# Effects of Gurney Flap Setups around Circuit of the Americas

Sreeramana Vellanki[1] and Anirudh Sriram[2]

*University of Central Florida, Orlando, Florida, 32816, United States*

Michael P. Kinzel[3]

*University of Central Florida, Orlando, Florida, 32816, United States*

**The current study explores the aerodynamic impact of varying Gurney flap dimensions on the performance of a two-element Formula 1 front wing. The downforce-to-drag ratio is a critical metric for evaluating the efficiency of aerodynamic components in motorsports. Through an extensive computational approach, a simplified dual-element front wing was used to understand the effects Gurney flaps have on the downforce, drag coefficients and their ratio. Using this understanding, computational simulations were conducted replicating a lap around the Circuit of the Americas to find an optimized Gurney flap setup. The study employed the NACA 6412 and the GOE 265 airfoils as the main and second elements for the wing setup. The simulations were conducted at various Reynolds numbers, corresponding to the wide range of velocities experienced by a race car around a typical racing circuit. The expected results of this study are predicted to align with a paper studying the effects of Gurney Flaps on Formula 1 front wings by Mattia Basso (et al) [1]. In this project the presence of Gurney Flaps showed an increase in the downforce coefficient and decrease in drag coefficient. Using these results, a further expansion into this preliminary study will be conducted. The goal of this study will be investigating the optimal configurations for achieving an enhanced downforce-to-drag ratio by varying both the Gurney Flap height and width. The insights gained from this research contribute to the ongoing pursuit of aerodynamic efficiency and performance improvement in race car design.**

## I. Nomenclature

| | | |
|---|---|---|
| *AoA* | = | Angle of Attack |
| $c$ | = | chord length |
| $C_l$ | = | Coefficient of Lift/Downforce |
| $C_d$ | = | Coefficient of Drag |
| $C_l/C_d$ | = | Lift to Drag Ratio |
| $h$ | = | Height |
| $w$ | = | Width |
| *GF* | = | Gurney flap |

[1] Graduate Student, College of Engineering and Computer Science, Student Member 1595751.
[2] Undergraduate Student, College of Engineering and Computer Science, Student Member 1603452.
[3] Professor (Ph.D.) & Advisor, AIAA Faculty (504433), College of Engineering and Computer Science, University of Central Florida.

## II. Introduction

Motorsports, particularly open wheel formula racing, stands at the forefront of technological innovation, where even the smallest aerodynamic adjustments can yield significant performance gains. The front wing plays a pivotal role in dictating the balance between negative lift (downforce) generation and drag reduction – a delicate balance that directly impacts a car's lap time and competitiveness on the track. This is especially important at technically challenging tracks such as the Circuit of the Americas (COTA) in Austin, Texas. COTA has unique characteristics, as while the first and third sectors of the track are laden with many medium and high-speed turns, the middle sector features a long straight, requiring the race engineers to compromise between downforce and drag. A Gurney flap (GF) is a small component which mounts perpendicular to the airfoil, along the trailing edge on the pressure side. As illustrated in Fig. 1 a region of increased pressure is accompanied by two counter rotating vortices which form on either side of the flap. The effect is an increase in lift coefficient upwards of 30% [2].

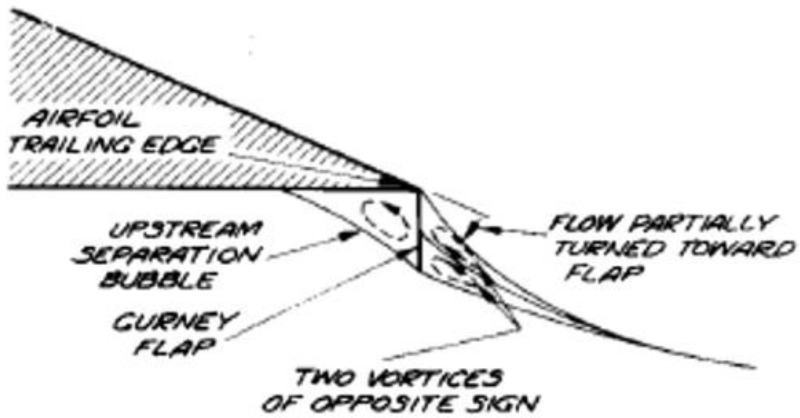


**Fig 1. Liebeck's hypothesized Gurney flap trailing edge.**

The dimensions of Gurney flaps are a critical parameter in their design. The dimensions of Gurney flaps are typically measured as a percentage of their chord length (c). Their height typically ranges from 1% to 5% c and their width typically ranges from 0.25% to 0.75% c of the aerodynamic surface to which they are attached [3]. This height limitation is primarily dictated by the aerodynamic requirements of the racing car and the peak velocity it achieves during competition. As the car accelerates to higher speeds, the boundary layer of the airfoil decreases in thickness, impacting the effectiveness of aerodynamic devices like Gurney flaps. The interaction between Gurney flaps and the boundary layer in motorsports is a key focus of this research along with the impact it has on coefficient of lift ($C_l$ ) (coefficient of downforce), and the coefficient of lift over coefficient of drag ratio ($C_l/C_d$). However, the design of Gurney flaps must consider the trade-offs between downforce and drag. Extending the Gurney flap beyond the boundary layer can increase downforce at lower velocities, it can also result in increased drag at higher velocities [4]. Therefore, finding the optimal balance between maximizing downforce at lower velocities and minimizing drag at higher velocities is crucial in Gurney flap design for racing cars.

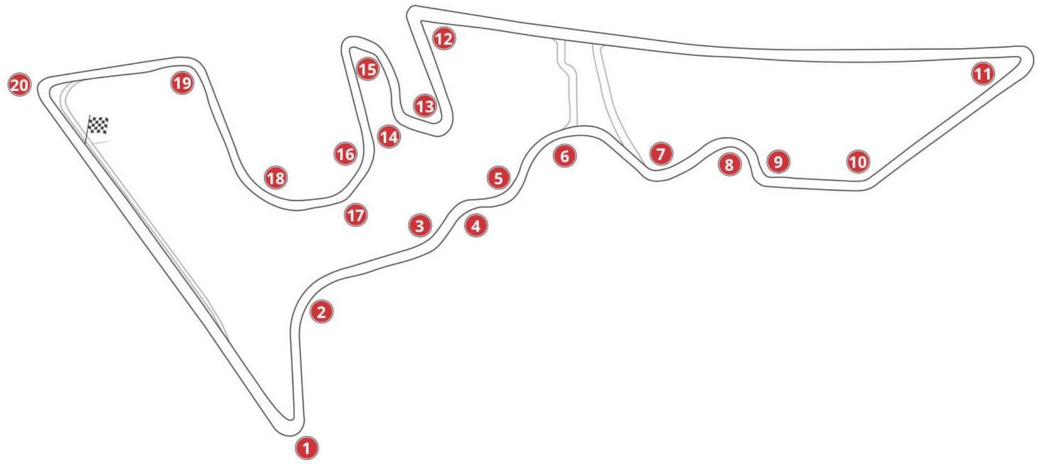


**Fig 2. Track Map of COTA.**

The expected results of this study are predicted to show similar characteristics with a paper studying the effects of Gurney flaps on Formula 1 front wings by Mattia Basso et al. [1]. In this project the presence of Gurney flaps showed an increase in the downforce coefficient and a minimal increase in drag coefficient. While this paper was focused on GFs in a motorsport application, it primarily analyzed the effect of GFs on the aerodynamic control surfaces aft of the front wing. A more in-depth analysis of Gurney flap heights was conducted by Giguere et al. [4], in which optimal dimensions for maximizing $C_l / C_d$ was discussed. It was concluded that to maximize the efficiency, the Gurney Flap should not go past the boundary layer. Kinzel et al. [5] analyzed the benefits of deployable miniature trailing-edge effectors (deployable Gurney flaps), primarily in rotorcraft applications. However, an in-depth analysis of the optimal height of GFs in ground effect wings for motorsports is yet to be done. The goal of this study will be investigating the optimal configurations for achieving an enhanced $C_l$ value with minimal decrease in the $C_l / C_d$ ratio in a motorsport application. The insights gleaned from this research aims to contribute to the ongoing pursuit of aerodynamic efficiency and performance enhancement in race car design.

## III. Methodology

In this study, computational fluid dynamics (CFD) simulations were conducted using Star-CCM+ to analyze the flow characteristics around a simplified dual-element front wing setup (Fig. 3). The primary objective was to investigate the aerodynamic performance of the configuration and assess its suitability for achieving optimal downforce and drag characteristics. Through these simulations, the downforce coefficient and drag coefficient were determined for each airfoil of the front wing setup, as well as their combined total. These coefficients served as key metrics for quantifying the aerodynamic performance and efficiency of each configuration. Subsequently, these metrics were utilized to evaluate the optimality of the setups.

**Fig. 3 Set of 2D wing model**

The simulations were conducted utilizing the Reynolds Averaged Navier Stokes (RANS) equations (Eqs. (1)-(3)) which come from the continuity, momentum, and energy equations [6]. This was coupled with the k-ω SST turbulence model [7]. The RANS equations captured the mean flow behavior, while the k-ω SST turbulence model accurately captured turbulent flow phenomena [6,7]. These models were selected based on the knowledge that the range of Reynolds numbers for the simulations were between 630,000 and 2.4 million.

$$\frac{\partial \rho}{\partial t} + \nabla\cdot(\rho\overline{\mathbf{v}}) = 0 \tag{1}$$

$$\frac{\partial}{\partial t}\left(\rho\overline{\mathbf{v}}\right) + \nabla\cdot(\rho\overline{\mathbf{v}}\otimes\overline{\mathbf{v}}) = -\nabla\cdot\bar{p}_{\text{mod}}\mathbf{I} + \nabla\cdot\left(\overline{\mathbf{T}} + \mathbf{T}_{RANS}\right) + \mathbf{f}_b \tag{2}$$

$$\frac{\partial}{\partial t}\left(\rho\overline{E}\right) + \nabla\cdot\left(\rho\overline{E}\overline{\mathbf{v}}\right) = -\nabla\cdot\bar{p}_{\text{mod}}\overline{\mathbf{v}} + \nabla\cdot\left(\overline{\mathbf{T}} + \mathbf{T}_{RANS}\right)\overline{\mathbf{v}} - \nabla\cdot\overline{\mathbf{q}} + \mathbf{f}_b\overline{\mathbf{v}} \tag{3}$$

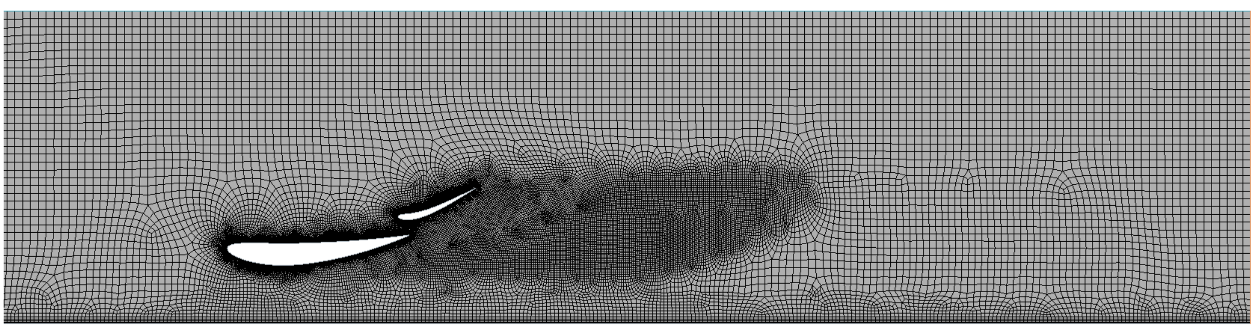

**Fig. 4. Meshed model of 2D wing setup with 0.01 base size.**

A 2$^{nd}$ order, implicit unsteady computational scheme was implemented to increase temporal discretization. To select a base mesh size for this scheme, a grid convergence study was conducted. For this study, three base mesh sizes were tested. These test mesh sizes consisted of a coarse mesh (0.1m), a baseline mesh (0.01m) and a fine mesh (0.001m). The coefficient of lift ($C_l$) and drag ($C_d$) were used as the comparison metric. It was observed that the coarse mesh resulted in a 0.55% increase in $C_l$ and 15.35% increase in the $C_d$ from the baseline mesh size. The fine mesh showed a 0.01% decrease in $C_l$ and a 0.67% decrease in $C_d$ compared the baseline mesh. Based on these results, a base mesh size of 0.01m (Fig. 4) was utilized as the results varied slightly from the fine mesh. Subsequently, using 0.01m as the base mesh size allowed for reduced computational time.

**Table 1. Mesh Study**

| Base Mesh Size (m) | $C_l$ | Percent Difference | $C_d$ | Percent Difference |
|---|---|---|---|---|
| 0.1 | 2.585540 | 0.55% | 0.080389 | 15.35% |
| 0.01 | 2.571337 | - | 0.069690 | - |
| 0.001 | 2.571005 | 0.01% | 0.070159 | 0.67% |

The computational fluid domain (Fig. 5) was constructed as a 2D representation. The simulation was restricted to 2D modeling on account of the primary aerodynamic forces, including downforce and drag, which predominantly act in vertical and longitudinal directions. This made analysis along the lateral (3D) axis unnecessary for the objectives of this study. Focusing on the two-dimensional representation allowed for the effective modeling and analysis of airflow interactions with the front wing setup in a computationally efficient manner.

The 2D domain measures 2m x 0.5m. The dual element front wing was positioned at 0.375m from the left boundary and 0.09m above the bottom boundary. The reason for setting the height at 0.09m will be discussed below. The boundaries were set as a velocity inlet, pressure outlet (ambient pressure), wall, and symmetry plane for the left, right, bottom, and top boundaries respectively. These were set to simulate how a front wing would be set on a track.

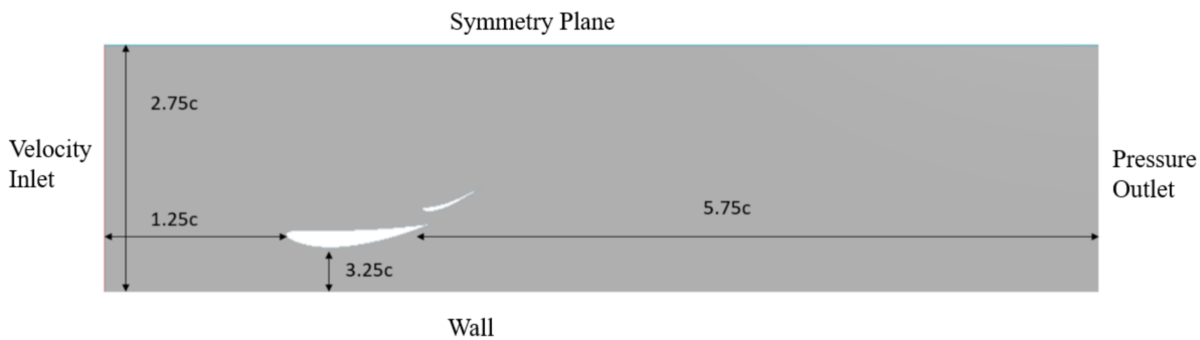


**Fig. 5 Computational Fluid Domain.**

The airfoils used for the dual element front wing were the NACA 6412 (main) and the GOE 265 (flap) airfoils. The NACA 6412 airfoil was selected as it displayed the highest Cl and Cl/Cd in a study comparing three airfoils in ground effect [8]. The flap airfoils used in the studies conducted by Basso et al. [1] and David et al. [9] had an aft section of approximately 1mm thick. These airfoils were not standardized airfoils and were purpose created. As such, the flap airfoil employed in this study was a GOE 265 airfoil which was similar in physical characteristics with a similar camber and an aft thickness of 2mm. The angles of attack were held constant at 4° and 20° respectively [1,9].

The leading edge of the flap was positioned 0.26m behind the leading edge of the main airfoil with a slot gap separation of 0.02m (Fig. 6). These distances between the airfoils were used based on the setup used in the research done by David (et. al.) [9].

In selecting ground clearance height (height from the base of the airfoil to the ground) the approach used by David et al. [9] and Vogt et al. [10] were used as well. In those studies, it was observed that as the ground clearance height was reduced, the downforce generated increased, up to a threshold height. At this threshold height, maximum downforce was produced by ground effect. Any further reduction in height saw a significant reduction in downforce generated. Using the same methodology, a ground clearance height of 90mm was observed to generate the most downforce from ground effect for this study.

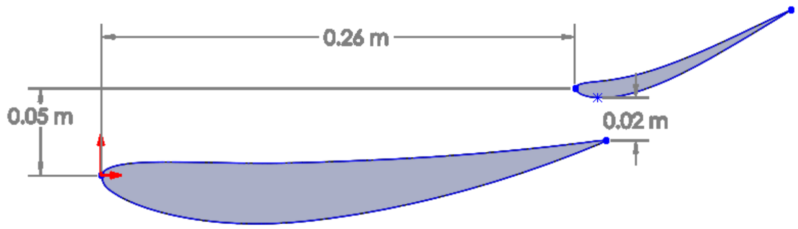


**Fig. 6. Dual Element Front Wing Orientation**

In order to analyze and identify the Gurney flap dimensions which yielded the optimal performance, multiple configurations were made. These configurations were defined by the various speeds experienced around the Circuit of the Americas track (Fig. 2), along with the height and the width of the Gurney flaps. Table 2 shows the list of speeds used to set up the configurations. Certain turns were grouped together and averaged, as they had similar speeds, in order to reduce the number of configurations that needed to be simulated. In the study done by David et al. [9], Gurney flap heights were tested at 1.5 and 3%c. Taking these values as a base, a larger range of 1-6%c was selected to analyze a wider set of configurations and their effect on the performance around a lap. Similarly, the range of Gurney flaps widths selected were 0.35-0.75%c. This range was based on a study conducted by Yang et al. [3].
The combination of the run parameters (disregarding duplicate speeds of 23.61m/s and 73.61m/s) resulted in 270 individual runs for a total of 18 test cases. An additional test case without Gurney flaps was created as well to serve as a benchmark for the aerodynamic performance of the 18 test cases.

**Table 2 Speeds per Turn at COTA**

| Turn | Speed (m/s) |
|---|---|
| 1 | 23.61 |
| 2 | 63.89 |
| 3-4 | 80.56 |
| 5-6 | 73.61 |
| 7 | 58.33 |
| 8 | 34.72 |
| 9 | 50.00 |
| 10 | 73.61 |
| 11 | 25.00 |
| Backstraight | 89.72 |
| 12 | 27.78 |
| 13-14 | 41.11 |
| 15 | 23.61 |
| 16-18 | 75.00 |
| 19 | 56.94 |
| 20 | 31.94 |
| Pit-straight | 76.04 |

To quantify the aerodynamic performance of the test cases, the coefficient of downforce, coefficient of drag, and the downforce-to-drag ratio were calculated for each wing element in Star-CCM+.

## IV. Results

Upon completion of the runs, the downforce-to-drag ratios for each test case were plotted against the corresponding velocity. The plots showed that the test cases that yielded the highest downforce-to-drag ratio across all velocities were the no Gurney flap case, the 0.55%c x 1%c Gurney flap case, and the 0.75%c x 1%c Gurney flap case. Figure 7a plots the $C_l/C_d$ vs velocity for those three cases. In this plot, it is seen the run with the highest downforce-to-drag ratio across the full spectrum of velocities is the no Gurney flap test case. On plotting the $C_l$ and $C_d$ of those three cases against the velocities (Fig. 7b and 7c), the 0.55%c x 1%c Gurney flap shows the highest $C_l$. The No GF run shows the lowest $C_d$. Further expanding on this data, the $C_l$, $C_d$, and the $C_l/C_d$ values were tabulated at four key areas on the track (Table 3). These areas are Turn 1, Turns 13-14, Turns 16-18, and the backstraight. These areas were specifically selected as Turn 1 records the lowest levels of downforce, due to being the slowest corner on the track. Turns 13-14 are technical, medium speed corners requiring greater aerodynamic grip. Turns 16-18 are the fastest corners of the track, where drivers push flat out for the entirety of the right-hand turn complex. This requires downforce generated to counter understeer. The last section chosen is the back straight, where although the downforce generated may be far less important, the highest drag is experienced, thus this is where the efficiency of the setup is most important.

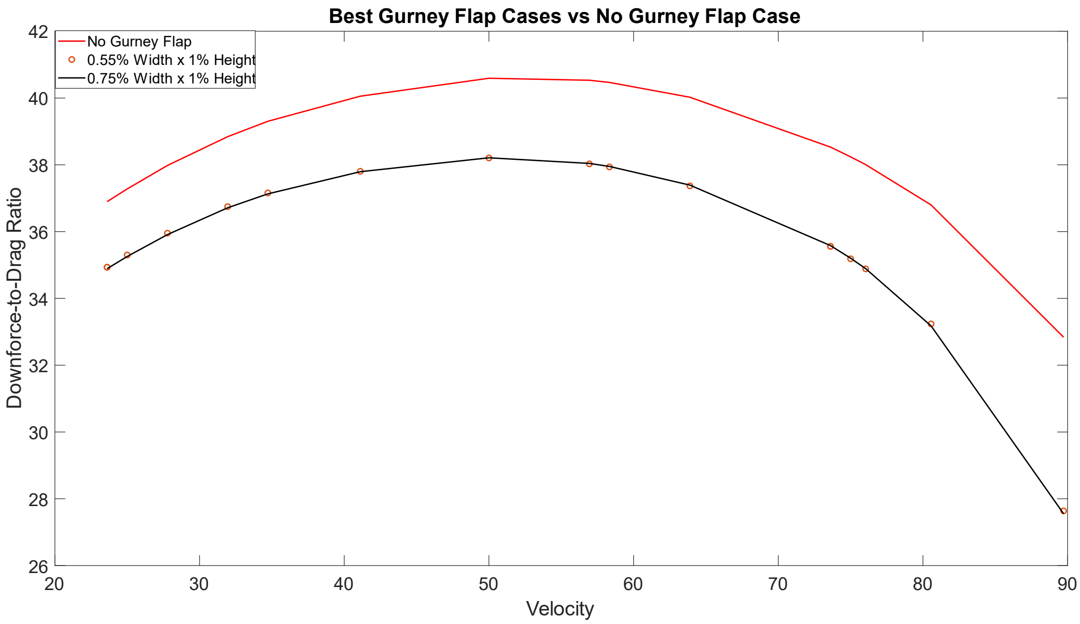


**Fig. 7a $C_l/C_d$ vs Velocity for No GF, 0.55%c x 1%c, and 0.75%c x 1%c**

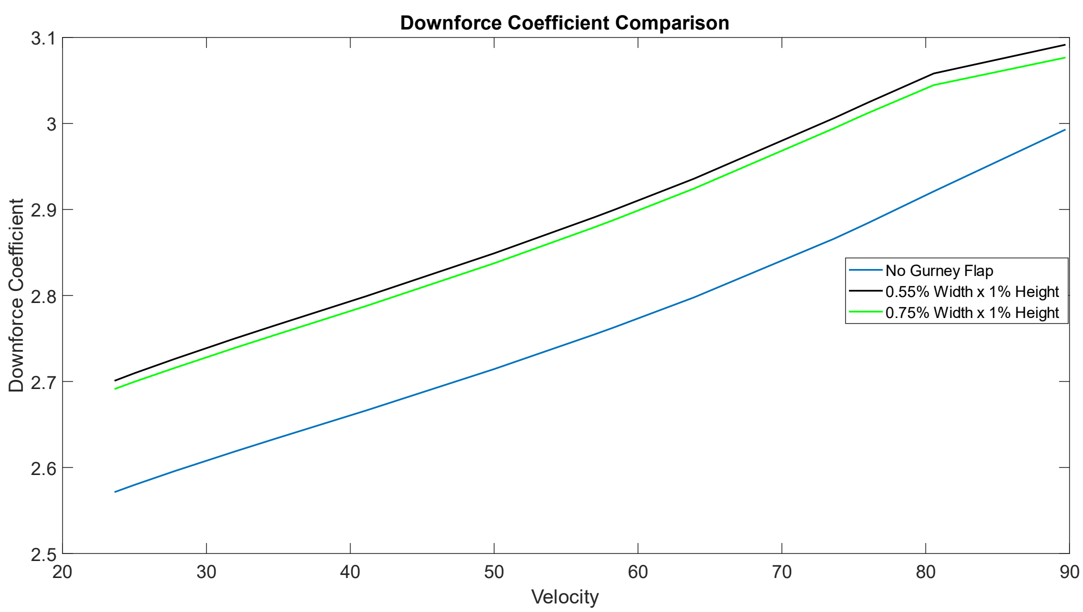


**Fig. 7b Downforce Coefficient vs. Velocity for No GF, 0.55%c x 1%c, and 0.75%c x 1%c**

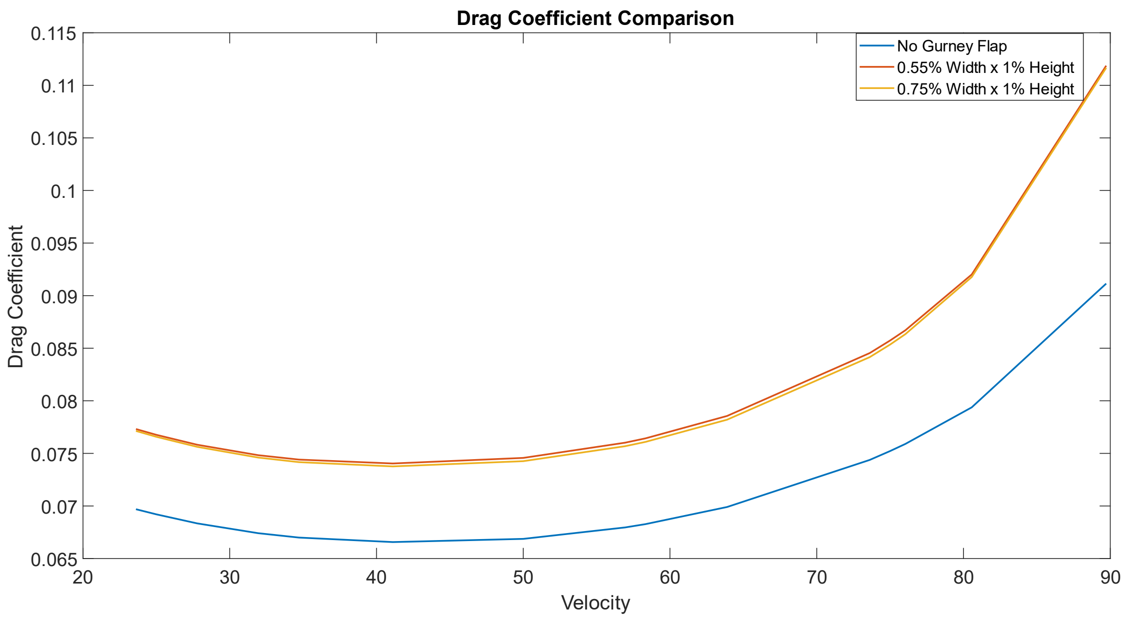


**Fig. 7c Drag Coefficient vs. Velocity for No GF, 0.55%c x 1%c, and 0.75%c x 1%c**

**Table 3 No GF, 1% c, and 2% c GF comparison**

| | No Gurney flap | | | Height: 1% c, Width: 0.55% c | | | Height: 1% c, Width: 0.75% c | | |
|---|---|---|---|---|---|---|---|---|---|
| | $C_l$ | $C_d$ | $C_l$ / $C_d$ | $C_l$ | $C_d$ | $C_l$ / $C_d$ | $C_l$ | $C_d$ | $C_l$ / $C_d$ |
| Turn 1 | 2.57 | 0.07 | 36.9 | 2.70 | 0.08 | 34.9 | 2.80 | 0.08 | 33.3 |
| Turns 13-14 | 2.99 | 0.09 | 32.8 | 3.09 | 0.11 | 27.6 | 2.90 | 0.08 | 36.0 |
| Turns 16-18 | 2.67 | 0.07 | 40.1 | 2.80 | 0.07 | 37.8 | 3.12 | 0.10 | 32.3 |
| Backstraight | 2.88 | 0.08 | 38.2 | 3.02 | 0.09 | 35.2 | 3.17 | 0.13 | 24.3 |

Based on the values shown in the table, the $C_l$ and $C_d$ values for both the 0.55%c and the 0.75%c Gurney Flaps show slightly larger values when compared to the no Gurney Flap run. As the difference between the 0.55%c and 0.75%c runs are very similar in values, another approach needs to be used to quantify the aerodynamic efficiency between these two runs. As such, the power loss as a result of the coefficient of drag was calculated and plotted against the various corners of the track. In order to conduct this study, the distances between the selected turns in Table 2 were measured. Upon collecting these measurements, the approximate time spent in the list of turns were calculated with the purpose of creating a plot that emulates the changes experienced by the front wing over a lap. Figure 8 shows the coefficient of drag plotted against the time around the track, along with the power loss per unit length ($\dot{E}/_l$) . This was calculated by multiplying the initial turn's velocity and its coefficient of drag. The subsequent turns also carried out the same operation with the addition of adding the previous turn's value. This resulted in the power loss per unit length to increase as time progressed. For the 0.55%c Gurney flap, the power loss per unit length totals to 76.15W/m.

The same methodology was conducted on the 0.75%c Gurney flap test case. The resulting data was then plotted, showing the power loss per unit length due to drag for each test case over a lap. Figure 9 shows the resulting plot. Based on the plot and the calculated data, the 0.55% Gurney flap set up was slightly more inefficient by 0.36% when compared to the 0.75%c Gurney flap. Based on this, it can be said that out of all of the Gurney flap configurations, the 0.75%c width x 1%c height Gurney Flap setup would be the optimal setup around the track.

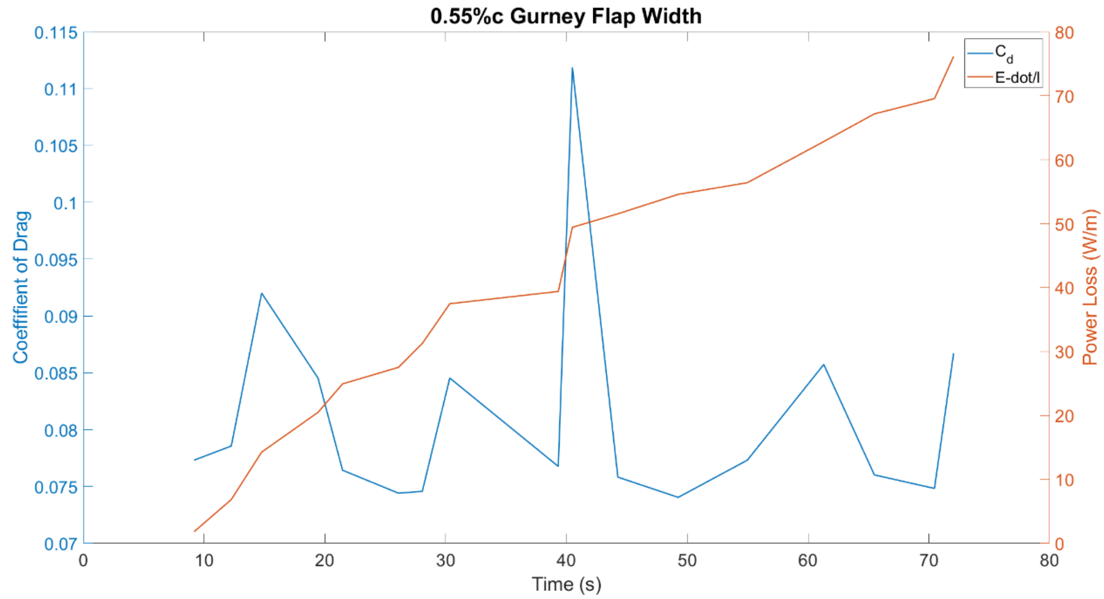


**Fig. 8 0.55%c GF Cd vs Time and Power Loss per Unit Length vs Time**

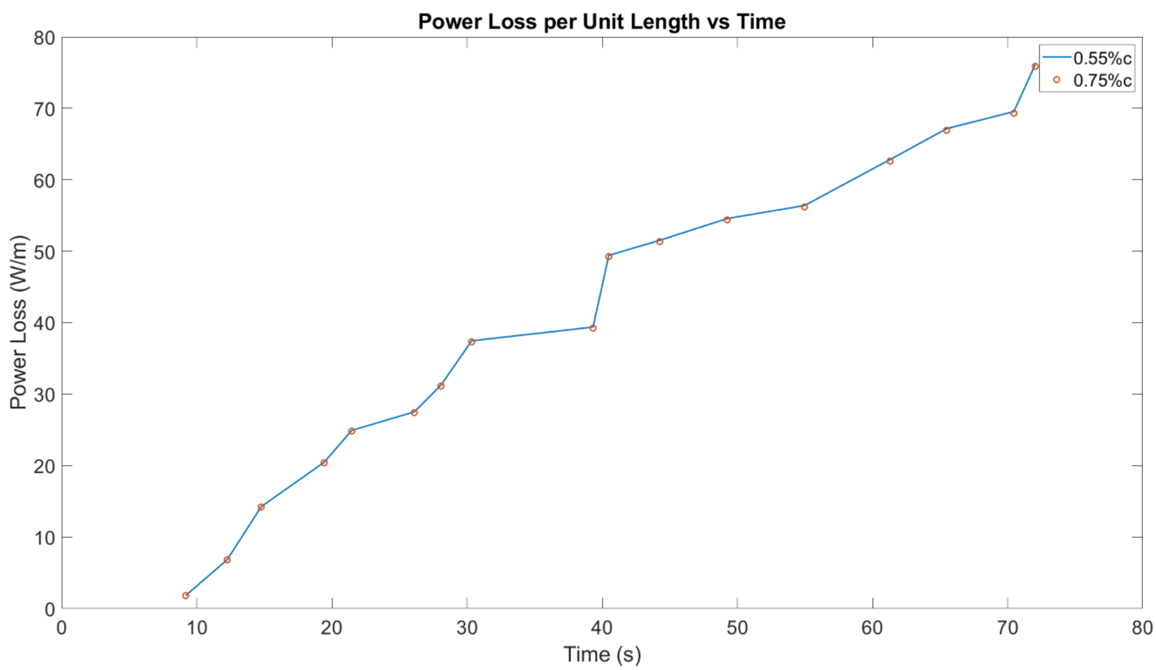


**Fig. 9 Power Loss per Unit Length vs. Time for all three cases**

## V. Conclusion

The results of this CFD analysis on a front wing configuration have been used to identify the effects of Gurney flap dimensions on the overall downforce and drag produced on a simplified dual-element front wing. It has been observed that an addition of even a small aerodynamic component can have a major influence on the fluid flowing around the airfoil. Several setups have been identified, tailored to the goal of the race car. A Gurney flap of height 1% c and width 0.75% c has been identified as the most optimal set up for efficiency purposes.

## References


[1] Basso, M., Cravero, C., and Marsano, D., “Aerodynamic Effect of the Gurney Flap on the Front Wing of a F1 Car and Flow Interactions with Car Components,” Energies, Vol. 14, No. 8, 2021. https://doi.org/10.3390/en14082059

[2] Cole, J. A., Vieira, B. A. O., Coder, J. G., Premi, A., and Maughmer, M. D., “Experimental Investigation into the Effect of Gurney Flaps on Various Airfoils,” 2013.

[3] Yang, J., Yang, H., Zhu, W., Li, N., and Yuan, Y., “Experimental Study on Aerodynamic Characteristics of a Gurney Flap on a Wind Turbine Airfoil under High Turbulent Flow Condition,” Applied Sciences (Switzerland), Vol. 10, No. 20, 2020, pp. 1–21. https://doi.org/10.3390/app10207258

[4] Giguère, P., Dumas, G., and Lemay, J., “Gurney Flap Scaling for Optimum Lift-to-Drag Ratio,” AIAA Journal, Vol. 35, No. 12, 1997, pp. 1888–1890. https://doi.org/10.2514/2.49

[5] Kinzel, M. P., Maughmer, M. D., and Duque, E. P. N., “Numerical Investigation on the Aerodynamics of Oscillating Airfoils with Deployable Gurney Flaps,” AIAA Journal, Vol. 48, No. 7, 2010, pp. 1457–1469. https://doi.org/10.2514/1.J050070

[6] Alfonsi, G., “Reynolds-Averaged Navier-Stokes Equations for Turbulence Modeling,” Applied Mechanics Reviews. 4. Volume 62, 1–20. https://doi.org/10.1115/1.3124648

[7] Wilcox, D., “Formulation of the K-w Turbulence Model Revisited,” AIAA Journal, Vol. 46, No. 11, 2008.

[8] Zaheer, Z., Reby Roy, K. E., Nair, G. S., Ragipathi, V., and Niranjan, U. V., “CFD Analysis of the Performance of Different Airfoils in Ground Effect,” Vol. 1355, 2019. https://doi.org/10.1088/1742-6596/1355/1/012006

[9] David, J., and Zerihan, C., “An Investigation into the Aerodynamics of Wings in Ground Effect,” 2001.

[10] Vogt, J. W., and Barber, T. J., “Ground Effect Phenomena about Lift and Downforce Generating Cambered Aerofoils,” International Journal of Numerical Methods for Heat and Fluid Flow, Vol. 22, No. 2, 2012, pp. 153–174. https://doi.org/10.1108/09615531211199809