344 Final Project - Daniel Clark - dcla189

Partner - Michael Terekhin - mter153

Introduction

Over the past ten weeks, Michael Terekhin and myself have been analysing the aerodynamic and structural performance of an airfoil, using the popular engineering simulation software Ansys. Specifically, we are interested in determining the lift and drag coefficients of the airfoil, and the optimal design of an aluminium spar within the airfoil to minimise its mass, while maintaining structural integrity. In this report, we have outlined our most important findings for non-expert clients, and provided some background into our computational design process, for technical experts.

Summary of Main Findings

For the aerodynamic performance assessment of the airfoil, we were provided the airfoil's cross-sectional shape, and asked to determine the coefficients of lift and drag that would characterise the airfoil. *Figure 1* shows the coefficients of lift and drag plotted against the airfoil's angle of attack. We can see that the lift being generated increases approximately linearly with a greater angle of attack, however the drag coefficient remains negligible until a very large angle of attack is reached when it increases rapidly. This is likely because of the "stall" effect, when the drag coefficient will increase rapidly if the angle of attack is too high, causing the aircraft to stall [4]. The stall angle of this airfoil is ~15 degrees. A plot of the coefficient of pressure (C_P) can also be found in the appendices.

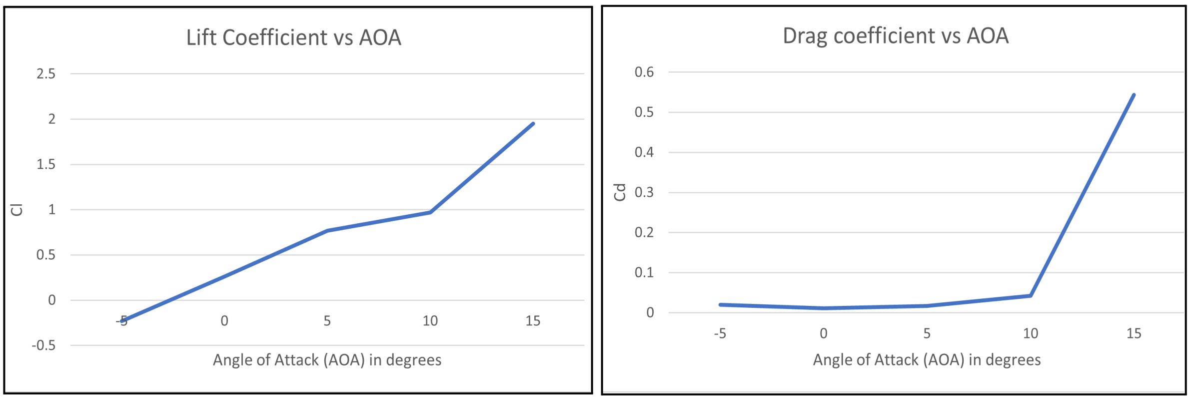

Figure 1: Lift and Drag coefficients vs Angle of Attack

The ratio of C_L/C_D, when plotted against the angle of attack, can help us to understand the efficiency of the airfoil at that certain angle. We wish to maximise this ratio, to provide the most lift while creating the least drag. *Figure 2* plots the results of this analysis, and shows that flying at an angle of attack of 5 degrees is the most optimal angle. This will also maximise fuel efficiency of the aircraft.

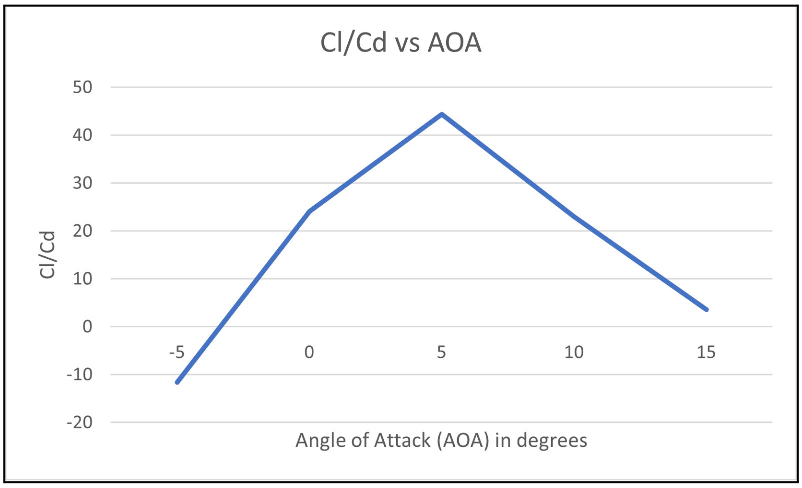

Figure 2: C_L/C_D against angle of attack showing efficiency against Angle of Attack

Similarly, for the structural analysis, we investigated many different spar designs. After considerable research (LINK), we determined that a cylindrical shape is one of the strongest 3 dimensional beam-type shapes, because it has no sharp corners, so can spread stress evenly across its cross-section [1]. *Figure 3* shows the results of this analysis. We must keep an appropriate aeronautical engineering safety factor, which means it must be above 1.5 [2]. The lightest spar geometry that meets this criteria, and the one that we are proposing, is a hollow cylinder with outer radius of 190mm, and a thickness of 15mm. This will have a mass of 60.4kg, a safety factor of 1.58 and a maximum strain of 0.44%

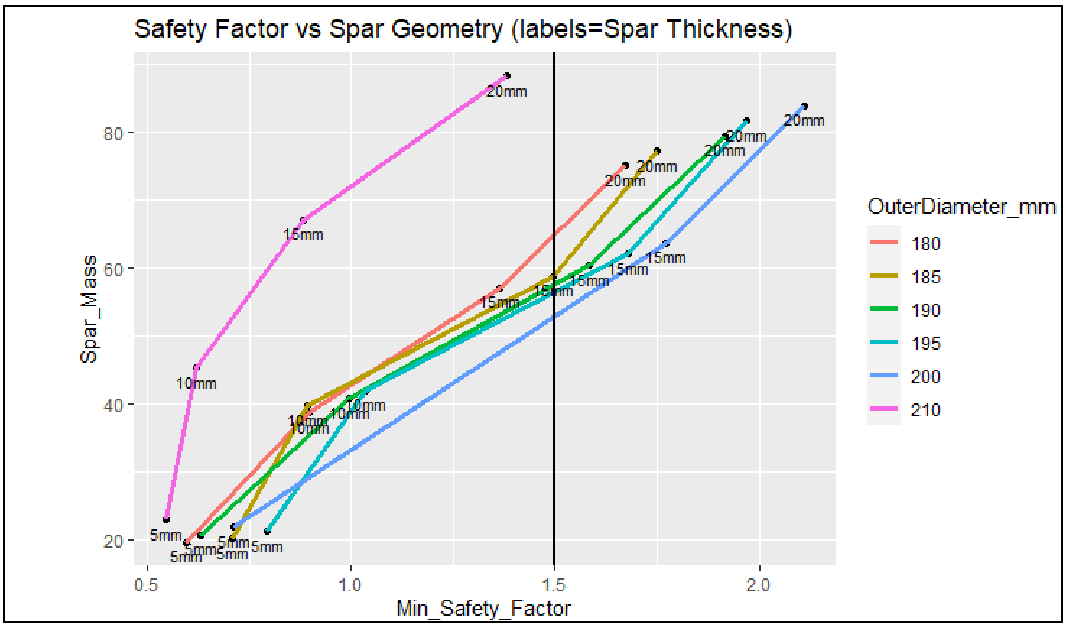

Figure 3: Spar Mass and Minimum Safety Factor of different spar shapes and sizes

For our simulation approach for both the aerodynamic and structural problems, we had to make a number of assumptions. These include assuming wind and other weather factors are negligible, assuming the aerodynamic analysis can be simplified to a 2 dimensional problem, assuming gravity could be neglected for the structural analysis, assuming the aircraft flies at a constant horizontal velocity, assuming the entire airfoil is made of aluminium-7075 alloy, which we are assuming can be modelled as a perfectly linear-elastic material (LINK). We also use a mesh discretization to allow us to numerically solve the aerodynamic and structural situations, which is a simplification that comes from the fundamental assumption of continuity.

Information for Technical Expert

Aerodynamic Analysis:

For the aerodynamic analysis, we used the ‘Fluent’ package within ANSYS to conduct Computational fluid Dynamics (CFD) modelling. This package is one of the most versatile and powerful fluid modelling packages so suited our purposes well. To model the aircraft flying at a constant horizontal velocity of 65m/s, we placed the aircraft stationery in our domain, and used a horizontal airflow of 65m/s to maintain this relative velocity. For the reference values of the air fluid, we used standard atmospheric values for air at the given altitude of 1400m. We also used the SIMPLE solver scheme in Fluent. We set the spatial discretization gradient to least squares cell based and set the pressure to second order. The other SIMPLE options were set to second-order-upwind.

The inlet-velocity of 65m/s was our first boundary condition. We created a no-slip condition around our airfoil as the second boundary condition, which is a fundamental assumption for fluid modelling [6]. At the top and bottom edges of our domain, there is a free-slip boundary condition, so we included a no shear-stress condition for these edges. For the outlet edge, we added our last boundary condition, which was a 0 gauge-pressure condition, based on our assumption that the air pressure eventually returns to atmospheric pressure after interacting with our airfoil.

Next we had to determine the optimal size and shape of our domain, which was large enough to give us accurate results, but small enough to run quickly and efficiently. To do this, we conducted domain convergence analysis, where we modified the domain size, and investigated how the coefficients of lift and drag were affected by the change. We chose to use a domain which extended much further to the right than left of our wing, as the majority of turbulent flow occurs after contact with the airfoil, and this is a region of interest for us. The results of our domain convergence are shown in *Figure 4*, which shows that the results converged for a domain area of 288 m^2 (length of 24m and height of 12m).

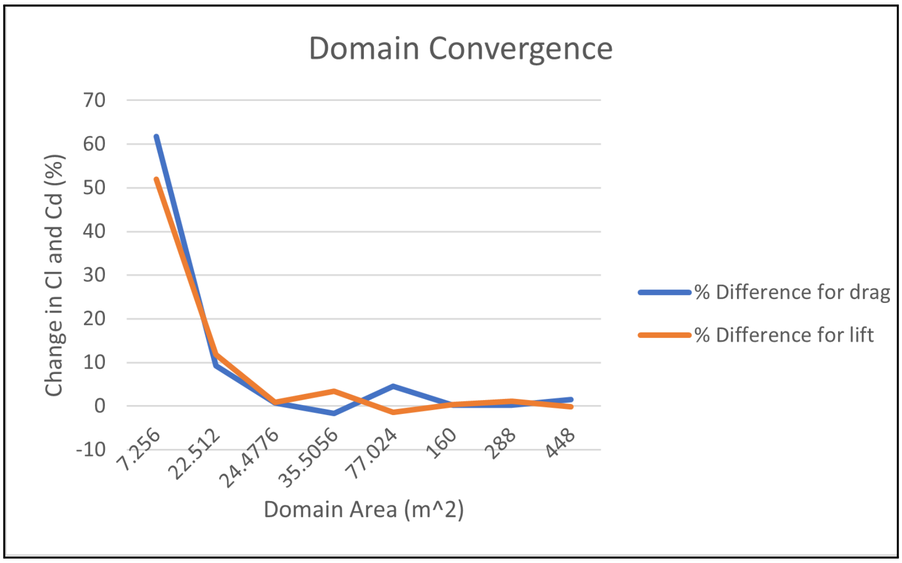

Figure 4: Domain Convergence Plot

Next we had to determine the most accurate yet still efficient settings for our CFD mesh discretization. Because our domain is quite large, it was not feasible to have a constant mesh size throughout our domain. We decided to use inflation layers in the main areas of interest around our airfoil, which would allow us to solve more accurately in this region, while keeping the size of our mesh elements larger further away from our region of interest, to increase our efficiency. Again we did a convergence analysis to determine the optimal mesh size, which we determined to be 3.27mm. This had a low enough percentage change to satisfy our accuracy requirements, while taking much less time to solve than the 2.10mm or 1.09mm options. Our final mesh had 93,163 nodes, and was solved quadratically with triangular elements.

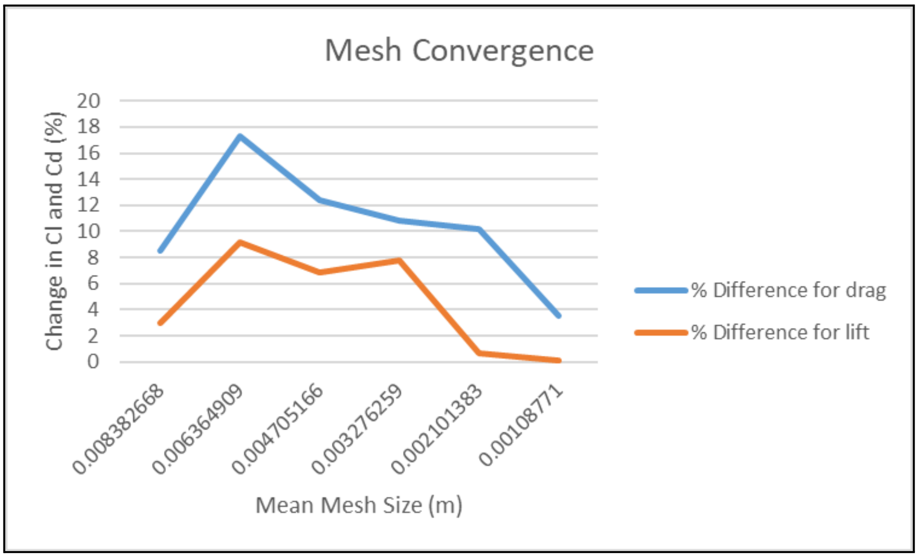

Figure 5: Mesh Convergence plot for CFD analysis

Now that we had determined the mesh configuration and domain, we could begin solving for our values of interest for different angles of attack. *Figure 6* (below) shows our domain and mesh that we used, and has the airfoil at an angle of attack of 5 degrees.

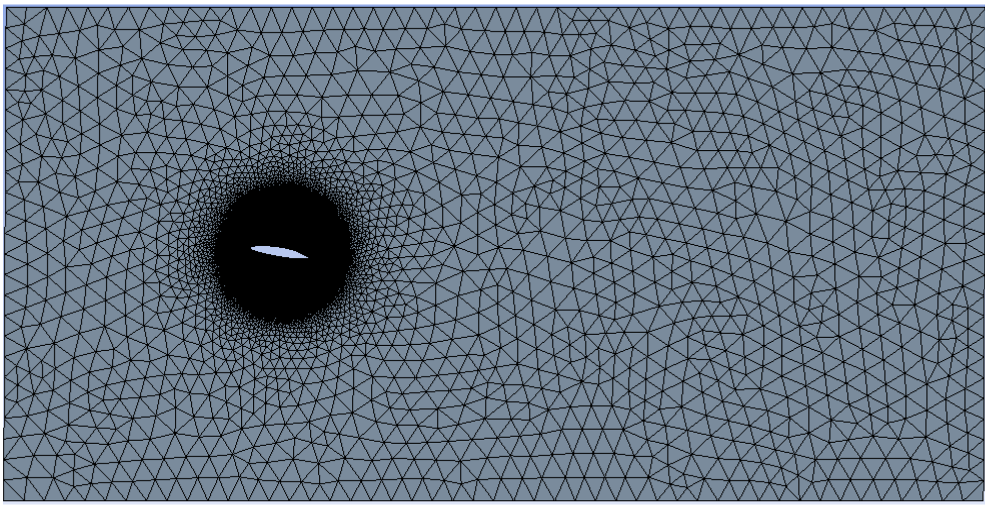

Figure 6: Mesh discretization of our domain for our CFD analysis

After running this analysis on multiple possible angles of attack, we plotted and interpreted our results. We also verified our results by comparing them with the 'reference airfoil data' provided. Figure 7 shows how our simulation results compared with the reference data. The similarities between these results is very encouraging, and helps to validate our result.

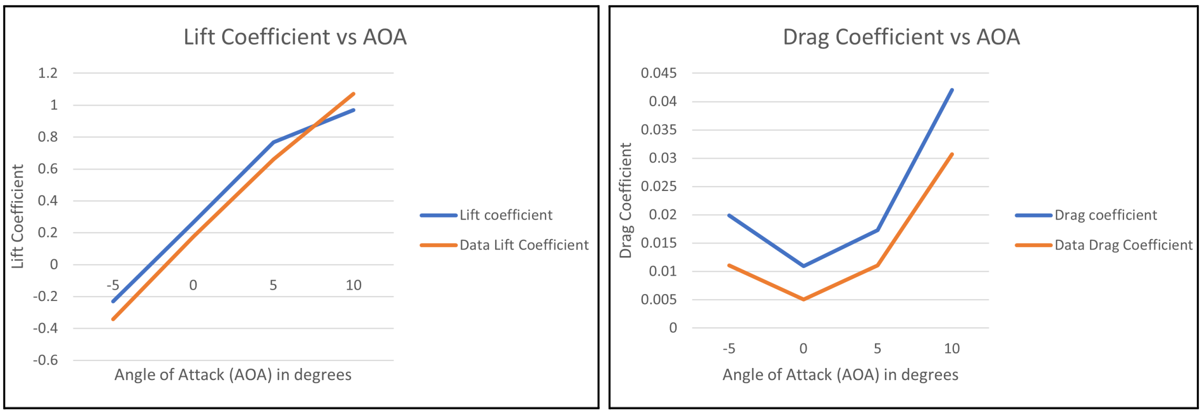

Figure 7: Lift and Drag coefficients by angle of attack compared with reference data

The next steps for the aerodynamic analysis would be to run more simulations, for different angles of attack. This would allow us to more confidently verify results and solve for a more precise estimate of the airfoil's 'stall angle'. We should also solve this problem in 3 dimensions, to ensure that it is correctly modelled. From here we could begin to model the rest of the plane, as well as only the wing, which would also greatly improve the accuracy of our results.

Structural Analysis:

We used the Structural Finite Element Analysis module in ANSYS to solve the structural problem. We have assumed for this analysis that thermal effects are negligible, that the force applied is steady, and that the stress-strain relationship of Al-7075 alloy is purely linear-elastic. The elastic limit of Al-7075 is approximately 1% strain, so if our model undergoes more than 1% strain, this assumption will have been broken. We monitored the maximum strain throughout, and ensured it stayed below 1%. When creating the geometry for our structural analysis, we took the cross-section of the airfoil, and created 5 'ribs' of this shape connecting a central spar to an outer skin.

To find the optimal mesh size, we again used mesh convergence. We tested the change in value of maximum deformation, not maximum stress, because there is the possibility of encountering 'stress singularities' in our solution because we have sharp corners in our geometry. For the structural problem we tried both linear and quadratic mesh arrangements to find which was the most efficient option. *Figure 8* shows this result, which shows that the quadratic solvers have far more nodes than the linear solvers, even for the same mesh size, and do not improve the results enough to encourage the increased solve-time. We decided to use linear 50mm mesh, as this gives accurate results, but is far more efficient than using smaller mesh elements.

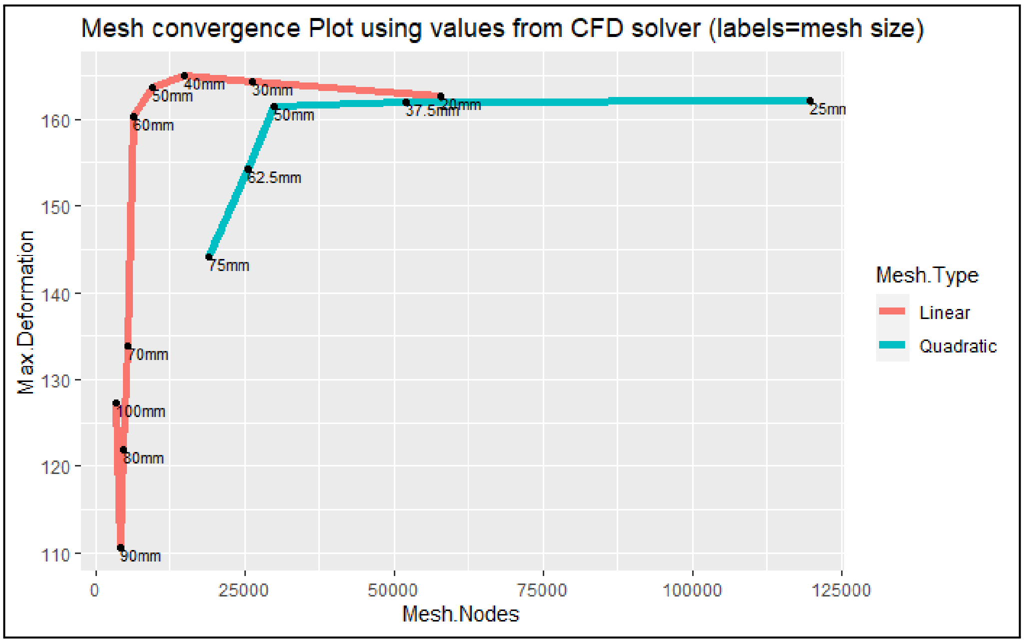

Figure 8: Structural Mesh Convergence Plot

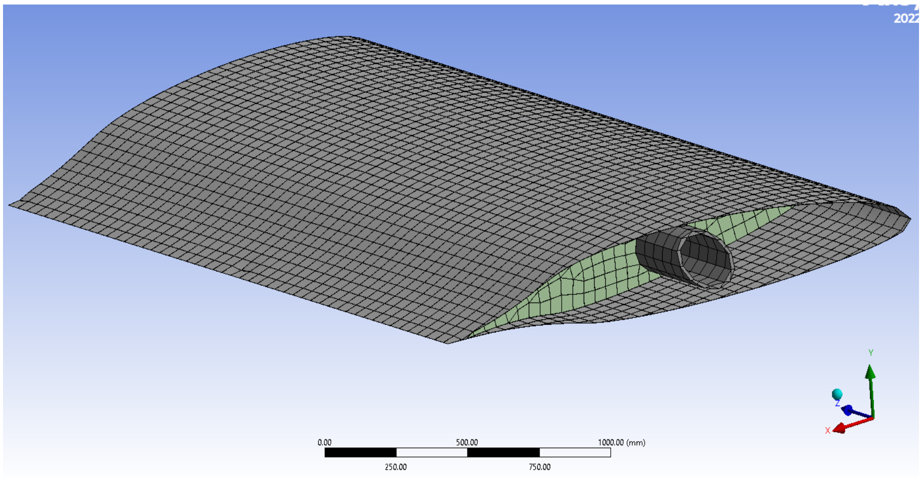

Figure 9: Mesh Discretization of Structural geometry

After this, we decided to evaluate the effect flying at different angles of attack has on the maximum equivalent stress in the airfoil. After removing outlying values, we used *Figure 10* to determine that the stress in the airfoil is highest when flying at an angle of attack of 0 degrees. However, using our results from the CFD analysis, the lift force is much greater for larger angles of attack, which far outweighs this small difference. Therefore we decided to continue the structural analysis at an angle of attack of 5 degrees, as suggested in the project brief.

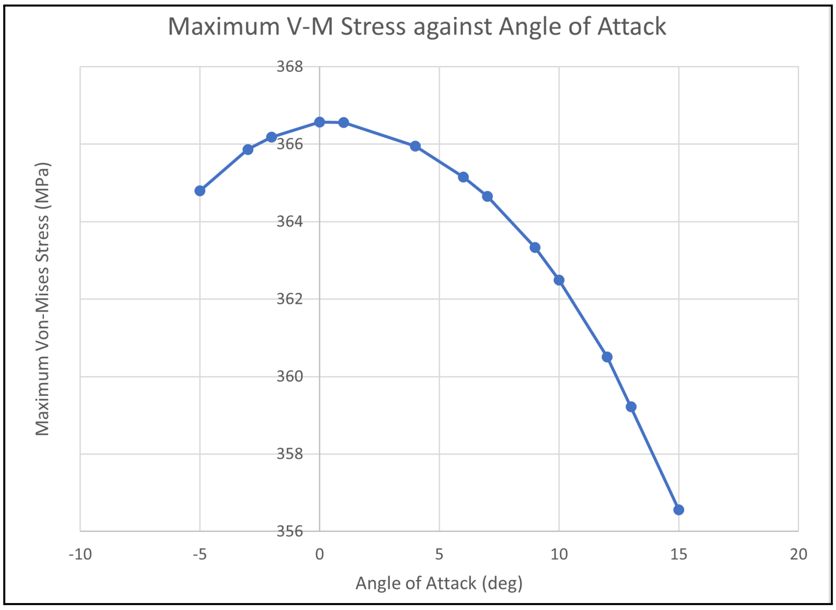

Figure 10: Maximum Equivalent Stress at different angles of attack (for constant lift force)

For the boundary conditions in our structural analysis, we added a fixed support to the face of the spar which is attached directly to the aircraft, as this must remain attached at all times. We also applied the lift force and drag force that we found from the CFD analysis at an angle of attack of 5 degrees, onto the top and bottom faces of the airfoil [3]. We are assuming that the force is evenly distributed along the wing, which is not entirely reasonable, as the nose of the plane is likely to have aerodynamic effects that interfere with the lift generated near to the plane.

Next we solved our structural problem for many different spar geometries, with the aim of finding the geometry that minimises the mass of the spar while maintaining a viable safety factor. An appropriate safety factor for aeronautical engineering was found to be 1.5 [2]. *Figure 11* shows the results of these analyses, where the spar mass is plotted against the minimum safety factor. From these results, we selected the geometry which had the lowest mass, but had a safety factor greater than 1.5. This led us to select a hollow cylindrical spar with a circular outer diameter of 190mm and a thickness of 15mm. The maximum strain of this model was 0.44% which is well below the 1% maximum, so it appears that our assumption of purely linear-elastic behaviour is accurate. Full results of the analysis of our chosen spar can be found in the appendices.

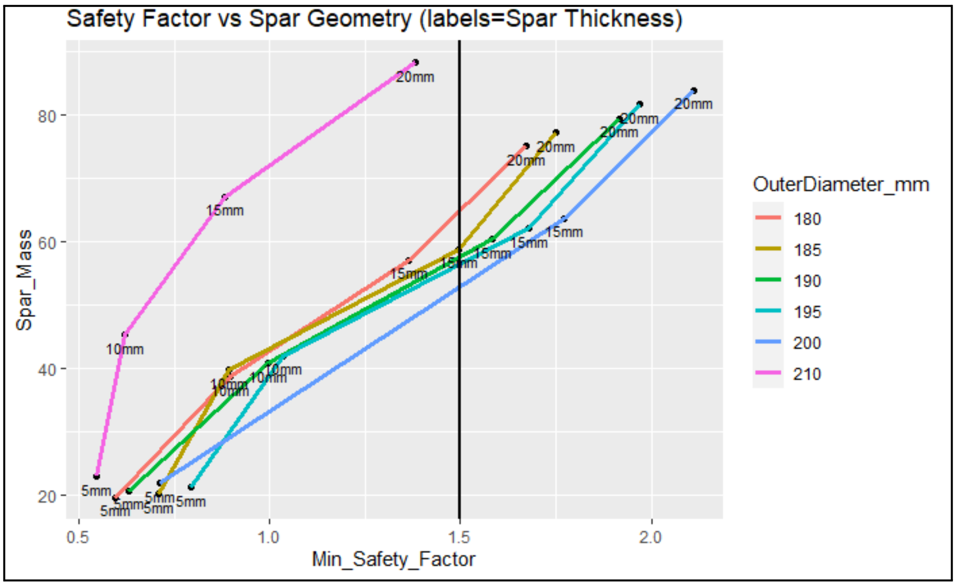

Figure 11: Spar mass against minimum safety factor (for many spar geometries)

To validate that our solver is working correctly, I used a baseline test where I solved for the maximum deformation of a 10in diameter, 5m long solid bar subjected to 1 N/m force distribution. I compared the results I got from the ANSYS structural solver to the results provided by an online beam deflection calculator [5]. Ansys predicted 46.25mm maximum deformation compared to the calculator's prediction of 46.41mm. This is only a 0.3% difference, which verifies that we have solved the problem correctly.

We would not expect such high accuracy for our full airfoil model because it is a far more complicated model where we have had to make more simplifications. To improve the accuracy of our results, the best thing to do would be to model the structural performance of the whole aircraft. The gravitational effects also should not be ignored, as this would be a significant force when modelling the entire mass of the aircraft. It would also be important to not just model the aircraft in steady horizontal flow, but also to model non-steady takeoff and landing procedures, to ensure the aircraft is structurally sound during the whole flight. For both the CFD and structural problems, it would also be good to treat wind as a parameter, to allow for testing under a variety of weather conditions.

As with all computational analysis, there may be slight inaccuracies introduced to our model by the assumptions we have made, so continued analysis and verification will be needed. In particular, I would sometimes encounter 'outlier' values when solving for the minimum safety factor in the structural problem. For example, *Figure 12* shows how according to my solver, having a 10mm thick cylinder would result in a worse minimum safety factor than a 5mm thick cylinder, for an outer diameter of 200mm. This does not make sense, particularly as all of the other results were following a very clear trend, so I decided this must have been due to a flaw in the computational model, and removed it from my dataset. It is possible that for this specific geometry, we were encountering a stress-singularity which was causing erroneous results.

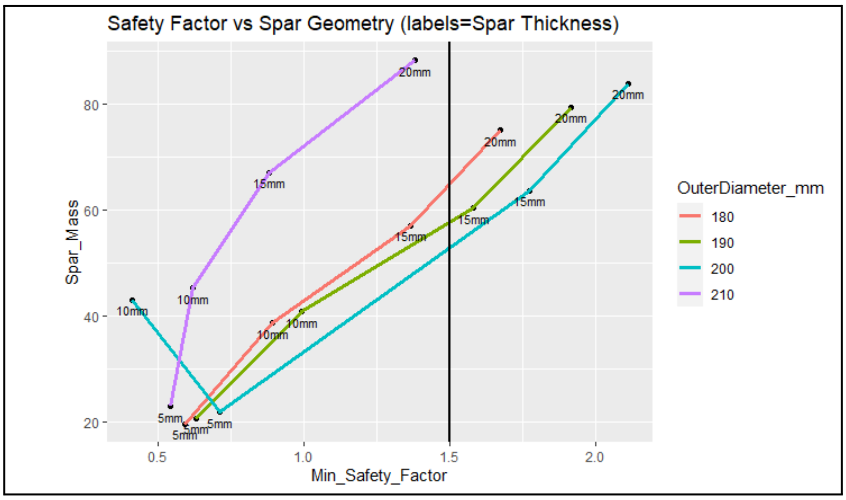

Figure 12: Outlier value included in my initial results

Personal Reflections

- Overall I found this project to be an interesting one. There was a good balance reached between the help that the lecturers and TAs provided, and the parts which were left open to interpretation.
- We encountered many challenges during the project, possibly the most difficult part of which was the creation of the pressure plots C_p against angle of attack. This turned out to be a large headache but eventually Michael managed to get them working well.
- One thing that I initially found daunting but am now comfortable with is the interpretation of results in ANSYS. Creating output plots, and using parameters to output a range of values became a very comfortable thing for me throughout the process, and I am confident I could apply this again in the future.
- I found the geometry generation to be the most difficult part of this project. During the structural analysis I attempted to create spars of non-uniform thickness

or shape, but found this to be extremely difficult. I often seemed to end up with more components than I expected, which interfered with each other and sometimes caused the meshing to fail. In the end I decided to stick to a uniform cylindrical spar as this was something which I was more confident in creating its geometry.

- One strategy which I developed was about running one model, before attempting to run many models. This is helpful because I could give the results a quick check, and ensure everything is working correctly before expanding the parameter space and running multiple iterations.
- I also believe that the divide-and-conquer approach that Michael and I took was a successful one, because he was able to focus on the CFD analysis and I was able to focus on the structural analysis, allowing each of us to complete these both to a higher level of accuracy than if we had been sharing our attention between the two problems.

References

[1] Katelyn E. Shipp J0227 - California science and engineering. https://csef.usc.edu/History/2007/Projects/J0227.pdf

[2] The ultimate factor of safety aircraft spacecraft history ... https://ntrs.nasa.gov/api/citations/20150003482/downloads/20150003482.pdf

[3] Www.iop.org/journals/physed how do wings work? - department of engineering. https://www3.eng.cam.ac.uk/outreach/Project-resources/Wind-turbine/howwingswork.pdf

[4] 2.972 how an airfoil works. https://web.mit.edu/2.972/www/reports/airfoil/airfoil.html

[5] Beam analysis (nd) *MechaniCalc*. https://mechanicalc.com/calculators/beam-analysis

[6] No slip condition (nd) *StudySmarter UK*. https://www.studysmarter.co.uk/explanations/engineering/engineering-fluid-mechanics/no-slip-condition/

Appendices

	A	B	C	D	E	F	G	H	I	J
1	Spar_Outer_Diam	Spar_Inner_Diam	Mesh_Size	AoA	Max_Deformation	Max_Strain	Max_VM_Stress	Min_Safety_Factor	Spar_Thickness	Spar_Mass
7	190	175	30	5	169.778234	0.004413053	317.7398096	1.583056277	15	60.41577197

Figure 13: Results of our final analysis for our chosen spar geometry (structural)

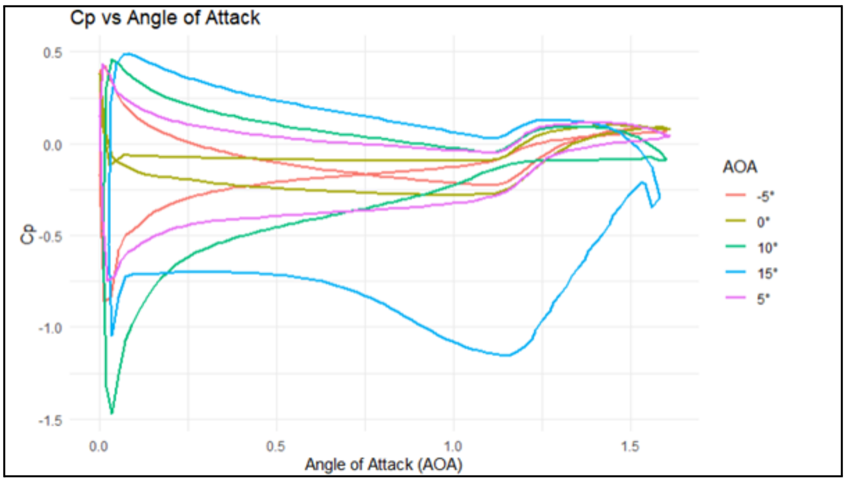

Figure 14: Plot of Coefficient of Pressure C_P against angle of attack