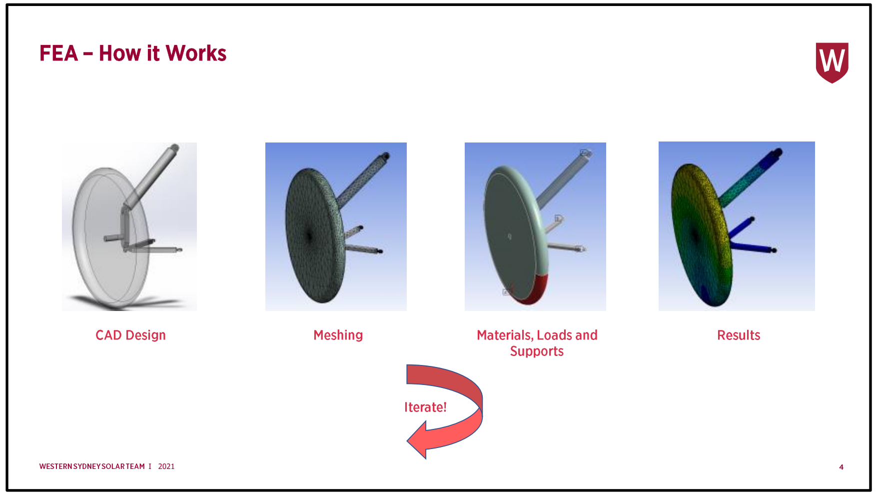


Hi Everybody, Thank you for joining me in this deep dive into Validation through computer simulations. I am your presenter for this session, Divyam Shah. I want to start off by acknowledging the Darug people who are the traditional custodians of the land from which I am joining you. I would like to pay my respect to elders both past and present of the Darug nation and extend that respect to other Aboriginal people present here today.

The purpose of this presentation is to showcase the ways in which the team uses various forms of simulation technologies to create a lighter, safer and cheaper vehicle. We want to share the amazing capabilities of these software, some really exciting features it offers and why skills in this area should appeal to any business or engineering professional. I also want to let you know that a lot of the geometry or simulation data in this presentation is not representative of the actual solar car components, but more like some demo images. Now this wont be a tutorial on how to use these software, but I am happy to help more if anyone does reach out afterwards with further questions!

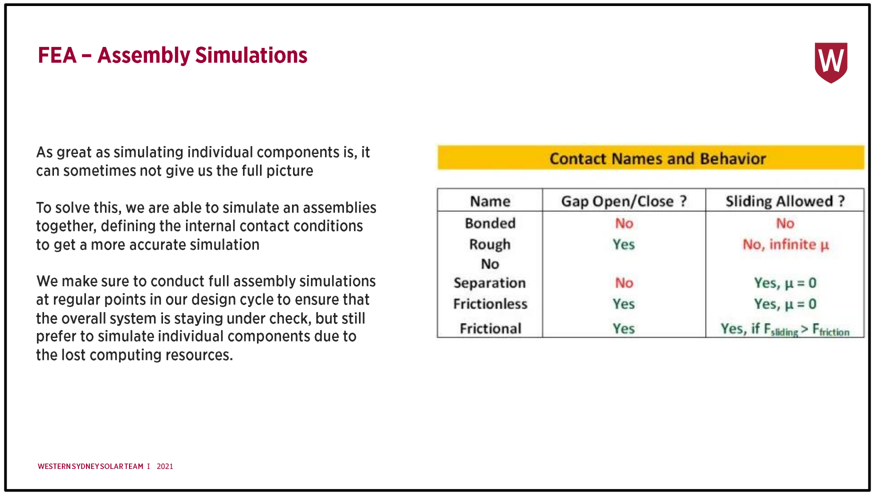


FEA, or Finite Element Analysis is a way to computationally simulate the mechanical stresses and strains going through a part in response to input loads. Almost all mechanical parts of our solar car is designed in-house. This allows us to optimise that part to be as light as possible for its given loading cases. However, we can only get so far based on first-principles and intuition. To produce a highly efficient and safe machine, we employ FEA to analyse each an every mechanical component that we design. The team uses the Ansys software package to do these, however there are many different products available.



The general process of performing FEA analysis is to design a part in a CAD package. The output of this CAD can be imported into Ansys. In Ansys, we first need to mesh the part, which I will go into further detail later in the presentation. Next, you can specify the material that the part is made out of, the load points and types, the support points and types and what results you want out of this simulation. It supports simulating direct point loads, distributed loads, remote loads, pressure forces, moments and many more. Additionally, it supports many types of support joints, like fixed, pin, roller etc.

Based on these conditions, the simulation result can output many different parameters that describe the behaviour of the part. The mains ones we use are the stress, strain, deformation and Factor of Safety outputs. The amazing part of this is that it is displayed as a 3D colour graph overlayed on the part itself. This allows a visual understanding of weak points and over-engineered sections. Using this, you can add material where the FOS is too low, and remove where its too high. Bear in mind, this is a very simplified explanation of the process. We use FEA to iterate our designs multiple times until we are pleased that it is up to our safety standards and is as light as possible.



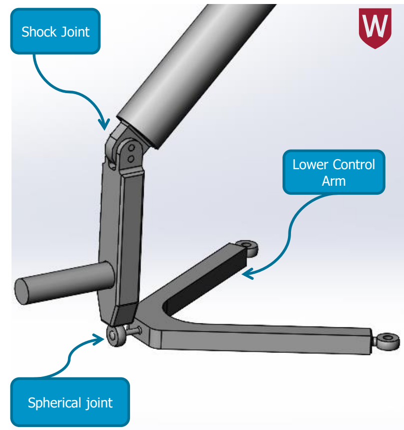
Simulating individual components is quite useful, however it doesn't always show an accurate picture. However, using Ansys we also have the ability to perform FEA analysis for entire systems that contain the components. Here, we can set various contact conditions that define the interaction between the various components as shown on the screen.

FEA – Assembly Simulations

As great as simulating individual components is, it can sometimes not give us the full picture

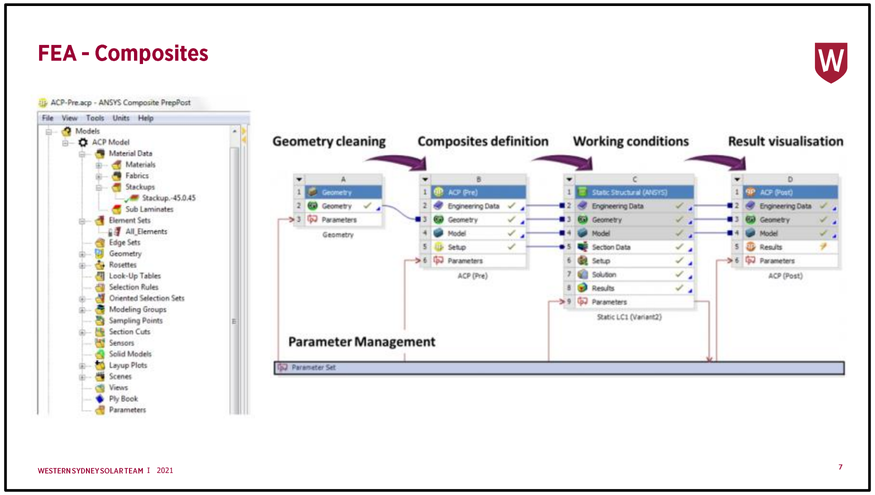
To solve this, we are able to simulate an assemblies together, defining the internal contact conditions to get a more accurate simulation

We make sure to conduct full assembly simulations at regular points in our design cycle to ensure that the overall system is staying under check, but still prefer to simulate individual components due to the lost computing resources.

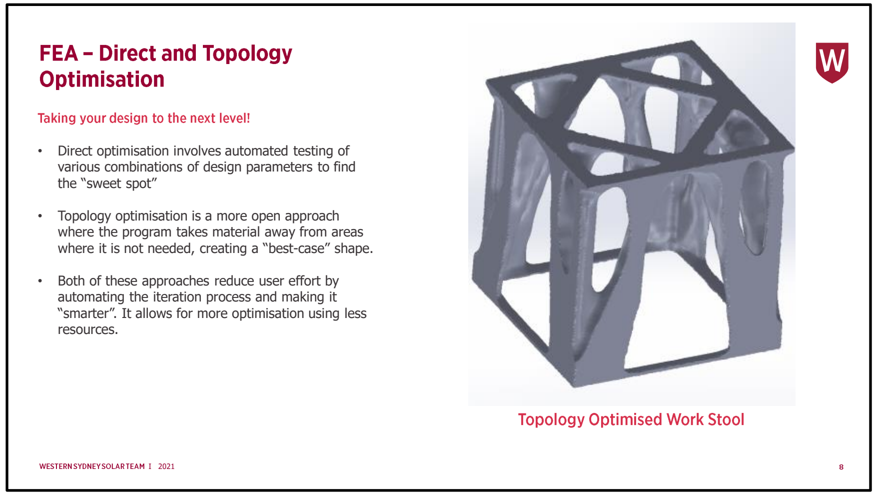


WESTERNSYDNEYSOLARTEAM I 2021

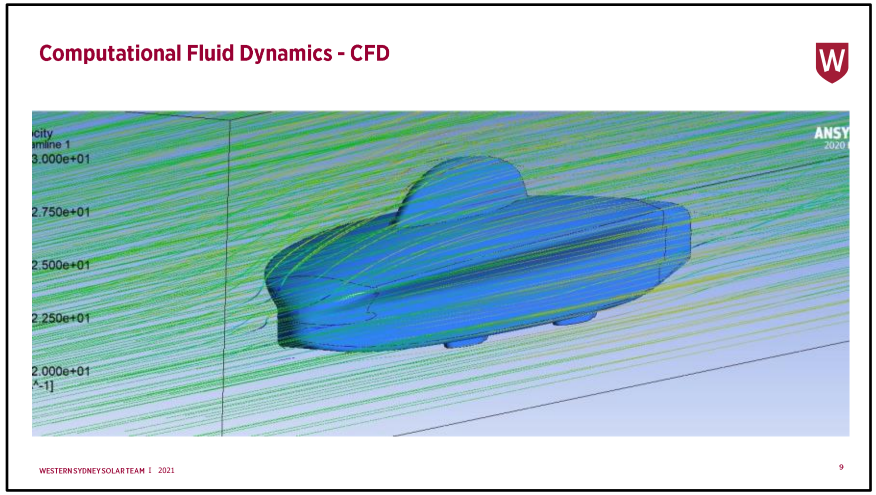
So for example, the contact surfaces of a spherical joint can be set as frictionless, allowing it to rotate freely but not come out of its housing. Its important to note that this type of FEA is still on a purely rigid body, with no mechanical movement designed. But setting appropriate conditions is still very important. In the example, setting the spherical to be movable, doesn't let the upright turn, but it does ensure that all of the moment that is applied to it is counteracted by the shock joint, and not by the LCA. Full assembly simulations account for all such internal interactions, as well as the adding up of deflections through the assembly, overall providing a much more accurate understanding of how the parts are behaving.



Another great tie into the FEA process, is Ansys's ACP tool. This allows the simulation of parts that are manufactured out of composite materials. A very large portion of the solar car chassis is made of composite, so this allows us to perform FEA and optimise it. The ACP FEA method involves the design starting as surfaces and not solids as before. Onto the various faces of your surface bodies, you can specify the layup-schedule that you have in mind. We can define the type of composite, the number of layers, the angles of each layer, any core material as well as any glue films using ACP. It also allows you to specify any relief cuts that can be areas of concern if not designed appropriately. After specifying the composite makeup, you can perform the normal FEA analysis process, but you can view additional solution parameters regarding composite specific failure modes. The ACP toolbox is instrumental in the design of the Solar Car and can prove useful for any composite design process.

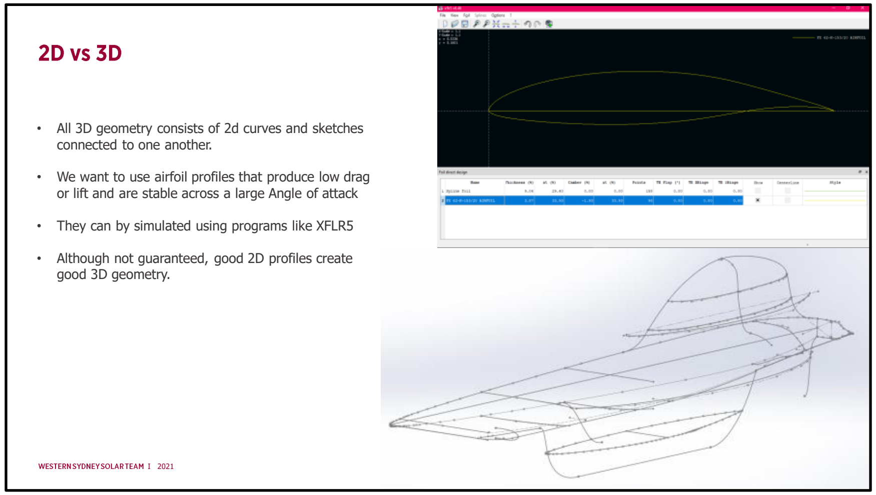


The last thing I want to discuss about FEA are the extensive special features that programs like Ansys enable. Within the FEA workflow, you can specify things bolts and their pretension, glue joints and weld joints, increasing its accuracy and versatility drastically. Ansys also offers 2 amazing optimisation options that allow you to take your design to the next level. One is direct optimisation. In this, you make your itself parametric. For example, the spars on this panel can have a thickness, depth and material that is set as parameters. Through a direct optimisation loop, you can set constraint, like a minimum FOS of X, a target like "minimise weight", and let Ansys automatically perform various simulations to determine the combination of the defined parameters that best satisfy the your set goal. This is pretty incredible, as it cuts down on the number of iterations you need to refine your design, greatly improving your productivity. Ansys also have a tool that takes this one step further, which is topology optimisation. In this, you can set similar constraints and goals, but instead of specifying parameters, you input a vastly excessive design, like such. And through various rebounds of FEA analysis, it automatically removes material in strategic areas to meet your set goal, as you can see in the example. This still feels like magic to me sometimes. There is still one major downside to this process, and that is manufacturability. An annoying trend is that highly optimised designs are often not manufacturable through conventional methods. The team is still learning some of the manufacturability constraints that you can add to you optimisation and are also trying to manufacture more components through additive methods that can actually make crazy "organic" designs. However, if nothing else, topology optimisation allows you to find the theoretical "best case shape" which you can then use as inspiration and edit to make manufacturable.

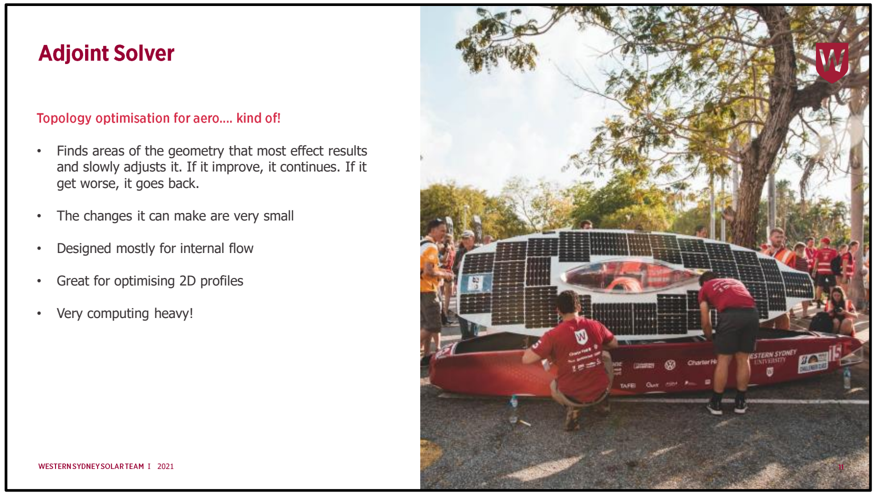


The second really important simulation type I want to talk about is computational fluid dynamics, referred to as CFD. Aerodynamics is such an important part of a solar car's performance, estimated to make-up close to 70% of its power losses. How there are many intuitive ways to design an aerobody that is aerodynamically efficient. This includes reducing the frontal area, reducing the number of and harshness of junctions in the body, ending all components in a sharp trailing edge and trying to follow efficient 2D air foil profiles as much as possible. However intuition will only get you so far. To continue optimising any further from a rough concept, you need to have a way of knowing for certain how the air is behaving around the car. For many applications, this step would be done using a wind tunnel. However this is simply to expensive for the team to reasonably achieve. Instead, we use what is essentially a virtual wind tunnel, in the form of a CFD simulation. In this, we once again start with the CAD of the aerobody we want to simulate. The main difference is that instead of importing the aerobody itself, we import all the air around the aerobody and remove the area where the aerobody was. This is because instead of simulating a car moving through the air, CFD looks at wind moving over a stationary car at the same speed. The empty area where the aerobody was acts like a wall that the air can't pass through.

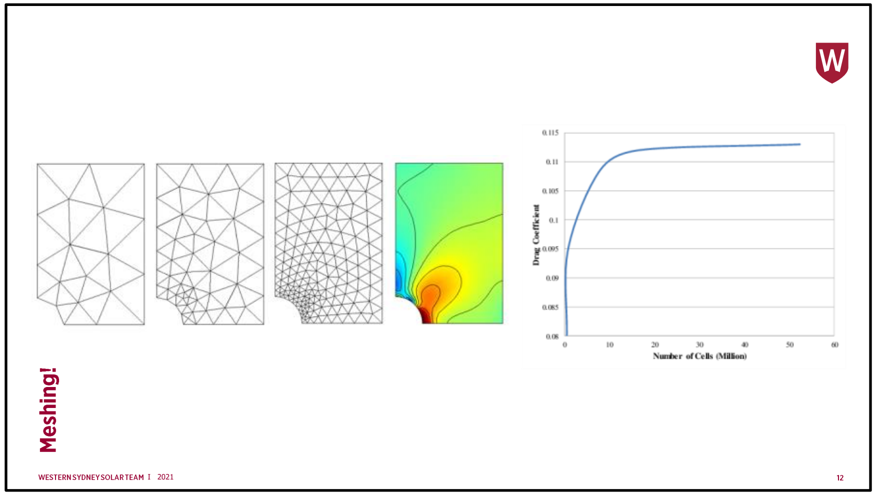
Once the CAD is in, we do the meshing process. Now like I said, I will dive deeper into meshing later on. But the important thing to know about a lot of aerodynamic studies is that a lot of complex interaction occurs very close to the surface of the object in what is called the boundary layer. This is why the mesh also needs to be incredibly fine in the few centimetres directly above the skin of the aerobody. Once you have a mesh, you can specify your boundary conditions. This include how the air is entering the wind tunnel and at what speed, how it is exiting the wind tunnel area, you can set the ground to be moving to better simulate the real world and you can also set your tyre surfaces to rotate in order to capture the complex interaction between an exposed tyre and the air. Compared to FEA simulations, CFD has many, many more cells in its mesh. And the mathematical model it needs to solve is also much more complex, leading to many more simulation parameters that you can tune and also cause much higher simulations times. Fluid mechanics involved non-linear processes, dealing with inherently unstable phenomena such as turbulence meaning that solutions are not constant. Which means that the calculation iterates until the results start converging. This makes CFD simulations very hard to get right and accurate. Once you do have good results, what you can do with it is incredible. The main things I used to analyse an aerobody's performance is look at various forms of pressure plots to see if there is separation occurring or if unnecessary forces are acting on the body. We can also visualise vortexes and other turbulent flow generated around the car and quite often track what part of the aerobody is generating it. Using this and many more results variables, we can compare iterations of a design and evaluate if changes made are good. Over many iterationsof CAD and CFD, we can end up with a highly optimised solar car that produced very little aerodynamic drag.



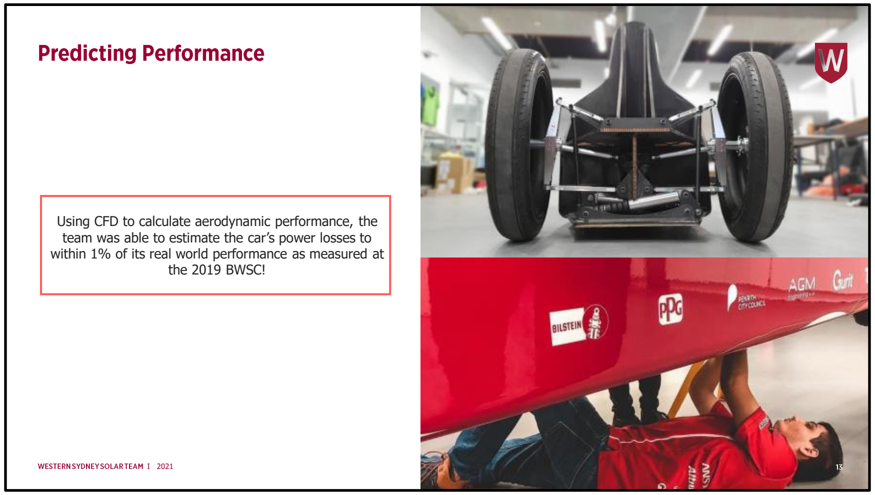
Now like I mentioned in the previous slide, a lot of the aerodynamic performance of a solar car revolves around using the right airfoil profiles to construct your 3d geometry. As the bottom image shows, any solar car aerobody consists of various shapes and sizes of airfoils places at various locations in various orientations across the car. These are joined from one to the other using complex surfaces that flow really nicely into one another, creating smoother junctions. However, for the final shape to be good, each airfoil needs to be good. As such we use programs like xflr5, which is shown in the top image. In this we can design airfoils that are the appropriate shape to cover the insides of the cars (like the tyres, driver etc). We can tweak these profiles and simulate them to see plots of Cd, Cl vs the angle of attack. The best performing ones go into our 3D geometry.



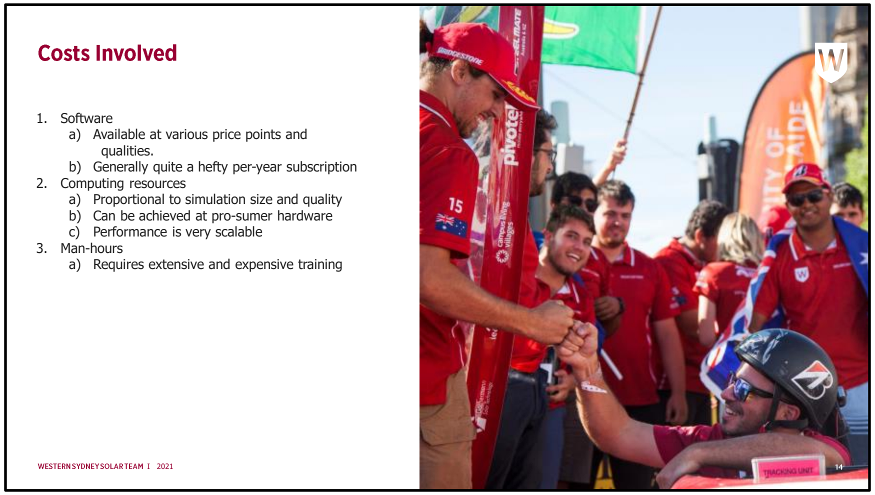
The cool bit of the software that I want to talk about that is related to CFD is a tool in Ansys called Adjoint Solver. This is like toplogy optimisation for CFD. It allows you to import a design, create restrictions like it cant pass through this area or it cant get any smaller than this size etc. Then you can set goals like to optimised for the lowest drag force. Using this is creates analyses what areas of the geometry cause the biggest change in the results if moved. It then moves them around to try improve the design. If a change causes improvement, it keeps going, if it doesn't it goes back and tries something else. In this way it tries iterating to find the most optimum aerodynamic shape. Now this has its limitations. You cant exactly put in a brick and expect to get a solar car out. The changes it can make are minute, like just tweaking the curvature of various surfaces to try improve overall performance. It also works better for internal flows and not external flows like the solar car needs. The compute time for those sorts of simulations is also incredibly high. Think about a single CFD simulation, and this is iterating it. However, we definitely do use it for smaller uses take optimising some 2D airfoil shapes etc.



Now in both of these analysis type, one very common step is meshing. Both simulations are based on specific mathematical models. FEA for example, would not provide you any more information than just solving a free body diagram, if it applied the mathematical model over the entire component, considering it a single item. However, if that component gets split into thousands of smaller cells that create mesh of the part, the equations can be applied to each cell individually. A very simplified visualision is that it analyses the behaviours of each cell individually, and passes the forces onwards to the adjacent cells. Once cells outputs are the next cells input. This way it analyses what's happening at thousands of locations within the part, allowing the functionality that it does. The same principle applies for CFD. This makes the mesh itself quite critical to the accuracy of the simulation. There are two main factors of the mesh that need to be considered, one is the quality. In simple terms, you want as uniform of a mesh as possible. You want cells that have very even aspect ratios, You don't want very large cells next to very small ones, rather them slowly transitions from big to small over many layers. For CFD, you want a very specific boundary layer of cells that is right for the part you are simulating. The second important thing is the density. The finer of a mesh you have, the more detail you will be able to see in the results. You will be able to differentiate the conditions at different locations with more granularity. And you will be able to capture disturbances caused by small features more easily, especially in CFD. However, there is a compromise. The finer your mesh is, the more computational resources it takes to run the simulation. This is why, it is important to perform sensitivity studies. Using one representative part, you can conduct the simulation with various different levels of mesh density. Once the results stop changing, you have reached diminishing returns and that is the level of mesh you should run to get the best results for the least computational resource. This is shown on the graph on the right as the point where the data levels out. Once you know this, all other similar parts can be simulated under these conditions, saving you time.



There is a big divide in the quality of FEA and CFD analysis that can be done. If using bad software, with little experience and with terrible meshing, you can get results that are completely irrelevant. However, if done properly, you can get incredible close to real world results. This is why, almost any race team in the world employs this type of simulations. We ourselves found that in 2019, the theoretical power efficiency of our car, estimated through CFD simulations was within 1% of how it performed in the real world. This was incredible exciting to us and is a statement to how amazing the simulations are.



Now that we've discussed what it can do, I want to talk about what it'll cost. Less in terms of dollars and more in terms of what you need to consider.

Software

The first consideration is software costs. There are many different software available with varying functionalities at varying price points. However any good packages are within the 10's of thousands per year per licence, plus what it would cost for support.

Computing resources

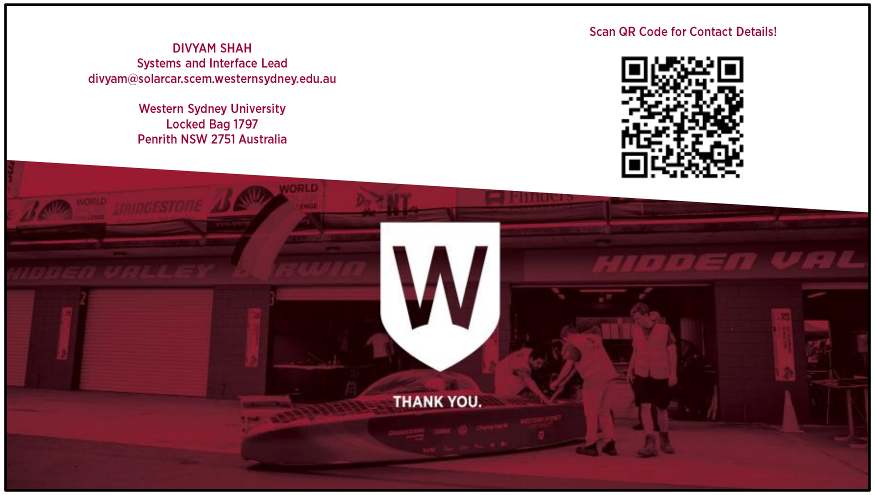
These simulations are also very computationally intensive, especially CFD, where the mesh sizes are are quite a bit larger. As an example, I was able to run the level of CFD simulation that got to within 1% of real world for the solar car aerobody, a sim that had approximately 50 million cells in 48 hours on a pro-summer workstation that was in the ballpark of $4000. However, it is very scalable. lesser machines can run it over a longer time and better equipment could run it within a matter of hours. The type of computing needed would heavily depend on your use-case, but it still an integral cost to consider.

Man-hours

Labour is also important to consider, although to us, a voluntary student-led team, its not something we think about. but, using these software is a highly skilful task and often requires extensive and expensive training, which can add up quite significantly.

Now we are proudly supported by the amazing Leap Australia who are our sponsor for the Ansys package. And we are also a very small team that is only building one vehicle every two years. However, even if we had to incur all the software and labour costs ourselves , we still believe it would be a cost-saver for us. We are all students who are still learning engineering design and we are trying to optimise our designs to the absolute leading edge. There is no way we would be comfortable with one of our team members driving such an experimental vehicle without appropriate validation. Without computer validation, this would mean we would need to mechanically test every component before we put it on the road. This would mean testing a lot of parts to destructions. Because we are still students learning, the mistakes we make would only be exposed through physical testing and not the computer simulations, leading to many more manufacturing runs and it would also drastically lengthen our project timeline. Overall we believe that this would be less cost-effective than spending the money on computer validation. I do urge you to explore whether it can also help you business/company.

The 2021 and 2023 regulations are looking to be much more stringent on stability. However, given the increased ride height requirements and introduction of 3 wheelers, there's real ease with which car configurations can become unstable. The regulators will conduct a 45-degree tilt test as a way to verify stability, but within the team, we are looking for further validation. Something that looks at the wheelbase, track, weight distribution, suspension geometry, shock characteristics and tyre characteristics altogether. And something that we can use to simulate various extremes of operational conditions that the car will be under to see how it would react. This is where multi-body vehicle dynamic simulations come into play. This is something that the team has never looked into before, but it is looking necessary now more than ever. We are attempting to use Adam's Car to create a digital twin of the car that accurately models all the characteristics listed. From here we can create various road surface maps and driver input graphs to simulate any common or extreme situations. We hope to be able to simulate hard cornering, braking while cornering, going over cattle grids and also swerving manoeuvres that may be a reaction of crosswinds. Out of this, we hope to be able to see a physical output of how the car behaves, forces through suspension components, reaction forces of the tyres among many other things. Not only will this be able to validate the overall design's stability, but it will also allow us to optimise the suspension geometry for stability, which before has just been looked at from a rolling resistance POV. As of right now, I don't have much more to share about this, but we hope to implement this into our design workflow and hopefully can share more results soon!



Thank you so much for listening to my presentation. I have on this slide, the contact details for myself, and also for Leap Australia, one of our sponsors. Please feel free to contact me if you have any more questions regarding what was presented or would like to get involved with the team as a member or a partner. For inquiries about purchasing such software for commercial application, please do contact Leap. There are incredibly knowledgeable and provide unbelievable support. Now, in the 15 minutes left, I'll be able to answer any questions from the audience. Please use the hand raise button on zoom, and I'll try get to as many people as I can.