



# Case Study

## STEM Racing Car Wing Configuration Study in Ansys Discovery Software

Developed and curated by the Ansys Academic Development Team

Gautham Varma and János Plocher

[education@ansys.com](mailto:education@ansys.com)

# Ansys Software Used

This resource uses Ansys Discovery™ 3D product simulation software

# Summary

This case study investigates the aerodynamic performance of a STEM Racing car by analyzing three front and three rear wing configurations using Ansys Discovery software. The study explores how variations in wing positioning and design influence key aerodynamic forces, namely drag and downforce. By leveraging the rapid simulation capabilities of Ansys Discovery software, the analysis provides insights into the complex interactions between car geometry and airflow. The study demonstrates the software’s effectiveness as a tool for early-stage aerodynamic design, enabling efficient evaluation of multiple configurations and guiding subsequent, more detailed simulations for optimization of the car’s performance.

**NOTE:** The simulations in this case study are **not** run using realistic STEM racing speeds. Instead, 85 m/s, which is the common straight-line speed of a professional race car with a standard aerodynamic setup, was chosen to better illustrate the results of wing impacts on design. The CAD file used for these simulations is provided with this case study, if you wish to re-run the results at more reasonable speeds.

# Table of Contents

1. Introduction.....	3
2. Theory of Drag and Downforce .....	3
2.1 Drag Force .....	4
2.2 Downforce (Negative Lift).....	4
3. Simulation Setup .....	5
4. Results and Discussions .....	6
5. Conclusions .....	10

## 1. Introduction

A STEM Racing car's performance, like any race car, is fundamentally shaped by its ability to manage airflow, balancing the opposing forces of drag and downforce. Drag, the air resistance that slows the car, must be minimized to achieve top straight-line speed. Downforce, a vertical force that presses the car onto the track, is crucial for generating the grip necessary to handle high-speed corners<sup>1</sup>. The front and rear wings are the primary components used to manipulate these forces, and their design represents a continuous compromise between cornering performance and straight-line velocity.

Traditionally, aerodynamic design has relied on costly and time-consuming physical wind tunnel tests. However, with the rise of powerful computational tools, engineers can now conduct quick and detailed simulations. This study employs Ansys Discovery software to illustrate the aerodynamic effects of various front and rear wing designs. By simulating three different wing setups for both front and rear wing (together with the side pods these are the most optimized components to reduce drag), we aim to quantify their influence on both drag and downforce.

In this case study, we will explore how these aerodynamic principles can be better understood and visualized using Ansys Discovery software, providing an intuitive way to connect theory with practical simulation. This document also highlights how Ansys Discovery software can be a highly effective and efficient tool for rapid prototyping and optimization.

## 2. Theory of Drag and Downforce

All forces acting on the STEM racing car are visualized in Figure 1.

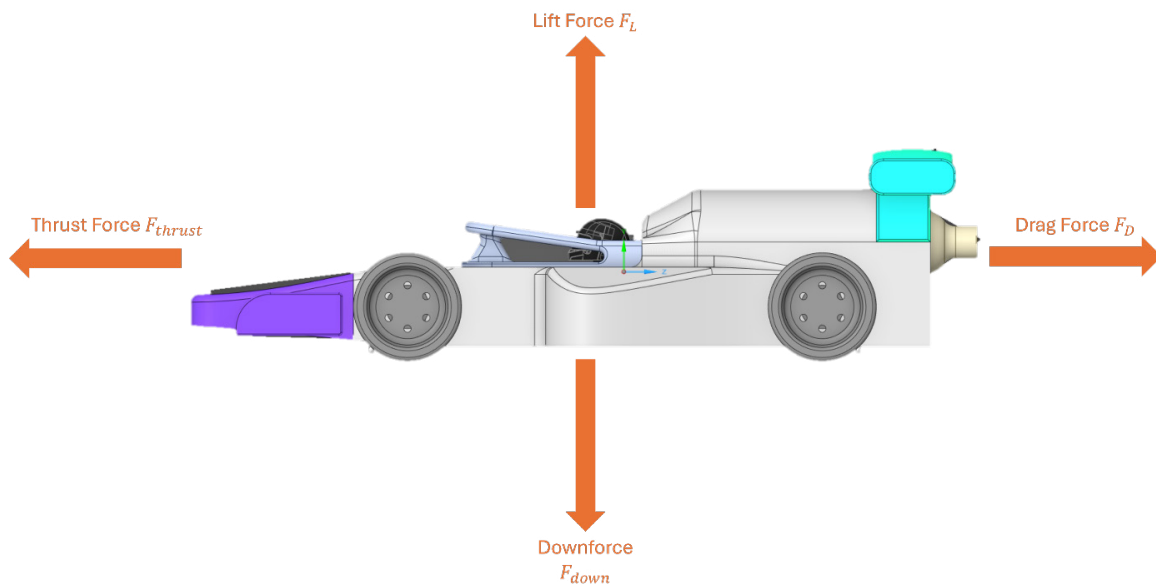


Figure 1 : A STEM Racing car diagram illustrating the different forces acting on the car.

---

<sup>1</sup> Downforce has minimal impact for STEM Racing, due to the track being a straight line. It has been included in this document to show a more complete picture of the aerodynamic forces at play, as well as the capabilities of the simulation tools shown.

The two main forces we will focus on in this case study are:

1. Drag ( $F_D$ ) – That pushes against the car slowing it down.
2. Downward Force (i.e. Negative Lift  $F_{\text{down}}$ ) – Pushes the car into the track to increase the grip and cornering ability.

## 2.1 Drag Force

The drag force on the model car can be expressed as:

$$D = \frac{1}{2} \rho V^2 C_D A$$

Where:

- $D$  = Drag force (N)
- $C_D$  = Drag coefficient.
- $A$  = Frontal area

## 2.2 Downforce (Negative Lift)

STEM Racing cars generate massive downforce using front and rear wings, underbody diffusers, and aerodynamic shaping. The downforce equation is:

$$F_{\text{down}} = \frac{1}{2} \rho V^2 C_L \cdot A$$

Where:

- $F_{\text{down}}$  = Downforce (N)
- $\rho$  = Air density ( $\sim 1.225 \text{ kg/m}^3$  at sea level)
- $V$  = Car speed (m/s)
- $C_L$  = Lift coefficient (negative for downforce)
- $A$  = Wing or reference area ( $\sim 1\text{--}1.5 \text{ m}^2$ )

### 3. Simulation Setup

The Ansys Discovery software simulation setup is employed to investigate the aerodynamic performance of a STEM Racing car (original geometry from [EduCAD Ltd](#) with slight modification made to the CAD), with a particular focus on evaluating the lift and drag forces arising from variations in wing and body configurations. Three front wing designs (hereafter referred to as FW1, FW2, and FW3) and three rear wing designs (hereafter referred to as RW1, RW2, and RW3) are examined, as illustrated in Figure 2.

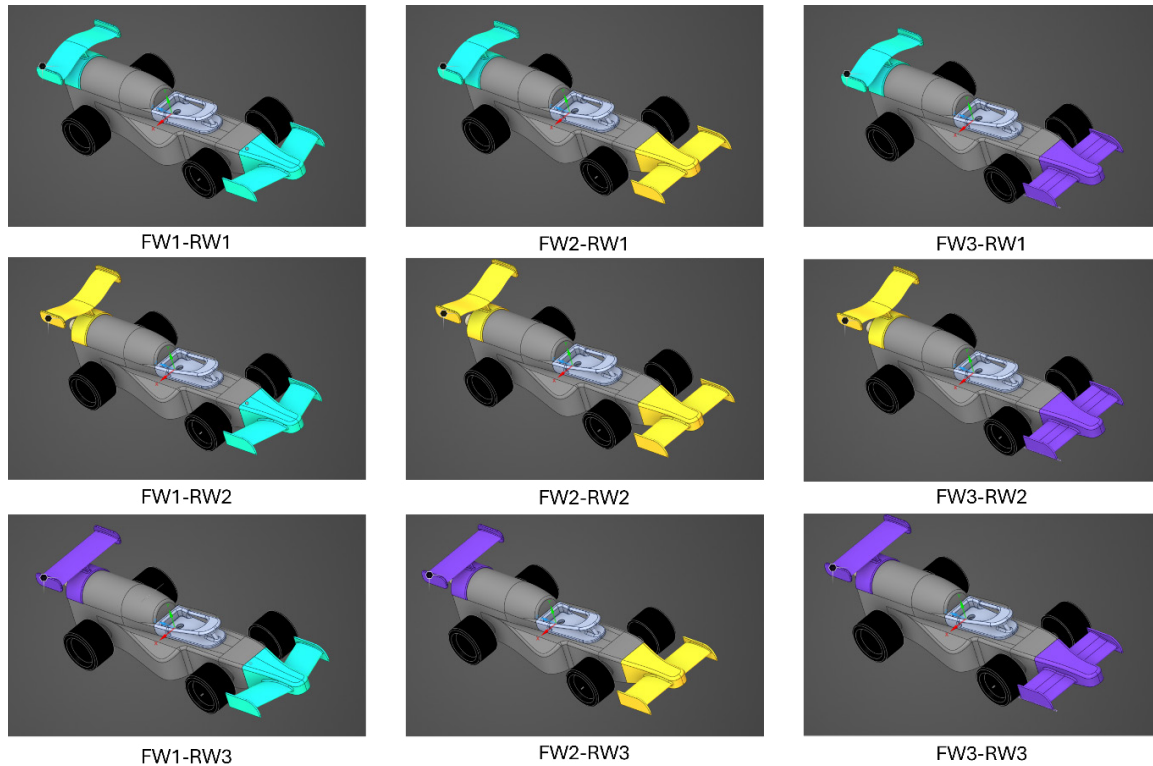


Figure 2: The different designs of Rear Wing and Front wing configurations are shown in the images that are used in the current simulations. FW1-3 denotes the front wing configurations and RW1-3 denotes the rear wing configurations.

In this case study Ansys Discovery software's Explore mode was utilized to exploit the fast GPU-based solver to quickly test the different wing configurations. Modifications to the surrounding enclosure were implemented for the various aerodynamic configurations using a Body of Influence (BOI) approach. A mesh refinement of 0.001 m was applied within the BOI region, while a finer resolution of 0.00025 m was used near the car surface to accurately capture the detailed flow behavior. This meshing strategy improves the reliability of the computed lift and drag coefficients, ensuring they closely represent the expected physical values. The resulting mesh configuration is shown in Figure 3 to accurately capture lift and drag forces. A body of influence (BOI) region and higher mesh fidelity near the car surface are employed to enhance the accuracy of aerodynamic force prediction.. The same meshing methodology was applied consistently across all car variants to maintain comparability of results.

The final mesh<sup>2</sup> contained approximately 2.79 million elements, providing a reasonable balance between computational cost and the accuracy required to evaluate the aerodynamic performance of the different configurations.

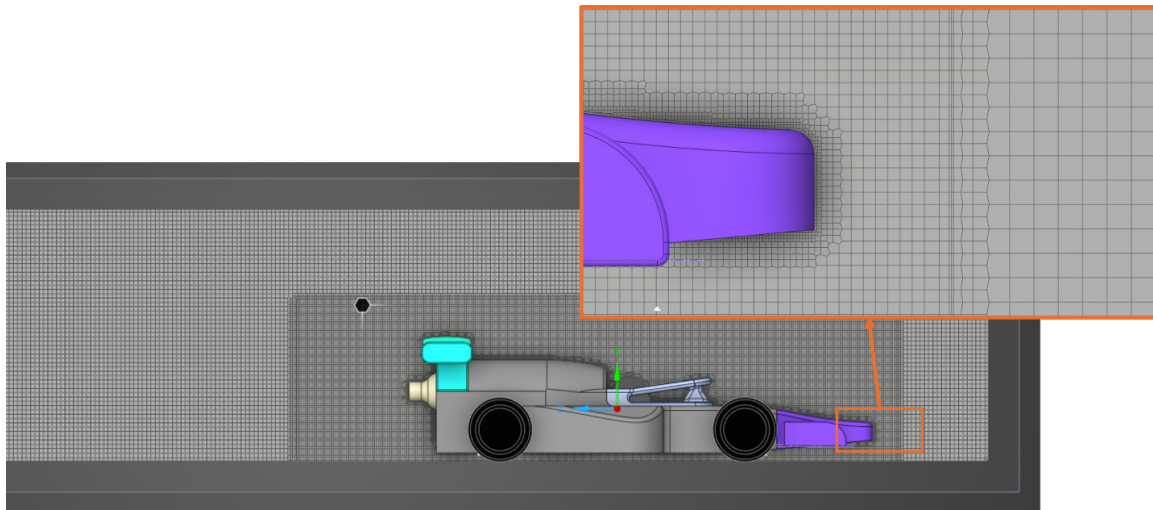


Figure 3: Mesh generated around the car to accurately capture lift and drag forces.

A body of influence (BOI) region and higher mesh fidelity near the car surface are employed to enhance the accuracy of aerodynamic force prediction.

The simulation was performed with an inlet velocity<sup>3</sup> of 85 m/s, employing a symmetrical plane to reduce computational effort while maintaining solution accuracy. The bottom wall was modeled as a moving ground plane translating at 85 m/s in the opposite direction of the movement of car (in the direction of airflow), replicating the relative motion between the car and the ground. These boundary conditions and setup details can be observed in the simulation file included with this case study.

## 4. Results and Discussions

To understand the force trends for each of the configurations, we plot the drag and lift force values of each combination described in Table 1. The table describes the different drag and lift forces associated with the various Front and Rear wing configurations.. It is important to note that the drag and lift force values presented in the results are twice the values obtained directly from the simulation file. This adjustment accounts for the use of a symmetrical plane, where only half of the car geometry was simulated. Consequently, the computed forces were doubled to represent the full-car configuration. This simplification was adopted to capture the overall aerodynamic trends and to provide an initial assessment of the lift and drag behavior, which can guide subsequent detailed analyses using more advanced Ansys tools.

---

<sup>2</sup> Higher fidelity mesh was used to increase accuracy of simulation results. To do this, significant GPU power was required; these results were run using an NVIDIA RTX 5000 Ada GPU with 16GB RAM dedicated.

If running this simulation on a less powerful machine, one may need to reduce mesh fidelity, which can impact result accuracy.

<sup>3</sup> See note at beginning of this document for clarity on why this speed was chosen

Table 1 : The table describes the different Drag and lift forces associated with the various Front and Rear wing configurations.

FW\RW	Lift Force(N)			Drag Force (N)		
	RW1	RW2	RW3	RW1	RW2	RW3
FW1	10.26	9.74	9.94	10.24	10.26	10.22
FW2	8.9	8.54	9.2	9.58	9.9	9.78
FW3	7.7	7.94	8.2	10.18	10.18	10.22

Upon analyzing the plotted aerodynamic forces, as illustrated in Figure 4, it is observed that the configuration with Front Wing 2 (FW2) exhibits the lowest drag among the three front wing designs with the rear wing RW3 showing the lowest of the drag. This indicates that FW2 provides a more aerodynamically efficient profile, minimizing resistance while maintaining flow attachment over the car body.

When examining the corresponding lift forces for the three Rear Wing configurations (RW1, RW2, and RW3) paired with FW2, the results show that the lift forces remain relatively consistent across all cases. This suggests that variations in the rear wing geometry have only a marginal influence on the overall lift behavior when the FW2 configuration is employed.

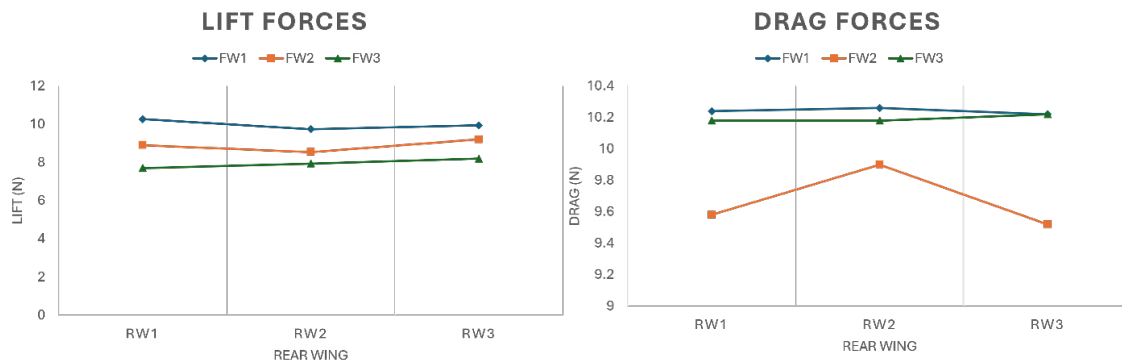


Figure 4 : Plot showing the changes in Lift and Drag forces of each of the Front and Rear wing configurations.

Based on these findings, the FW2 configuration can be considered the most favorable in terms of achieving a lower drag force without significantly compromising lift characteristics. However, since the lift variations among the rear wing configurations are minimal, a more detailed investigation using advanced Ansys tools like Ansys Fluent is recommended. These tools would allow for more refined meshing, turbulence modeling, and transient analyses to capture subtle flow interactions and wake effects that may not be fully resolved in Ansys Discovery software.

Furthermore, examining the velocity contours at the plane of symmetry of the car (Figure 5 : Velocity profile of different Rear Wing (RW1, RW2, RW3) Profiles for front wing FW1 profile, Figure 6 : Velocity profile of different Rear Wing (RW1, RW2, RW3) Profiles for front wing FW2 profile, and Figure 7 : Velocity profile of different Rear Wing (RW1, RW2, RW3) Profiles for front wing FW3 profile) for the different configurations reveals distinct patterns of flow separation and wake formation around the rear wing and diffuser regions. These visualizations provide valuable insight into how design modifications influence aerodynamic efficiency and can guide further optimization efforts. The velocity profiles



indicate that the wing positioned closer to the front tip of the car promotes smoother boundary layer development and delayed flow separation, resulting in a significant reduction in drag compared to the other two front wing configurations. Additionally, the rear wing configurations positioned closer to the car help minimize flow separation in the wake region, further reducing drag. This effect is clearly reflected in the plotted results.

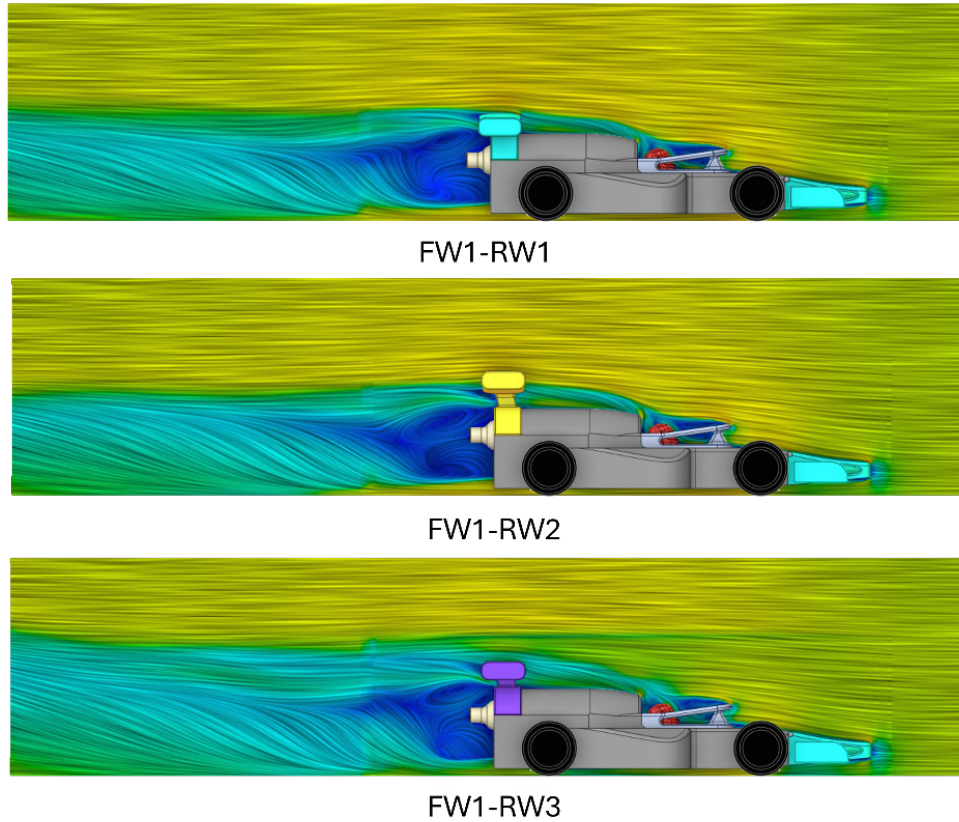


Figure 5 : Velocity profile of different Rear Wing (RW1, RW2, RW3) Profiles for front wing FW1 profile



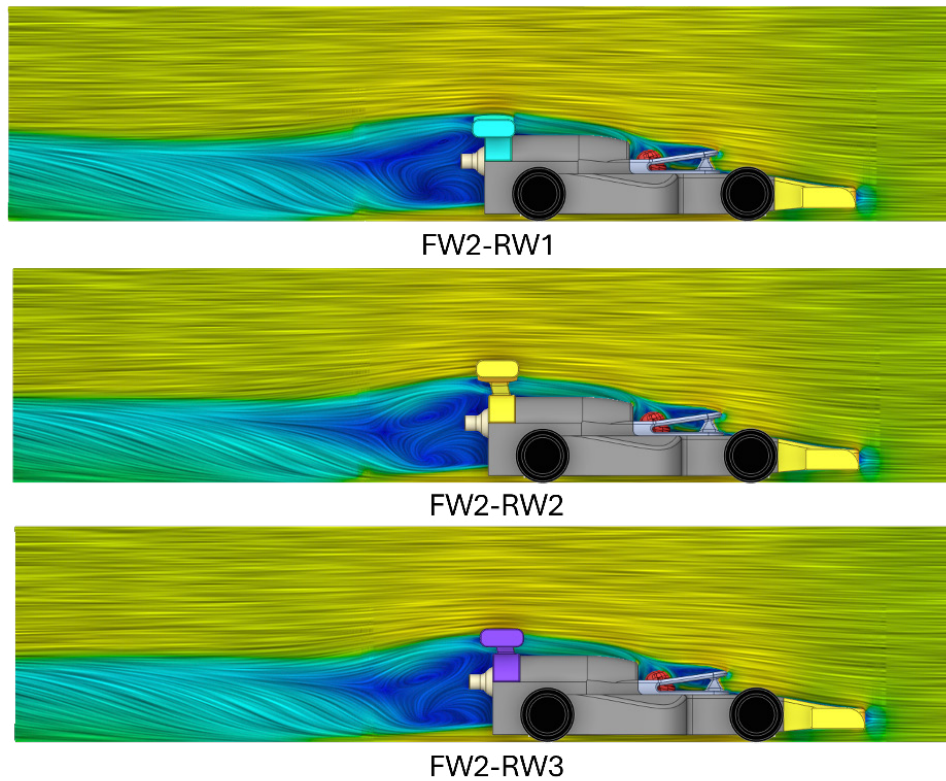


Figure 6 : Velocity profile of different Rear Wing (RW1, RW2, RW3) Profiles for front wing FW2 profile

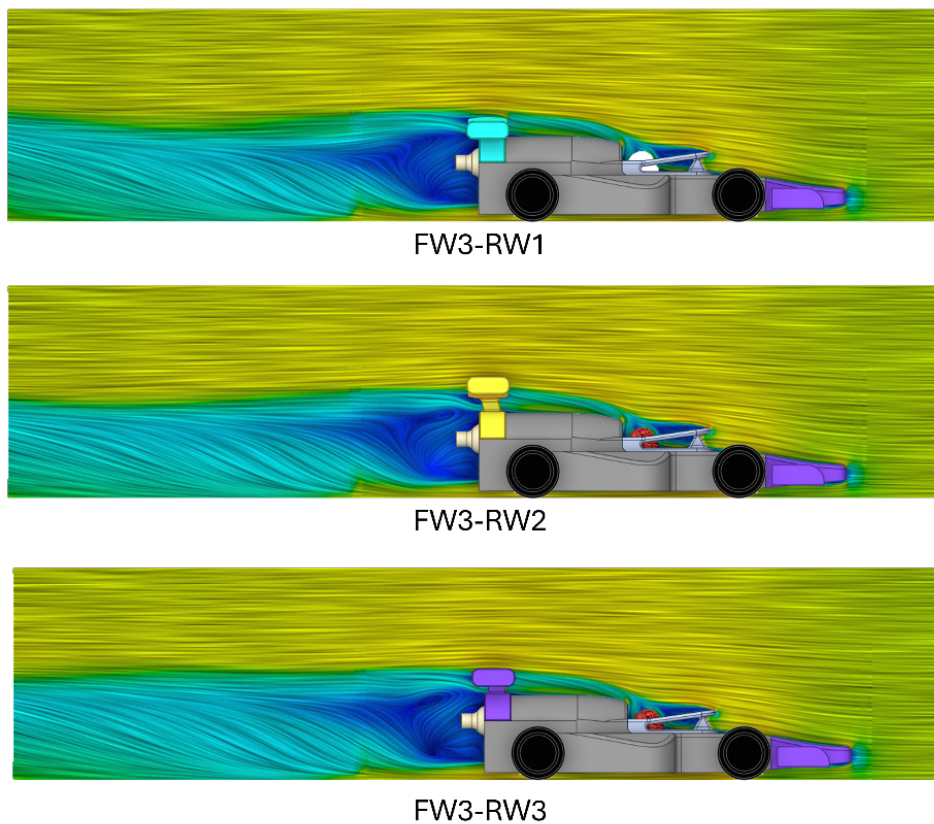


Figure 7 : Velocity profile of different Rear Wing (RW1, RW2, RW3) Profiles for front wing FW3 profile

## 5. Conclusions

The present study utilized Ansys Discovery software to evaluate the aerodynamic performance of a STEM Racing car under various front and rear wing configurations. A symmetry plane and a refined Body of Influence (BOI) meshing approach were employed to capture lift and drag forces accurately while maintaining computational efficiency.

Among the front wing configurations, Front Wing 2 (FW2) was found to produce the lowest drag. Velocity profile analysis shows that its positioning closer to the front tip of the car promotes smoother boundary layer development and delays flow separation, which contributes significantly to drag reduction. For the rear wing configurations paired with FW2, those positioned closer to the car further minimized flow separation in the wake region, leading to additional reductions in drag, as reflected in the plotted results.

Lift forces remained relatively consistent across the rear wing variations, suggesting that drag optimization can be prioritized without adversely affecting overall lift. These findings highlight FW2, in combination with an optimally positioned rear wing, as the most promising configuration for improved aerodynamic efficiency.

While the results provide valuable initial insights, it should be noted that the analysis is based on steady-state simulations with moderate mesh resolution. For a more detailed understanding of wake dynamics, pressure distributions, and transient effects, further studies using Ansys Fluent software are recommended.

This preliminary investigation using Ansys Discovery software provides a foundational understanding of the influence of wing configurations on STEM Racing aerodynamics and represents a critical first step in the vehicle's aerodynamic design process. These initial analyses serve as a steppingstone for more advanced simulations, enabling efficient exploration of design variations while reducing overall computational time in the evaluation of the car's performance.

© 2025 ANSYS, Inc. All rights reserved.

## Use and Reproduction

The content used in this resource may only be used or reproduced for teaching purposes; and any commercial use is strictly prohibited. The full Academic Terms & Conditions can be found [using this link](#).

## Document Information

This case study is part of a set of teaching resources to help introduce students to topics related to fluids.

## Ansys Education Resources

To access more undergraduate education resources, including lecture presentations with notes, exercises with worked solutions, microprojects, real life examples and more, visit [www.ansys.com/education-resources](http://www.ansys.com/education-resources).

## Feedback

Here at Ansys, we rely on your feedback to ensure the educational content we create is up-to-date and fits your teaching needs.

[Please click the link here](#) out a short survey (~7 minutes) to help us continue to support academics around the world utilizing Ansys tools in the classroom.

**ANSYS, Inc.**  
Southpointe  
2600 Ansys Drive  
Canonsburg, PA 15317  
U.S.A.  
724.746.3304  
[ansysinfo@ansys.com](mailto:ansysinfo@ansys.com)

If you've ever seen a rocket launch, flown on an airplane, driven a car, used a computer, touched a mobile device, crossed a bridge or put on wearable technology, chances are you've used a product where Ansys software played a critical role in its creation. Ansys is the global leader in engineering simulation. We help the world's most innovative companies deliver radically better products to their customers. By offering the best and broadest portfolio of engineering simulation software, we help them solve the most complex design challenges and engineer products limited only by imagination.

visit [www.ansys.com](http://www.ansys.com) for more information

Any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. All other brand, product, service and feature names or trademarks are the property of their respective owners.

© 2025 ANSYS, Inc. All Rights Reserved.