4-30-2018

Computational Fluid Dynamics Analysis of High Lift, Inverted Airfoils in Ground Effect

Michael Grabis
Washington University in St. Louis

Ramesh K. Agarwal
Washington University in St. Louis

Computational Fluid Dynamics Analysis of High Lift, Inverted Airfoils in Ground Effect

Michael M. Grabis[1] and Ramesh K. Agarwal[2]
Washington University in St. Louis, St. Louis, MO, 63130, USA

Formula SAE vehicles are formula styled (open-wheeled and open cockpit) racecars that are designed to race on an autocross circuit. Highly competitive vehicles in the competition implement aerodynamic devices, which generate negative lift for the vehicle. This negative lift, or downforce, increases the amount of traction between the racecar's tires and the ground that ultimately allows drivers to turn at faster speeds. Commonly used aerodynamic devices are a front and rear wing; the wing cross sections are defined by configurations of multiple 2D airfoils. This paper focuses on the systematic design of a Formula SAE specific front wing through the comparisons of high lift, inverted airfoils, in ground effect in order to maximize the negative lift coefficient. Five selected high lift, single element airfoils are iterated through multiple angles of attack and the three superior airfoils are iterated through a second study of height off the ground. A third study begins to look at combining the single airfoils into a two-element airfoil configuration to further increase negative lift generation.

I. Nomenclature

a	=	angle of attack, between chord and line parallel to ground
C_D	=	drag coefficient
C_L	=	lift coefficient
c	=	chord length
h	=	height (off the ground)
Re	=	Reynolds Number

II. Introduction

Formula SAE is an annual student engineering design competition centered around creating a formula-styled racecar. These cars compete in Formula SAE competitions world-wide and get evaluated on their design in static events as well as their actual performance in dynamic events, the final of which is a 22 km Endurance Race. Competition speeds can be considered low velocity racing; average speeds range from 40 to 48 km/hr while top speeds in a straightaway can reach around 100 to 105 km/hr [1]. The Endurance lap is mainly comprised of slalom sections, while an average course contains at least 3 straight-away sections. Although the racing is most frequently at a low average velocity, aerodynamics play a large part in vehicular design.

Generally speaking, a car body generates positive lift. Positive lift hinders high speed cornering ability due to a decrease in normal force (traction) the tires have with the ground. Aerodynamic devices can be implemented to negate this positive lift. Many aerodynamic components such as a front wing, a rear wing, a flat underbody tray, and rear diffusers can be designed to generate negative lift that counters positive lift. A front wing is a logical first design step because it has the most rules and placement constraints; when a satisfactory negative lift coefficient is achieved, a rear wing must be designed to couple the forces at the front and rear of the vehicle, effectively pushing the center of pressure towards the rear of the vehicle. This paper studies an iteration of 2D airfoil geometries to increase negative lift generation of a front wing attached to Formula SAE car's chassis.

The front wing design begins with a 2D cross section of an inverted airfoil that is located in close proximity to the ground. Airfoils behave much differently in close proximity to the ground than they do in free-stream air; pressure/velocity changes of flowing air due to the ground are called "ground effects". The shape of the inverted airfoil with the ground creates a venturi having an inlet shaped to increase velocity/decrease pressure and an outlet which

[1] Undergraduate Student, Department of Mechanical Engineering and Materials Science, AIAA Student Member
[2] William Palm Professor of Engineering, Department of Mechanical Engineering and Materials Science, AIAA Member

decreases velocity/increases pressure gradually. A primary study will be about the effects of increasing angle of attack, a, of multiple airfoils at a set chord and h, height off the ground. A second, single-element study is varying the airfoil's h, which is conducted on airfoils with the same chord length as the first and the optimal angle of attack a = 6º and the same chord length, c = 0.33 m. The final study, which is still currently being investigated, involves a two-element airfoil configuration that further increases the negative lift generation potential of the front wing over just a single element.

III. Geometry

Front wings on Formula SAE cars are created from extruded airfoil geometries. There are limitations on the size and location of a front mounted aerodynamic device, most notably that it cannot be "further forward than 700 mm forward of the fronts of the front tires" and "when viewed from the front of the vehicle, the part of the front wheels/tires that are more than 250 mm above ground level must be unobstructed". One must also consider the additional planar area from movement of the tire when the wheel is turned - therefore no part of the wing can enter the "keep-out-zone defined by…positions 75mm in front of and 75mm behind" [1] the front wheel. These rules set a general constraint on sizing and height of a potential front wing geometry.

Since this study primarily looks to maximize the downforce, or negative lift, that is generated by a single element airfoil in ground effect, the most efficient way to begin this process is by utilizing previously published high-lift airfoils (versus creating unique geometries from individual points and curves). The following five high-lift airfoils were examined:

- (CH10) Chuch Hollinger CH 10-48-13
- (E423) Eppler E423
- (FX74) Wortmann FX 74-CL5-140
- (LA5055) Liebeck LA5055
- (S1210) Selig S1210

All point data files for each airfoil were sourced from the UIUC Applied Aerodynamics Group Airfoil Coordinates database [2]. Point data files were imported into Microsoft Excel, formatted for ICEM import, and imported into the ICEM meshing software directly as a curve through the x and y coordinate points defined for each airfoil. Curves were fit to these points and smoothed to eliminate flat edges between the points.

The first case study was varying angles of attack with a constant height off the ground of h = 0.106 m. Each airfoil was iterated from a of 0º to 15º, as those are reasonable angles for a main element of an FSAE front wing based off of the rule constraints as well as the upper bound being the typical region where flow begins to separate from the trailing edge of the airfoil.

The second study was a height study to determine how the ground effects an airfoil at low velocities. The three airfoils with the highest $-C_L$ values from the first study were chosen for the second ground effect study. In this study, a = 6º for each run, while h was iterated from 0.0127 m to 0.9144 m with smaller changes in height between runs the closer the airfoil was to the ground (to increase data resolution).

The third study selected the superior single element airfoil to be the main element of the two-element airfoil, keeping c = 0.33 m, a = 6º, and h = 0.106 m while the three superior high lift airfoils with c = 0.1524 m (around 40% the c of the main element) were selected as a flap geometry. The lowest point on each flap airfoil is 0.015 m above the trailing edge of the main element (roughly 5% of the entire c of the two-element airfoil); both chord and height percentages are given in Competition Car Aerodynamics [3]. The top flap is then iterated through angles of attack from 20 to 50 degrees in increments of 10º; the 50º value reaches the height limit constraint of an FSAE front wing in front of a wheel. Horizontal spacing is adjusted so that the lowest point of the flap is still above the trailing edge of the main element.

IV. Grid Generation

The airfoil geometries were imported into the ANSYS meshing software ICEM for its superior control over creation of a structured grid. Instead of a traditional C-Grid that is normally used for airfoil geometries, a rectangular domain was chosen to be able to capture the ground effect forces the airfoil itself would be seeing. The bottom of the domain, directly under the airfoil, is modelled at a h away from the lowest point on the airfoil's lower surface to simulate the ground. The far field outlet was set at 20c rearward from the trailing edge of the airfoil while the far field inlet and far field top were set at 10c forward and above of the leading edge of the airfoil, respectively. Figure 1 shows the entire domain while a detailed view is presented in Figure 2. An initial grid spacing corresponding of a Y+ of 1 was chosen based off simulation parameters to proper capture flow for the selected turbulence model. Figures 3 and 4 show the entire domain as well as a close up view of the two-element airfoil geometry.

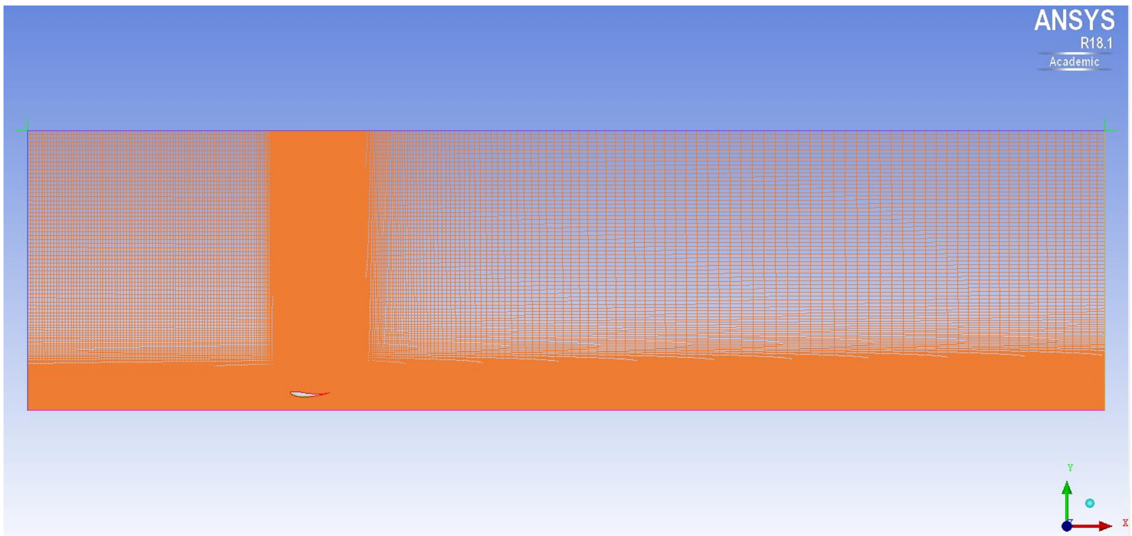

Figure 1 Entire rectangular domain for single-element.

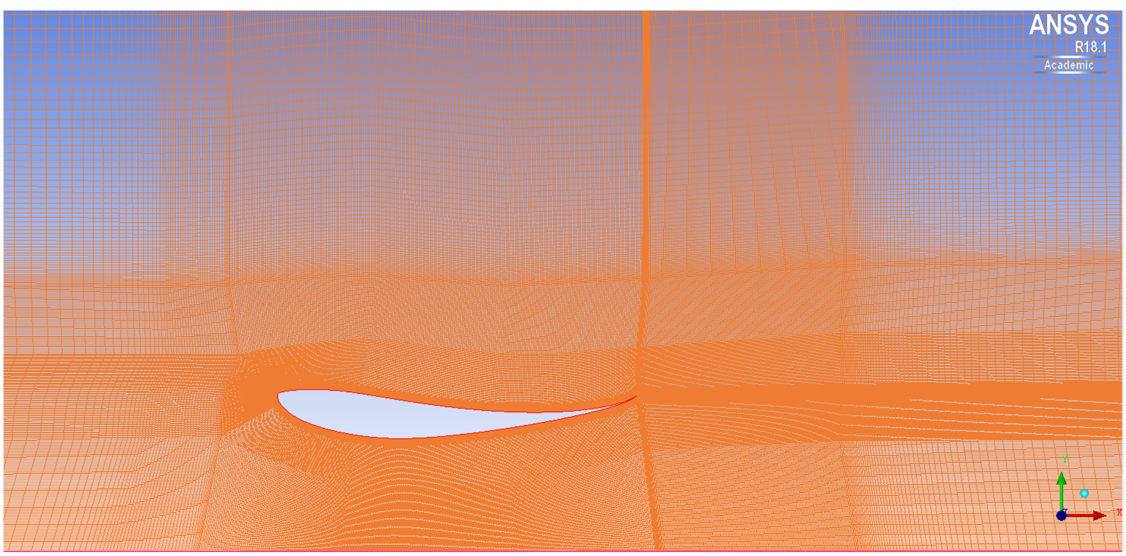

Figure 2 Close view of grid around the single-element airfoil geometry.

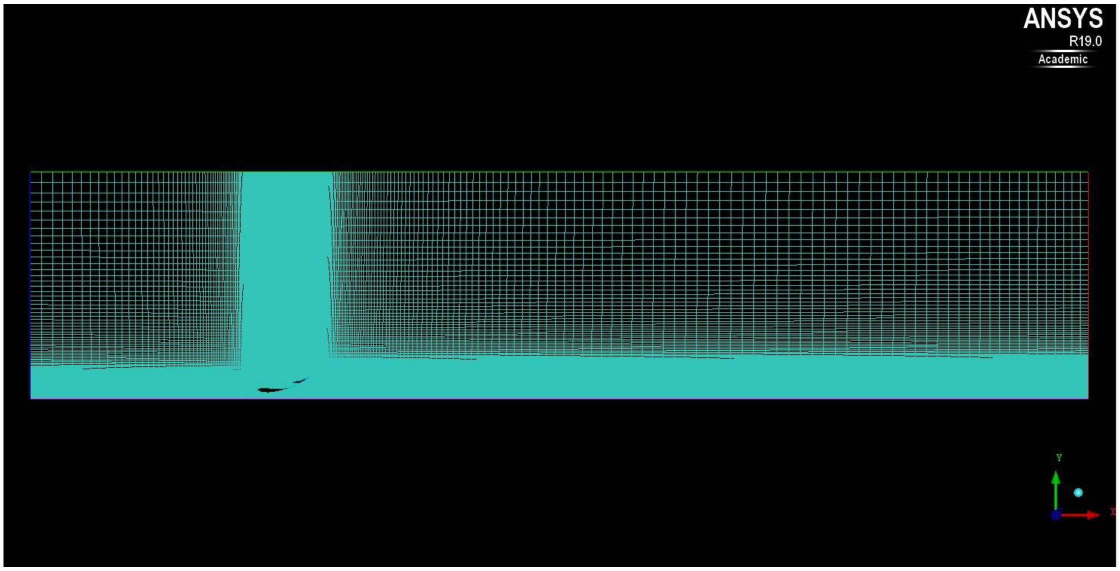

Figure 3 Entire rectangular domain for two-element.

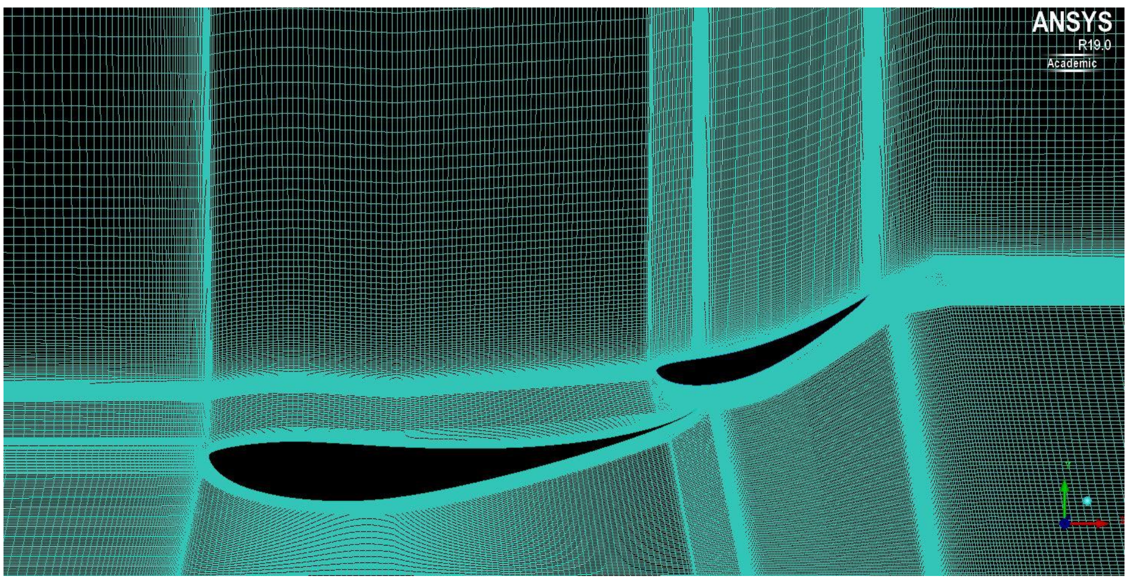

Figure 4 Entire rectangular domain for two-element.

A mesh independence study was created to determine the accuracy of various mesh densities. The benefit of a courser mesh is shorter run times, but there is risk of error in the calculated value. The mesh independence study run was simulated on the CH10 airfoil, a = 0º, h = 0.106 m with three mesh densities spaced apart by 100,000 elements. The results are shown in Table 1.

Table 1 Mesh independence study for three mesh densities.

Elements	C_D	C_L
150,000	0.04372	-1.11125
250,000	0.04371	-1.11307
350,000	0.04370	-1.1131

It is evident that the values for C_D and C_L do not vary greatly between the mesh densities. A mesh density of 150,000 was chosen to decrease run times so that more simulations can be completed. For the two-element case, a mesh density of 180,000 cells was chosen in order to refine the geometry around the flap element while still maintaining a similar density around both elements as appeared in the single-element case.

V. Numerical Setup of CFD Solver

ANSYS Fluent is the flow solver of choice for running the flow simulation. The 2D airfoil cases were output from ICEM with the proper boundary conditions of velocity for inlet, pressure for outlet, symmetry plane for the top of the domain, and wall conditions for the airfoil as well as the ground. The energy equation was selected on and the k-omega SST turbulence model was chosen for the simulations. The SST model, specifically, gives highly accurate predictions for regions experiencing flow separation and increased accuracy in boundary layer simulations. Due to the lower velocity nature of the race, the low-Re corrections were also selected for the viscous model.

Air was set as an ideal gas due to the incompressible approximation of air flow and the viscosity was set to Sutherland. The flow inlet speed was set at 14.5735 m/s (chosen from the average speeds stated in the FSAE rules) and the wall boundary condition was set as a moving wall, mimicking a moving ground, with a velocity of 14.5735 m/s.

The Pressure-Velocity Coupling solution method was chosen. It was first iterated with density, momentum, turbulent kinetic energy, specific dissipation rate, as well as energy set to first order upwind for 1000 iterations to improve speed of convergence. Once completed, the aforementioned factors were set to second order upwind and iterated until 10,000 iterations or convergence of all residuals to 1e-5.

VI. Numerical Results for 2D Airfoils

The Reynolds number for both runs was 295,000, assuming air properties in close proximity to sea level, a reference chord length of 0.33 m, and a desired y^+ of 1 for the SST k-omega turbulence model. The initial wall spacing for the boundary layers around the airfoils was set at 1e-5 m to adequately calculate the boundary layer's effect on the airfoil. The primary goal in the first study of angle of attack was to determine which airfoils have the highest C_L and at which angles of attack that peak value occurs, when subjected to a ground effect. Figure 5 shows how changes in angle of attack effect the negative lift (downforce) coefficient of each of the five airfoils examined and Figure 6 shows how changes in angle of attack effect the drag coefficient for each airfoil.

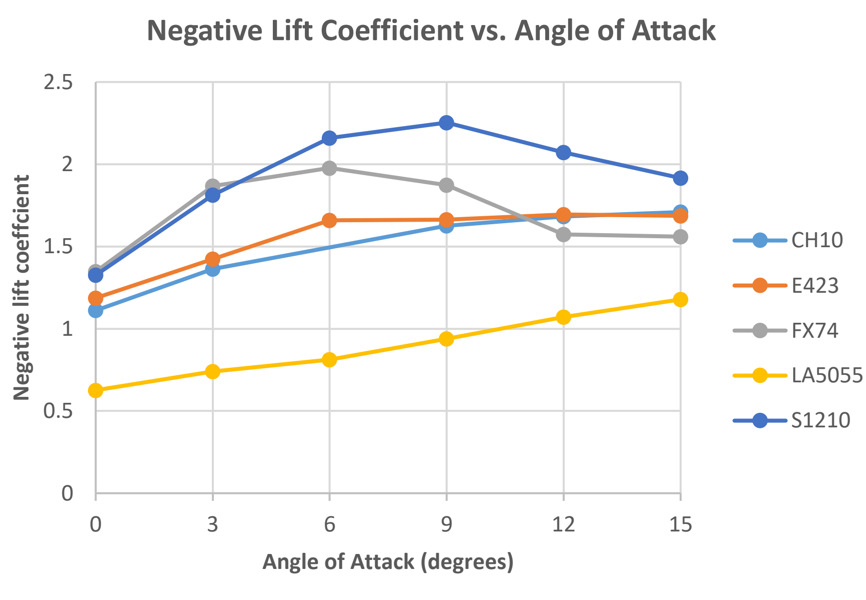

Figure 5 Numerical values for negative lift coefficient at various angles of attack.

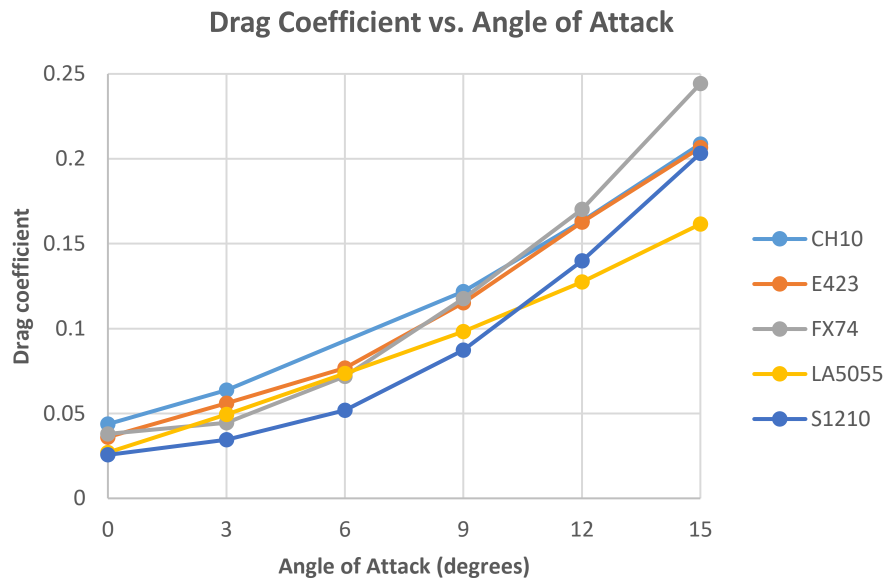

Figure 6 Numerical values for drag coefficient at various angles of attack.

There is a much larger spread of lift coefficients than there is for drag coefficients. The CH10, E423, FX74, and S1210 airfoils all have relatively high negative lift coefficients between 3º and 9º. It is difficult to directly compare the drag to the negative lift coefficients so therefore Figure 7 introduces a plot of the airfoil efficiencies to determine which airfoils are most aerodynamically efficient (C_L/C_D). Drag is not a large factor at average Formula SAE speeds (14.5735 m/s) so therefore negative lift generation has a much larger weight in airfoil selection, though, it is still useful to observe overall airfoil efficiency in this selection.

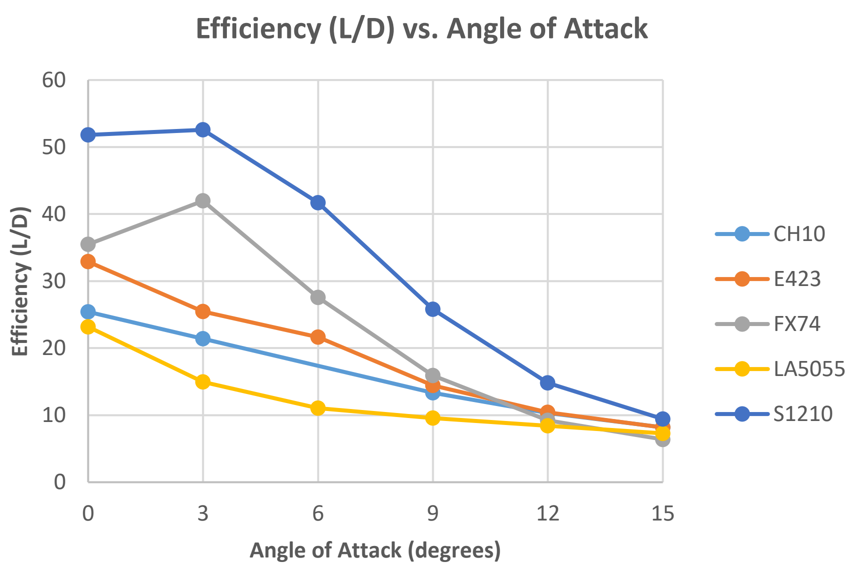

Figure 7 Numerical values for efficiency at various angles of attack.

When efficiency is considered, amongst lift and drag generation, it is evident that the S1210 airfoil is superior for a main element of a Formula SAE car front wing. It generates the highest amount of negative lift as well as has the highest efficiency for nearly every angle of attack that was tested. Figure 8 shows a velocity magnitude contour for the S1210 airfoil at a = 6º and 15º to quantify air behavior between two distinct changes in angle of attack to help examine why the changes in both coefficients occur.

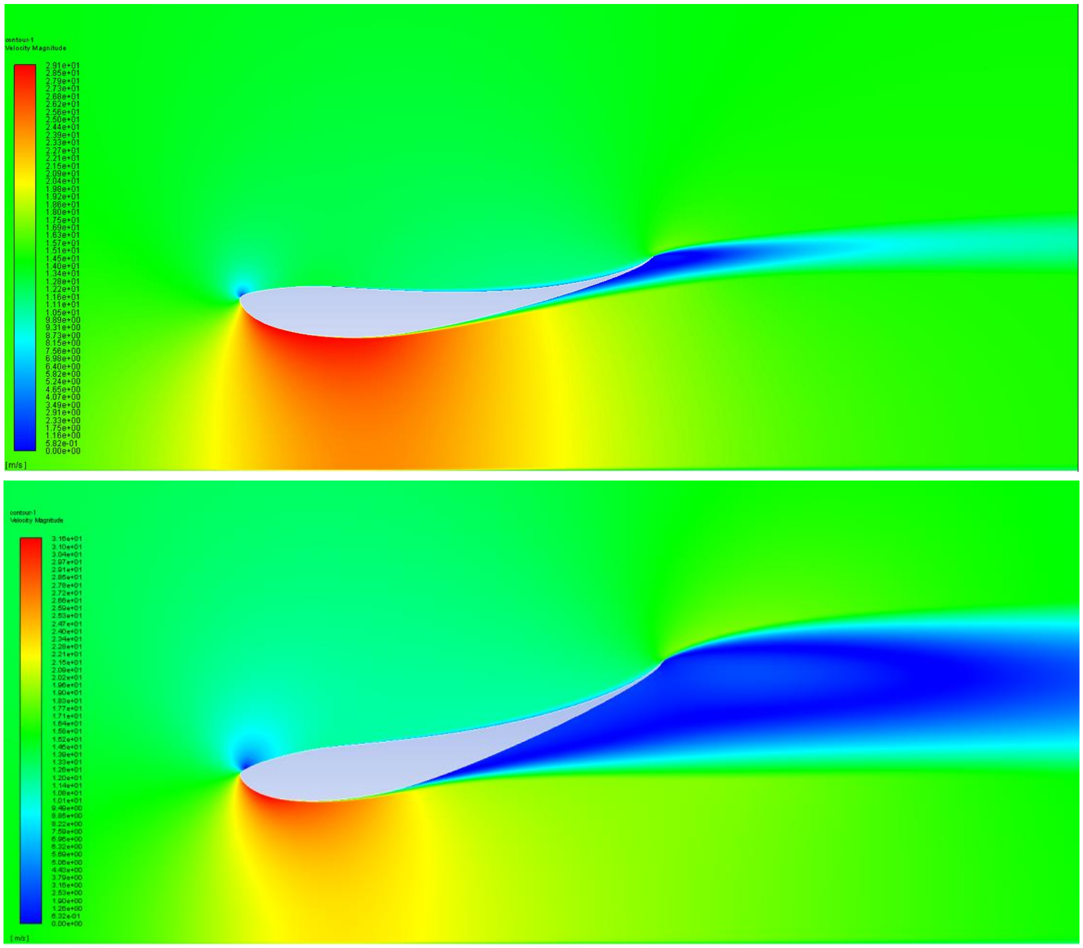

Figure 8   S1210, *a* = 6º (top) *a* = 15º (bottom), velocity contour plot.

The areas of high velocity in Fig. 8 are highlighted in colors ranging from yellow to red, free-stream air velocity is green, while lowest velocity regions are colored from light to dark blue. The negative lift generation peaks between *a* = 6º and 9º and then begins to drop. The larger area of dark blue in the bottom image shows the increased flow separation off of the airfoil at higher angles of attack. The air here is essentially at 0 m/s and therefore is stalling. Observing Bernoulli's principle, as velocity increases then pressure decreases (and vice versa) so the greater velocity region seen in the top image creates a larger, lower-pressure area on the bottom surface of the airfoil, contributing to an increased amount of lift generation. The flow separation impedes the airfoil's ability to create a low-pressure region so as we can see in the data in Figures 3 and 4, negative lift generation decreases steadily while the amount of resultant generated increases at a strong, positive exponential rate (R^2 =0.997).

After the S1210, the FX74 and E423 airfoils are the next two most efficient airfoils; these three airfoils are selected for the second height study. The height study will be very useful to ultimately examine how ground effect plays a role in Formula SAE racecar aerodynamics. To isolate these effects, a constant *a* = 6º is used and the airfoils are all run at the same speeds with the same chord lengths as the first study. Figures 9 and 10 respectively show the changes in negative lift coefficient and drag coefficient relative to changes in *h* (from the ground plane to the lowest point on each airfoil).

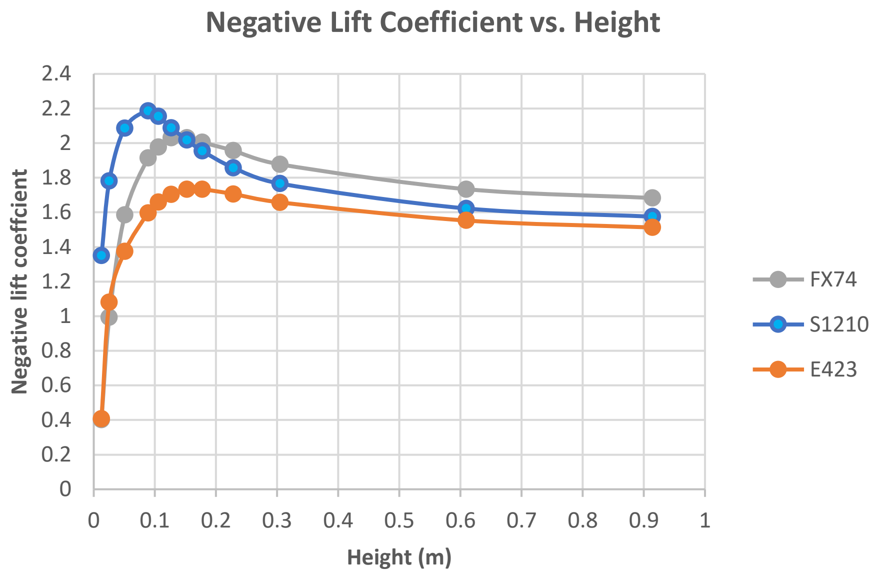

Figure 9 **Numerical values for negative lift coefficient at various heights off the ground.**

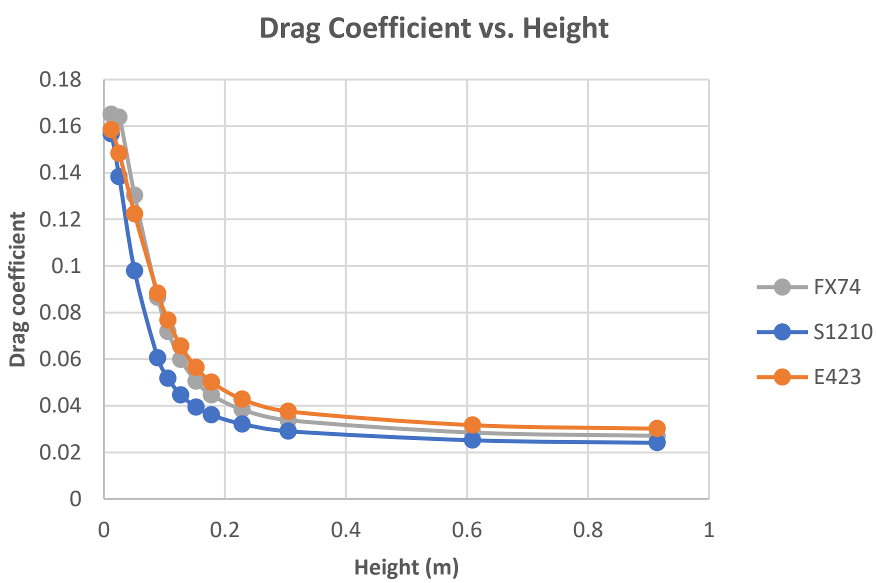

Figure 10 **Numerical values for drag coefficient at various heights off the ground.**

Observing Fig. 9, one can see that the 2D airfoils in the closest proximity to the ground do not generate the highest amounts of negative lift. The S1210 airfoil has the highest negative lift value between heights of 0.0127 m and 0.1524 m relative to the other airfoils studied, but the lift generated by the FX74 from 0.1542 m and to the maximum studied height of 0.9144 m is greater than that of the S1210 airfoil. This is a very useful insight since a rear wing must be created to balance the pressure and force effects of a front wing on a Formula SAE vehicle, therefore the FX74 will be the rational choice over the S1210 airfoil for the rear wing main element. The peak lift generation for the S1210 airfoil is at around 0.0889 m so ideally the bottom front wing element should be positioned at such height to have the largest negative lift generation. Fig. 10 shows the relationship between drag coefficient and height off the ground. The airfoils in the closest proximity to the ground have an exponentially higher ($R^2 = 0.942$) drag coefficient. An efficiency comparison can be observed in Figure 11.

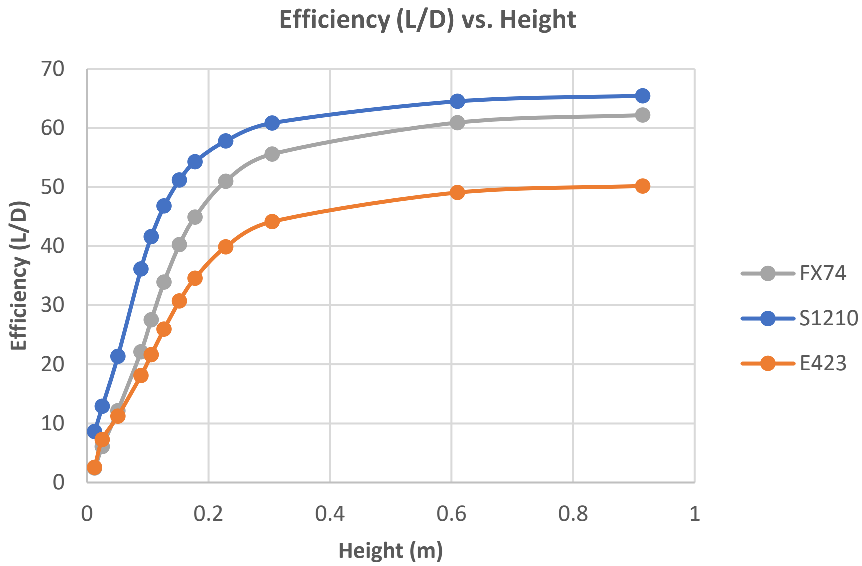

Figure 11 Numerical values for efficiency at various heights off the ground.

Yet again the S1210 airfoil has the highest value for efficiency as well as having the overall highest negative lift generation value for the height study. The S1210 will be the choice for the bottom element airfoil for the Formula SAE race car being designed by the Formula SAE Team at Washington University, Wash U Racing. The element ideally be positioned with an angle of attack between 6º and 9º and at a height off the ground between 0.0508 m and 0.0889 m, as found by this study.

Having selected the S1210 airfoil as the main element, a two-element configuration can begin to be iterated. The lower bounds of a = 6º and h = 0.0508 m values were selected for the main element positioning, with c = 0.33 m. The three highest downforce airfoils were selected again for the flap geometries. The flap is set with c = 0.1524 m and the angles of attack are iterated from 20º to 50º to study how the second element effects the main element in the same flow as utilized in the first two studies. Figures 12 through 14 show the negative lift coefficient, drag coefficient, and efficiency of the two-element configuration having flaps that are a S1210 and FX74 airfoil. Further runs with the E423 airfoil will be completed in the near future.

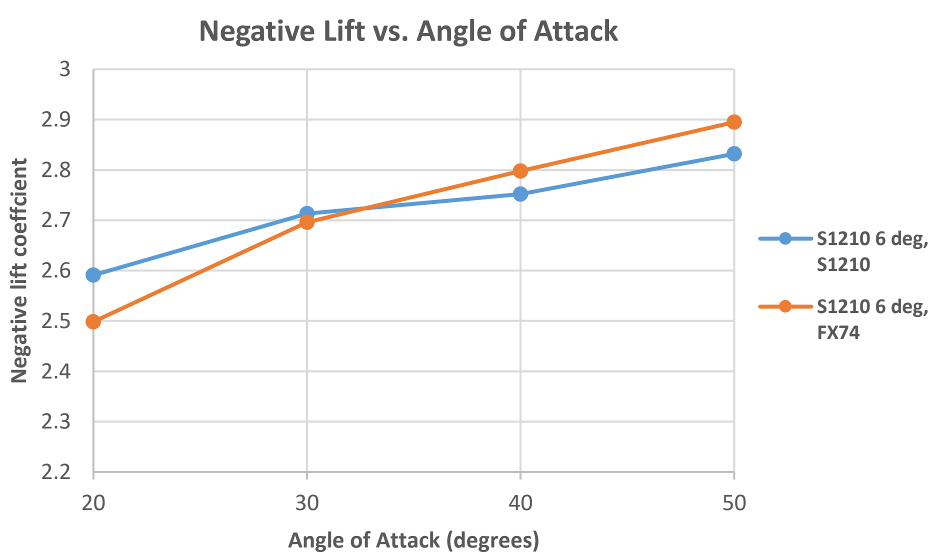

Figure 12 Numerical values for negative lift coefficient at various flap angles.

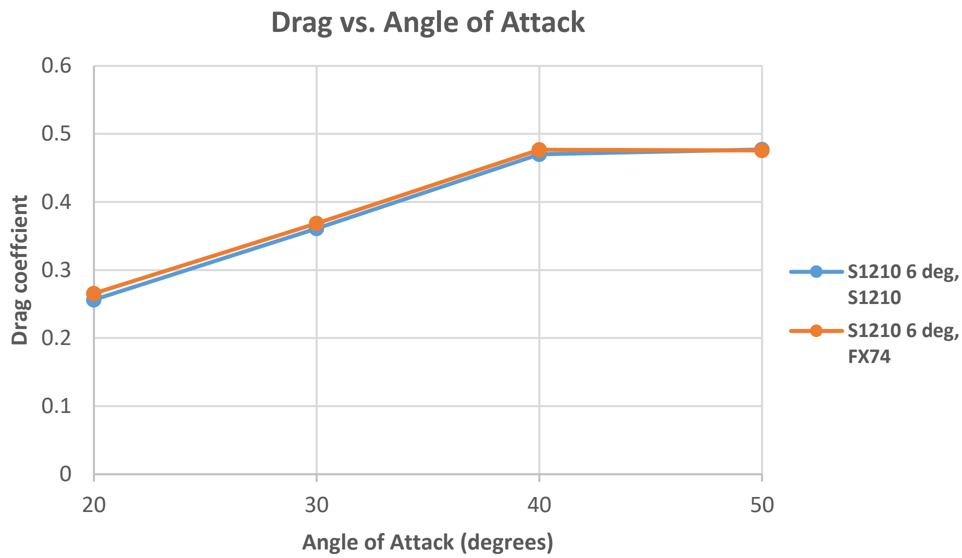

Figure 13 Numerical values for drag coefficient at various flap angles.

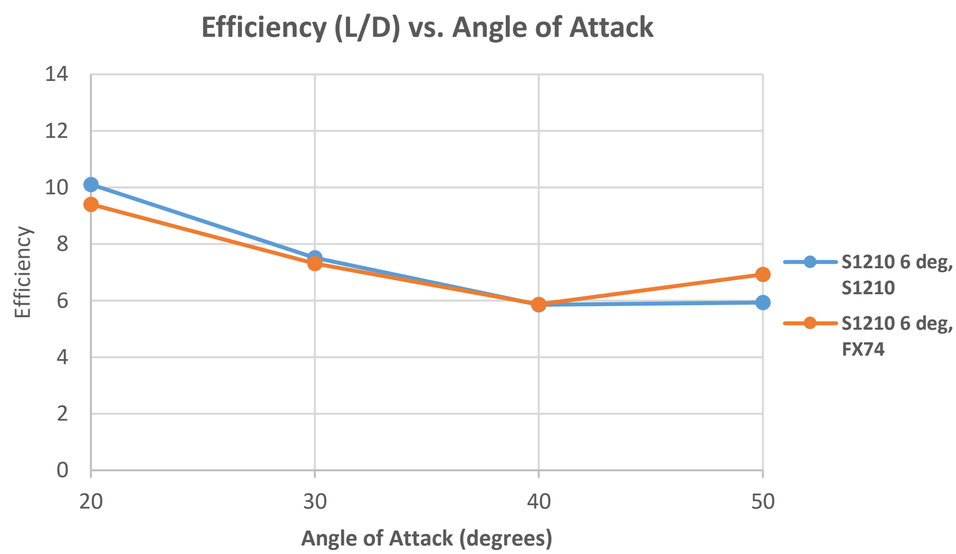

Figure 14 Numerical values for efficiency at various flap angles.

It is surprising to see that negative lift keeps increasing with increases in a from 20 to 50 degrees. Just to reiterate, 50 degrees is the upper limit for this specific configuration because it is at the maximum allotted height from the FSAE Rules. At lower angles of attack, the S1210 flap is superior to the FX74 flap but the values do cross between 30 and 40 degrees – since the angle is so extreme, the airfoil is further from the ground, and as shown in Fig. 9, the FX74 airfoil has a higher negative lift coefficient than the S1210 at higher h values. Figure 15 displays the velocity contour plot of the S1210, FX74 two-element airfoil at flap $a = 20^o$ and 50^o. It is worth noting that the flow separates from the flap airfoil between $a = 40^o$ and 50^o but it is interesting to see that drag stays almost constant between the two flap angles. The final combination of S1210 and E423 will be run subject to the same conditions and will be compared to the two shown airfoil configurations.

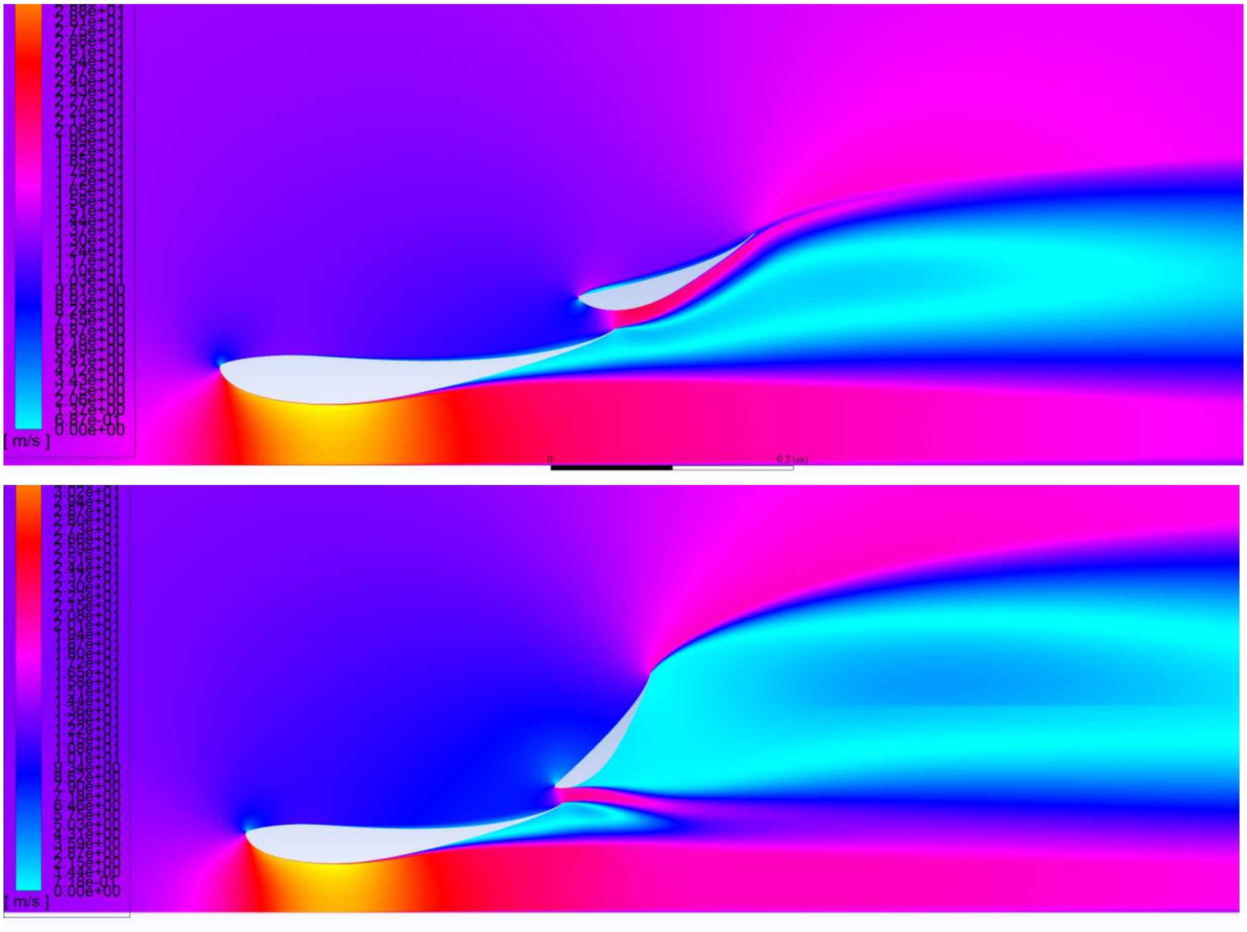

Figure 15 S1210, *a* = 6º, FX74, *a* = 20º (top) FX74, *a* = 50º (bottom), velocity contour plot.

VII. Conclusion, Future Work

An issue in front wing design for a Formula SAE racecar is that there is not a large amount of published airfoil data that has been run at a low speed and very close ground proximity. Formula 1 cars travel at much faster speeds and overall are scaled much larger than a FSAE car so therefore the same designs used for those cars may not be the most optimal configurations for a FSAE car. This study creates a fundamental understanding on how a variety of high lift airfoils behave at various angles of attack in close proximity to the ground as well as how a single angle of attack behaves at various heights off the ground. The choice of the S1210 airfoil is the first milestone in a front/rear wing package design.

Future work will further evaluate the second element geometry for the front wing. An angle of attack study for the E423 airfoil as a flap will be completed. Furthermore, main element and flap *h* can be varied to potentially increase negative lift. After the front wing design is finalized, a rear wing will be modelled with at minimum two elements, to balance the car's center of pressure between the front and rear of the chassis. There is room for automated optimization using tools such as the genetic algorithm to study a large population of configurations to assist in finding a global maximum negative lift value.

References

[1] SAE International. "2017-18 Formula SAE® Rules." FSAE Online, https://www.fsaeonline.com/content/2017-18-FSAE-Rules-091317.pdf.

[2] UIUC Applied Aerodynamics Group. "UIUC Airfoil Coordinates Database." UIUC Airfoil Data Site, m-selig.ae.illinois.edu/ads/coord_database.html.

[3] Mcbeath, Simon. Competition Car Aerodynamics. Veloce Publishing Ltd, 2017.