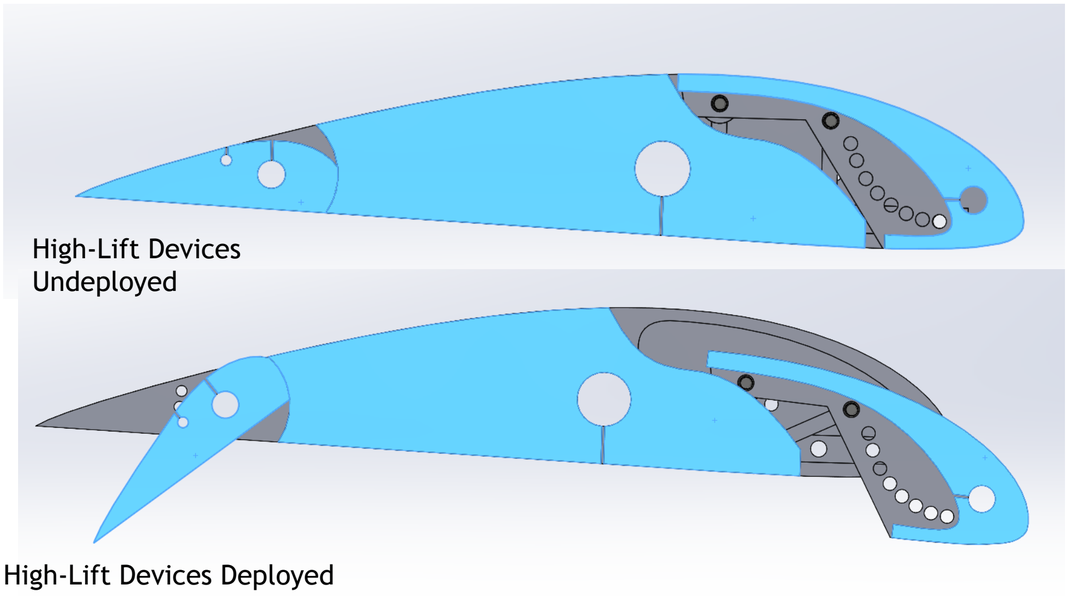
Design, Manufacturing & Testing of A Wing with High Lift Devices

Tan Yee Jie (31145914) - Supervised by Prof Andy Keane

Introduction

The project consists of building a testing wing with moving mechanism (high-lift devices) that changes the physical geometry of the wing to maneuver the aerodynamic characteristic of it. The wing is put into a wind tunnel to conducted tests and the results are discussed & compared using computational methods.

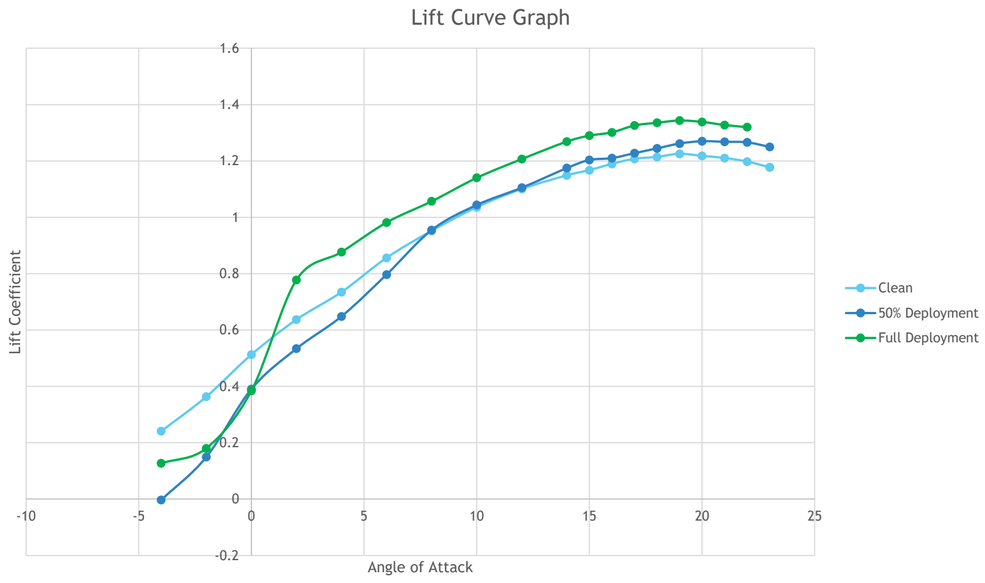


High-Lift Devices Undeployed

High-Lift Devices Deployed

The wing are mounter vertically on the force balance of the wind tunnel to obtained the force reports. The results are shown in the graph below.

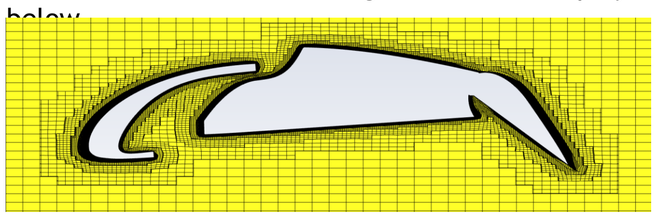
Wind Tunnel Result



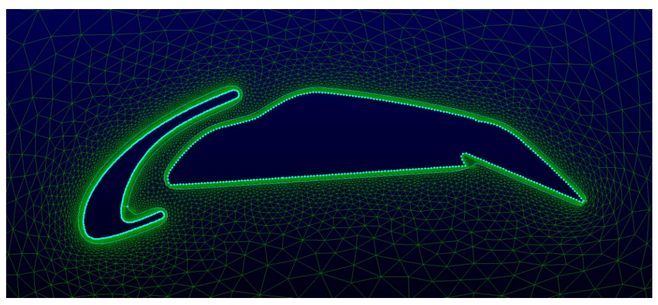
The graph above states that 100% deploying the high lift devices, the lift coefficient only increases by an amount of 9.71% which doesn't not reach the expectation. Thus, by further looking into the case, it is realised that the leading-edge slats are not contributing to the increment in lift. As this result has clearly not corresponded to the expectation and a further study into the flow field is needed to understand the abnormal performance of the slats.

CFD Methods

Meshes of the airfoils with high-lift devices deployment is created as shown as below.



Harpoon Mesher



Fidelity Pointwise

Both the meshes have their wall-yplus lower than 1 to increase the accuracy of predicting the flow field. Both meshes are created for different usage, the harpoon is for more accurate force prediction and the pointwise mesh stresses more on quicker demonstration of the contour-plot for different case and the force prediction of it is not the focus.

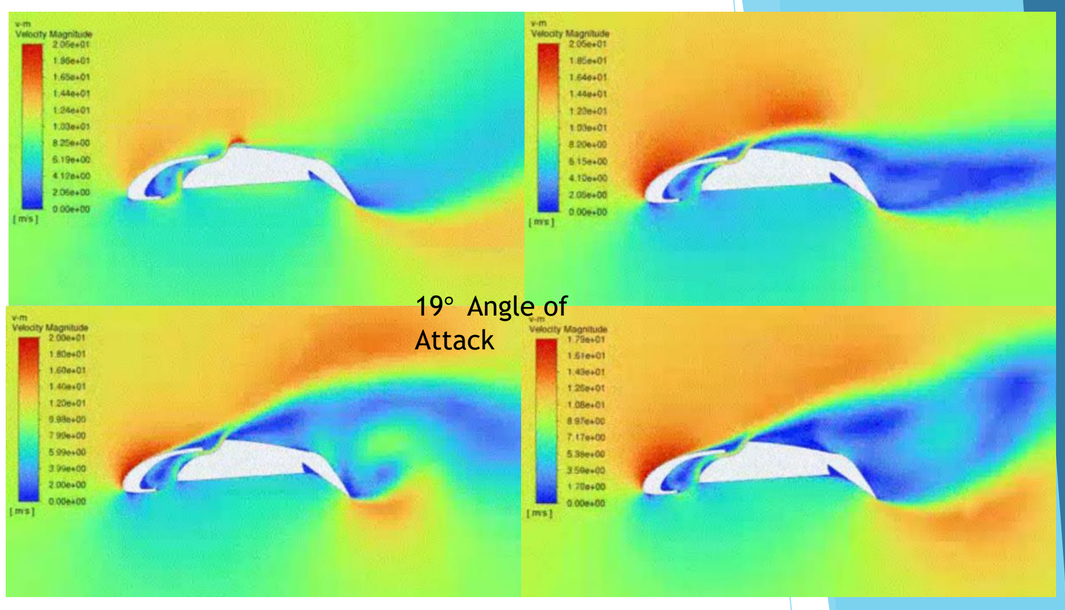
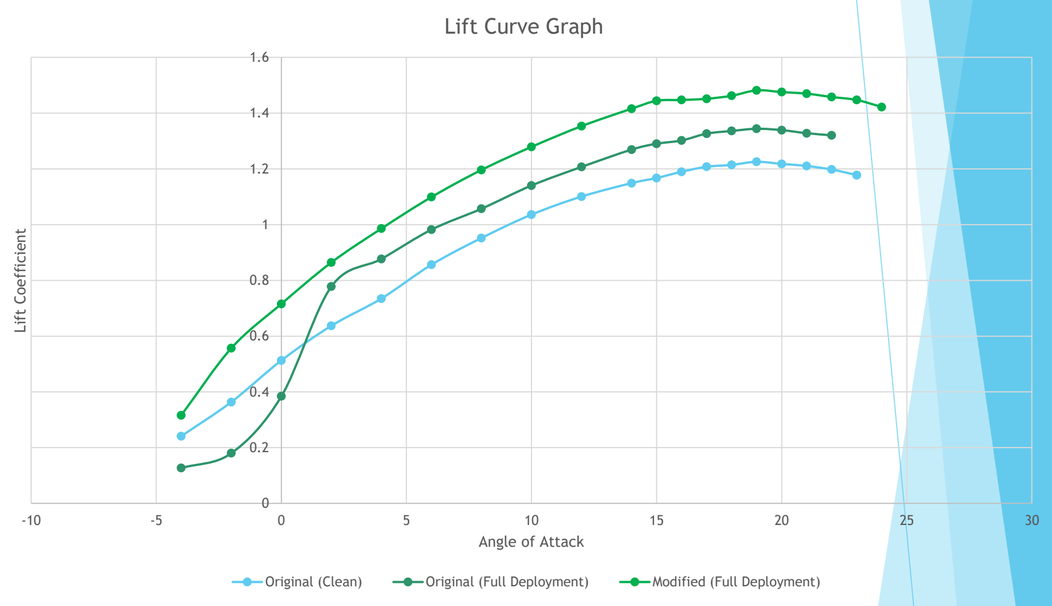


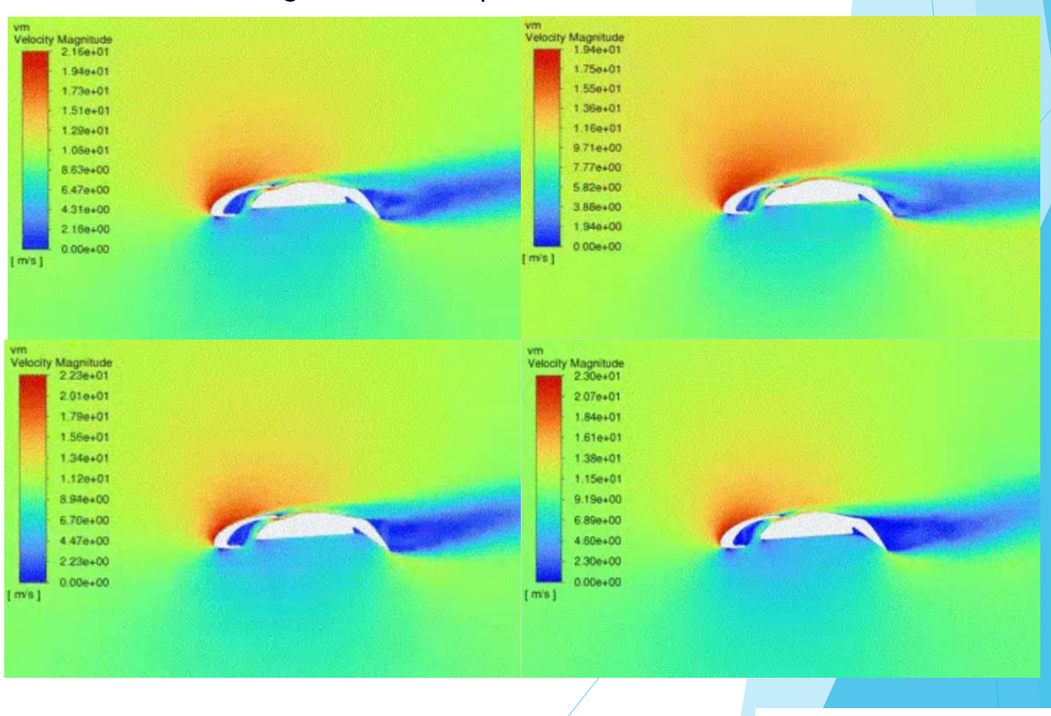
Figure above shows the velocity magnitude contour plot after conducted a transient approach to mimic the flow around the high-lift devices to investigate the flow behavior around the high-lift devices. Notice that the flow after the leading-edge slats (on the main body region) on the upper surface of the airfoil is the mixture of the low velocity flow from the lower surface and high velocity flow from the upper surface of the leading-edge slats. Due to the sharp corner, the flow mixing angle are too high, causing the flow to separate, thus leading to the loss of lift generated from the airfoil.

Modifications

Since it is the sharp edge of the body that cause the incorrect angle of flow, it is sanded off to provides the flow a smoother curve to be attached on. A comparison of the result of the lift coefficient are plotted as below and the difference between the cases can be easily observed.



To relate the increase in lift and the flow mixing angle, CFD analysis are again conducted using the same method. The velocity magnitude contour plot below have demonstrated that the flow are much more attached in the same given angle of attack after sanding down the sharp corner.



Conclusion

The project has introduced several major constraints towards the designing process of the high-lift systems. The angle of deflection of the leading-edge devices, is one of the major concerns. More process should be done beforehand to align the flow of each element of the airfoil to ensure the boundary layer stay attached longer. To further improve the performance, an amount of the slotted flaps can be increased as well as the angle of deflection of the flaps.

Scan the QR for the full detailed report for this project !!