AERODYNAMIC PERFORMANCE OF VEHICLE MODELS USING CFD

SASANGA ATHURALIYA

M20020921004

AUTOMOTIVE ENGINEERING

BATCH 05

TABLE OF CONTENTS

ABSTRACT

This report focuses on developing and analysing vehicle CAD models using SolidWorks. It explains the methodologies used to create accurate representations of vehicles with major details while excluding surfacing and internal components. Three models are developed and analyzed using SolidWorks Computational fluid dynamics to study aerodynamic forces at a speed of 30 m/s and a temperature of 25°C, assuming air behaves as an ideal gas. Drag coefficients are calculated for each model and the results are compared with mathematical results. The study highlights how design improvements influence aerodynamic performance and demonstrates the importance of combining CAD modelling with CFD simulations for optimizing vehicle designs to achieve better performance.

INTRODUCTION

In automotive design, creating accurate representations of vehicles and understanding their aerodynamic behaviour is crucial. Engineers use Computer-Aided Design (CAD) software like SolidWorks to design and analyze vehicles. SolidWorks is a powerful tool that helps create precise 3D models to enable engineers to test and optimize designs before physical prototypes are built. These models include essential elements like body shape and dimensions, excluding internal components such as the engine, gearbox and auxiliary systems.

Aerodynamics plays a vital role in vehicle performance, especially for high-speed cars like Formula 1 cars. Aerodynamics is the study of how air flows around objects, affecting factors like drag and downforce. The drag coefficient (C_d) is a measure of how much resistance an object faces when moving through air. Reducing the drag coefficient helps vehicles achieve higher speeds and better fuel efficiency. In F1 cars, aerodynamic design focuses on minimizing drag while maximizing downforce to improve stability and cornering performance.

To analyze these aerodynamic effects, engineers use Computational Fluid Dynamics (CFD). CFD simulates airflow around a vehicle model to provide insights into drag, lift, and other forces. By modelling air as an ideal gas in CFD software like SolidWorks Flow Simulation calculates aerodynamic forces under specific conditions. For this project, I simulated airflow at 30 m/s, exactly 108km/hr and 25°C to study how the design affects drag and overall performance.

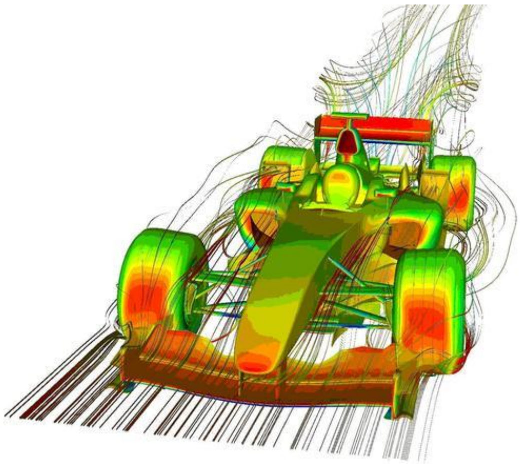


Figure 1 Aerodynamics analysis of F1 car: Jobs in F1: How to become a Formula 1 aerodynamicist (no date) https://motorsportengineer.net. https://motorsportengineer.net/wp-content/uploads/2021/10/Hadron-F1-Experience_grande.jpeg.

This study focuses on generating a detailed F1 car model using SolidWorks and analyzing its aerodynamic performance. Three versions of the F1 model will be created with improvements in design and their aerodynamic properties will be evaluated through CFD and mathematical calculations. This approach highlights how CAD and CFD tools contribute to optimizing the design of high-performance vehicles like F1 cars.

THEORY

PARAMETRIC DESIGN

Parametric design is a method in CAD modelling that uses parameters such as dimensions, constraints and relationships to define and control a model. In SolidWorks, parametric modelling allows engineers to create flexible designs that can be easily updated. For example, adjusting the wheelbase of an F1 car model automatically updates the related components to ensure accuracy and consistency. This approach is crucial in designing complex structures like F1 cars, where every dimension impacts performance. Creating a precise F1 car model requires attention to detail and adherence to best practices such as,

- Using accurate reference dimensions from existing F1 car designs and applying the design to refine key aerodynamic elements like wings and bodywork
- Ensuring smooth transitions between surfaces to maintain aerodynamic efficiency.
- Avoiding unnecessary details like surfacing, that do not influence performance.

Figure 2 ADVANCED CAD CAM (2022) Formula 1 modelling using SolidWorks: Referencing actual F1 car to the three dimensional planes. https://www.youtube.com/watch?v=d4JekwWQA3w.

BASIC PRINCIPLES OF DRAG, LIFT AND AIRFLOW BEHAVIOR

Aerodynamics is critical in high-performance vehicles like F1 cars. It involves understanding how air flows around the vehicle and the influence of the drag, lift and downforce.

- Drag: The resistance a vehicle faces when moving through air. F1 cars aim to minimize drag to achieve higher speeds.
- Lift and Downforce: While a lift is undesirable in F1 cars, downforce is crucial. Downforce pushes the car toward the ground to improve grip and stability at high speeds.
- Airflow: Streamlined shapes guide airflow smoothly around the car to reduce turbulence and drag.

In F1 motorsport, reducing drag is a key focus to enhance speed and performance. The drag coefficient (C_d) is used instead of drag force because it is a dimensionless value that simplifies aerodynamic analysis.

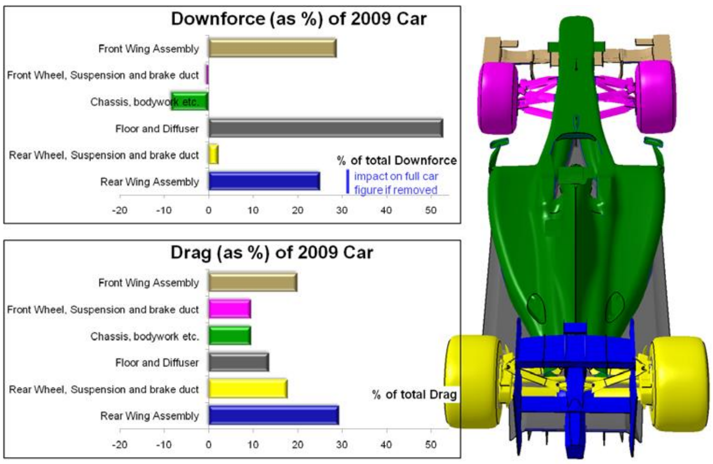


Figure 3 Insight into Aerodynamics basics – downforce and drag breakdown into major components. (From 'Aeronautical Journal Jan 2013', courtesy of Willem Toet, Head of Aerodynamics, Sauber F1 Team, Sauber Motorsport AG) (no date). https://www.formula1-dictionary

DRAG FORCE FORMULA

The drag force (F_d) acting on a vehicle is calculated using the formula:

$$F_d = \frac{1}{2}\rho V^2 C_d A$$

F_d: Drag force

ρ: Air density

V: Vehicle velocity

C_d: Drag coefficient

A: Frontal area of the vehicle

AIRFLOW BEHAVIOR

When air flows around an F1 car, it starts as **laminar air** (smooth and parallel flow) but becomes **turbulent air** (chaotic and disturbed) when it reaches the separation point, usually at the car's rear. This creates a low-pressure zone, leading to drag due to the pressure difference between the front and rear of the car.

F1 engineers apply several strategies like below mentioned to minimize drag.

- S-ducts: Smooth airflow from the front wing to reduce turbulence.
- Smooth Body Design: Reduces disruptions in airflow.
- Winglets: Enhance airflow management at the front and rear wings.

F1 cars have low drag coefficients and small frontal areas to help them achieve high speeds while maintaining stability. Despite these techniques, F1 cars have higher drag coefficients between 0.7-1.2 compared to regular cars which are usually between 0.3-0.7. This is due to the unique aerodynamic needs of F1 cars to balance drag reduction with the need for downforce to maintain stability and control at high speeds. By understanding and optimizing drag, F1 teams can improve their car's performance on the track.

BASICS OF CFD

CFD is a numerical method used to analyze airflow around objects. In this project, SolidWorks Flow Simulation is used to study the aerodynamic behaviour of an F1 car model. CFD divides the space around the car into small elements and solves equations for airflow behaviour within each element. This provides detailed insights into drag, lift and other forces acting on the vehicle.

To simplify the analysis, certain assumptions are made during the simulation.

- Air is treated as an ideal gas with uniform properties.
- The flow is steady and uniform.
- External factors like turbulence are minimized in the simulation setup.

CFD simulations help visualize airflow patterns and identify areas of high drag or turbulence. By combining these results with drag force calculations, engineers can optimize the F1 car's design for better aerodynamic performance.

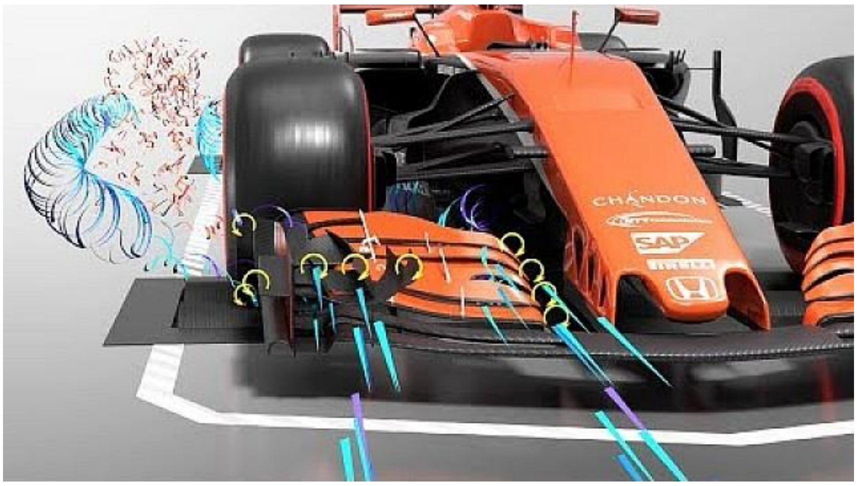


Figure 4 mcomstaff (2017) F1 3D airflow - the new era is upon us! (2/2). https://www.youtube.com/watch?v=Un7C5i9WsNs.

METHODOLOGY

CAD MODELLING

As mentioned earlier, the F1 car model was made using reference images through three three-dimensional planes by following the tutorials.

Figure 5 ADVANCED CAD CAM (2022) Formula 1 modelling using SolidWorks: Referencing actual F1 car to the three dimensional planes. https://www.youtube.com/watch?v=d4JekwWQA3w.

F1 Car Model 01

The first model is the base version out of three models and it shows the usual F1 car at the track.

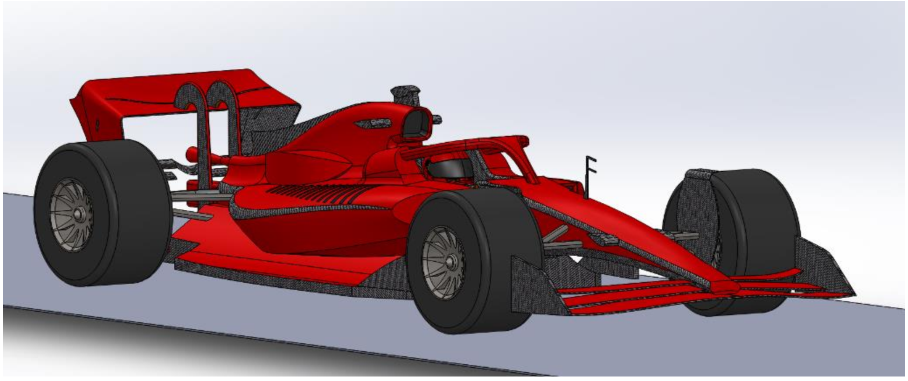


Figure 6 Base version of the usual F1 car

F1 Car Model 02

The second model is the base version with the Drag Reduction System (DRS) turned on. DRS is a technology used in F1 to improve overtaking by reducing aerodynamic drag. It is a movable flap on the rear wing of a F1 car and it allows a driver to adjust the rear wing of their car to create less downforce and drag to increase straight-line speed.

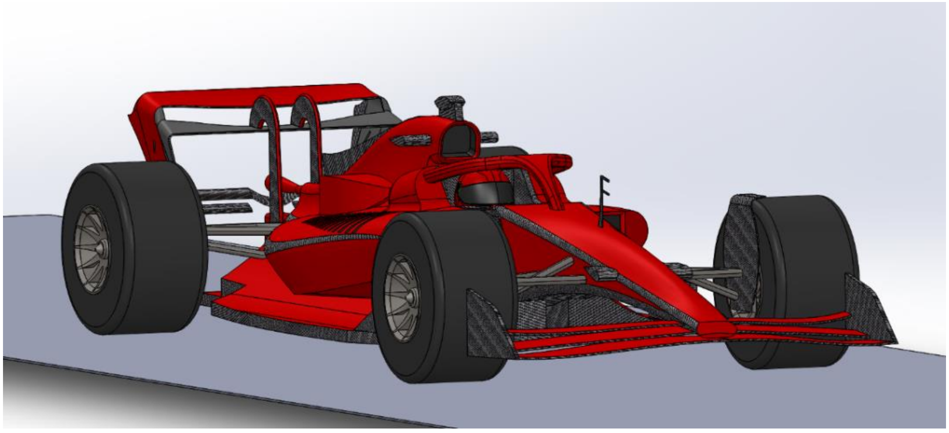


Figure 8 Base version with the Drag Reduction System (DRS) turned on

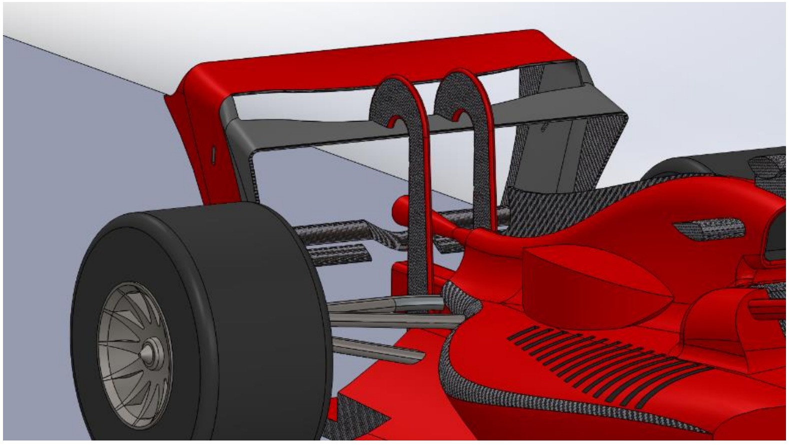


Figure 7 Close up view of the Drag Reduction System (DRS) of the second F1 car model

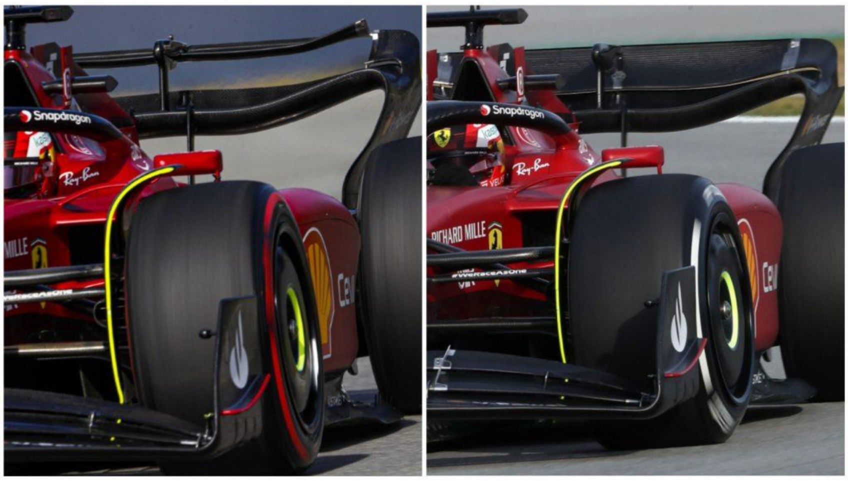


Figure 9 PlanetF1 (2022) 'What is F1's 'DRS' and how does it work?: At left side shows DRS active and right side shows DRS inactive,' PlanetF1, 15 July. https://www.planetf1.com/features/what-is-f1s-drs-and-how-does-it-work.

F1 Car Model 03

And the third model is the base version with the Drag Reduction System (DRS) and updated front Splitter (wing) pack.

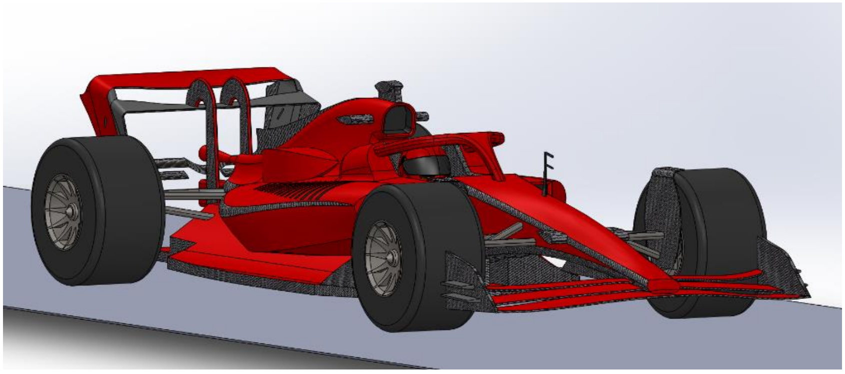


Figure 11 Base version with the Drag Reduction System (DRS) and updated front Splitter (wing) pack

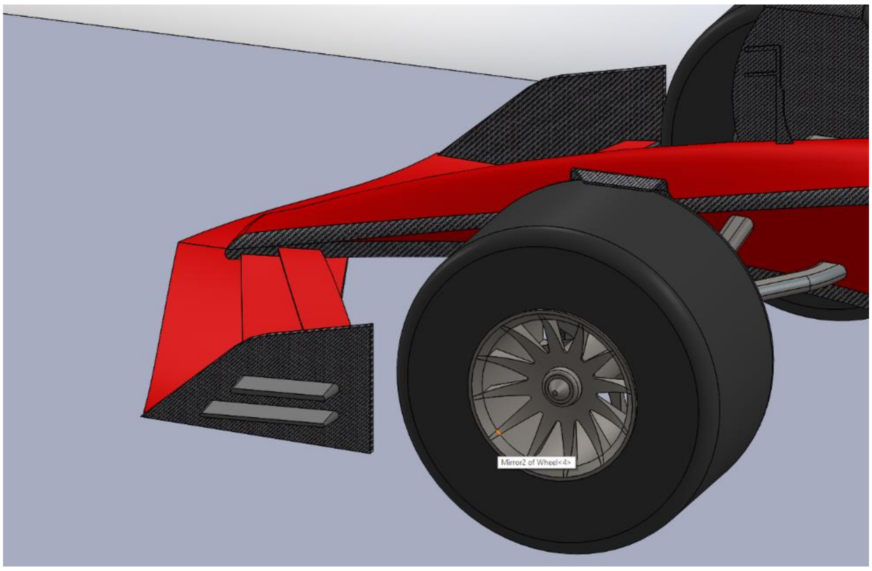


Figure 10 Close up view of the Drag Reduction System (DRS) and updated front Splitter (wing) pack of the second F1 car model

The front splitter is an aerodynamic component located at the very front of a F1 car, just below the nose. Its primary purpose is to manage airflow and enhance the car's aerodynamic performance. Engineers optimize the shape and angle of the splitter for maximum efficiency based on track requirements. The splitter works with the car's floor, diffuser and wings to create a balanced aerodynamic package.

Figure 12 Oracle Red Bull Racing 2022 F1® Front Wing & Nose Official Replica (no date). https://www.f1authentics.com/products/oracle-red-bull-racing-2022-official-replica-front-wing-nose.

CFD ANALYSIS

Following these steps ensures a comprehensive setup, simulation and documentation process for the SolidWorks flow simulation.

1. Setting Up the Model
   - Open your CAD model in SolidWorks.
   - Verify the model is properly sealed with no open surfaces or gaps to ensure simulation accuracy.
   - Save the file in a SolidWorks-compatible format.
2. Activating the Flow Simulation Add-in
   - Go to the Tools menu.
   - Click on Add-Ins and enable SolidWorks Flow Simulation.
3. Creating a New Project
   - Open the Flow Simulation Wizard from the Flow Simulation tab.
   - Select External Analysis since the simulation involves airflow around the object.
   - Choose SI Units for the unit system to match the input parameters.
   - Put gravitational force for the correct axis. Name the project and click Next.

- Name the project appropriately and click Next.

4. Setting the Analysis Parameters
    - Fluid Selection:
        - Choose Air as the fluid.
        - Ensure Ideal Gas behaviour is checked for air properties.
    - Initial Conditions:
        - Set Velocity to 30 m/s in the required direction.
        - Set Temperature to 25°C.
    - Accept the default turbulence model for the analysis.

5. Defining Boundary Conditions
    - Velocity Inlet:
        - Select a plane or face as the inlet where air enters.
        - Define the velocity as 30 m/s in the desired direction.
    - Pressure Outlet:
        - Assign a plane or face as the outlet.
        - Use the default atmospheric pressure of 101325 Pa.
    - Ensure all boundary conditions reflect the real-world scenario accurately.

6. Creating the Computational Domain
    - Adjust the computational domain to surround the model adequately:
        - The domain should extend far enough from the model to allow smooth airflow and minimize boundary effects.
    - Ensure that the domain size balances computational efficiency with accuracy.

7. Running the Simulation
    - Click on Run to start the flow simulation.
    - Monitor the convergence of the solution:

- Ensure residuals (errors) fall below acceptable thresholds.
- Check for consistency in drag and pressure values over iterations.

8. Viewing and Analyzing Results

- Use SolidWorks post-processing tools to examine the following:
  - Streamlines: Visualize airflow patterns around the model.
  - Velocity Contours: Identify areas with high and low airflow velocities.
  - Pressure Distribution: Analyze pressure changes across the surface.
  - Force Reports: Extract drag and lift force values for analysis.
- Capture screenshots of key results like streamlines, velocity contours and pressure plots.

9. Exporting and Saving Results

- Save all graphical results as images for documentation:
  - Go to File and Save As and choose image formats like PNG or JPEG.
- Export numerical data such as drag and lift forces into an Excel sheet or a text file:
  - Use the Export Data option in the results section.
- Generate a report directly from SolidWorks:
  - Go to the Reports section and select Generate Report.
  - Customize the report to include all relevant findings.
- Save the project with all results:
  - Go to File and Save to ensure the simulation setup and results are stored for future use.

10. Preparing for Documentation

- Organize the results into a Word document:
  - Import saved images and data into a structured report format.
  - Add descriptive captions to images and provide detailed explanations of findings.

- Save the document with a clear name (e.g., "F1 Flow Simulation Report").

11. Other model simulations
    - Follow the above mentioned steps for the other two models to find the Flow Simulation report to compare CFD results with theoretical drag force calculations.

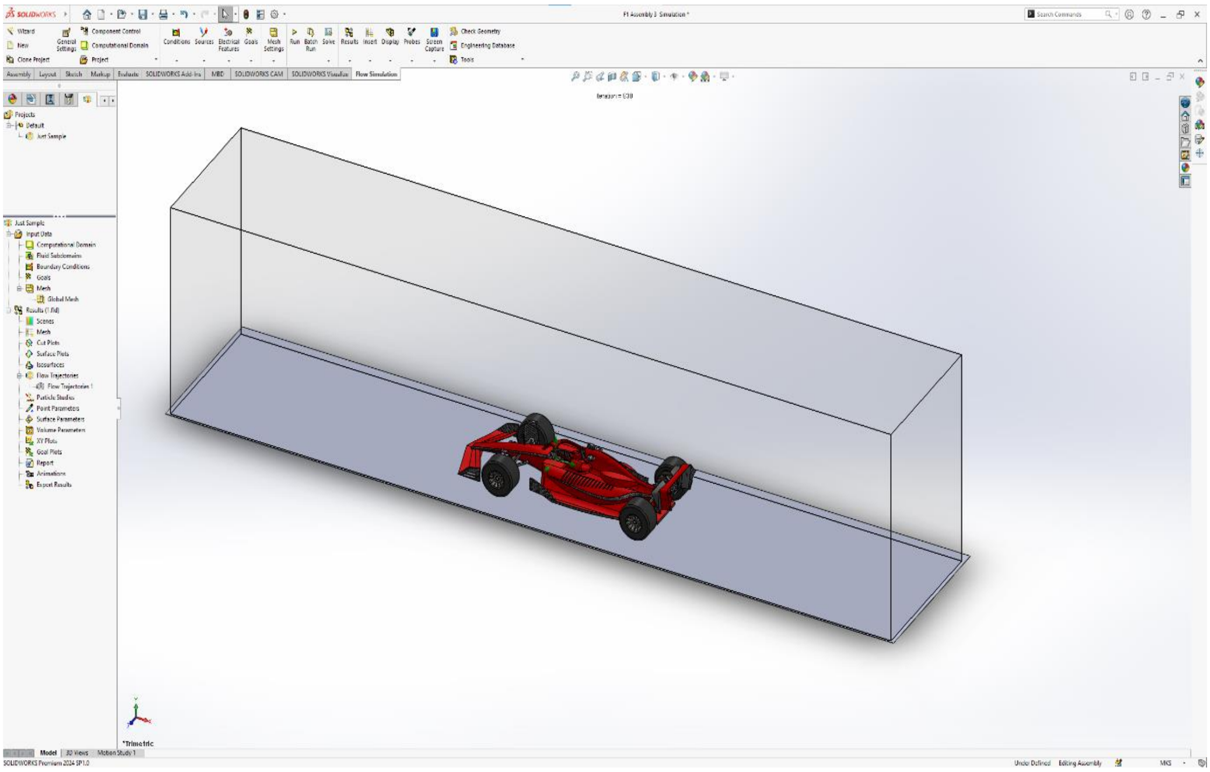


Figure 13 Interface after the CFD Simulation of the F1 car

RESULTS

Analysis Environment

Software Product: Flow Simulation 2024 SP1.0. Build: 6234

CPU Type: AMD Ryzen 7 7800X3D 8-Core Processor

CPU Speed: 4201 MHz

RAM: 31904 MB / 21383 MB

Operating System: Windows 10 (Version 10.0.19045)

Model Information

Model Name: F1 Assembly Simulation.SLDASM

Project Name: Simulation

Project Comments:

Unit System: SI (m-kg-s)

Analysis Type: External (not exclude internal spaces)

Additional Physical Calculation Options

Heat Transfer Analysis: Fluid Flow: On Conduction: Off

Flow Type: Laminar and turbulent

Time-Dependent Analysis: Off

Gravity: On

Radiation:

Humidity: Off

Default Wall Roughness: 0 micrometer

Material Settings

Fluids: Air

Initial Conditions: Ambient Conditions

Thermodynamic parameters	Static Pressure: 101325.00 Pa Temperature: 298.15 K
Velocity parameters	Velocity vector Velocity in X direction: 30.000 m/s Velocity in Y direction: 0 m/s Velocity in Z direction: 0 m/s
Turbulence parameters	Turbulence intensity and length Intensity: 0.10 %

	Length: 6.313e-04 m

Engineering Database

Gases: Air

Path: Gases Pre-Defined

Specific heat ratio (Cp/Cv): 1.399

Molecular mass: 0.0290 kg/mol

F1 CAR MODEL 01

Analysis Time

Calculation Time: 2716 s

Number of Iterations: 838

Warnings:

Results

Analysis Goals

Name	Unit	Value	Progress	Criteria	Delta	Use in convergence
GG Maximum Static Pressure 1	Pa	102834.20	100	532.166419	2.28067488	On
GG Maximum Total Pressure 2	Pa	103150.87	100	524.470924	6.6450657	On
GG Maximu	m/s	43.864	100	0.789445775	0.0097992844	On

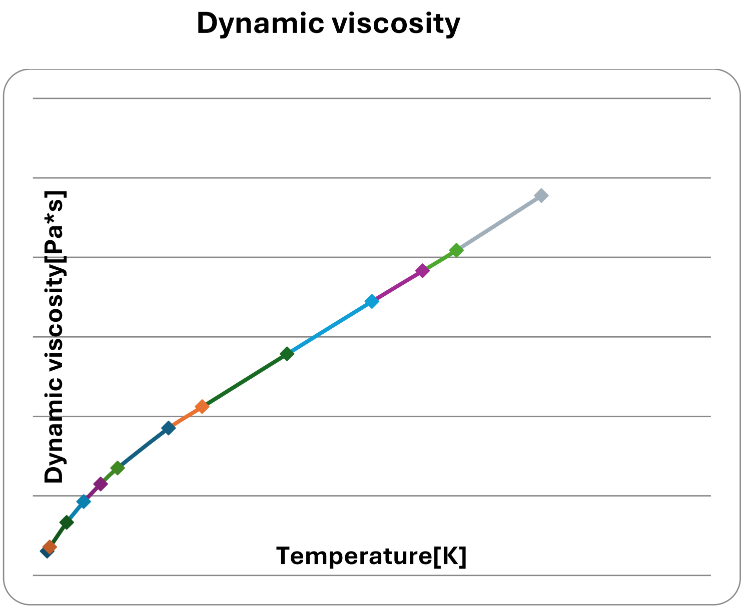
m Velocity 3						
GG Maximum Velocity (X) 4	m/s	43.139	100	0.546123266	0.0100887224	On
GG Maximum Velocity (Y) 5	m/s	26.897	100	0.49246432	0.250164787	On
GG Maximum Velocity (Z) 6	m/s	34.025	100	0.647847478	0.293322201	On
GG Force 7	N	1.595	100	0.257897261	0.00141437295	On
GG Force (X) 8	N	1.595	100	0.25540112	0.00140528289	On
GG Force (Y) 9	N	0.005	100	0.0390966633	0.00192790257	On
GG Force (Z) 10	N	0.003	59	0.00244996387	0.00415019597	On
GG Friction Force 11	N	0.263	100	0.0126692805	0.000668557905	On
GG Friction	N	0.263	100	0.0126612953	0.000666437333	On

Force (X) 12						
GG Friction Force (Y) 13	N	0.009	100	0.000470484933	7.45647769e-05	On
GG Friction Force (Z) 14	N	5.779e-05	26	5.16208987e-05	0.000194815681	On
Equation Goal 1	N	0.826	100	0.132358037	0.000728268081	On

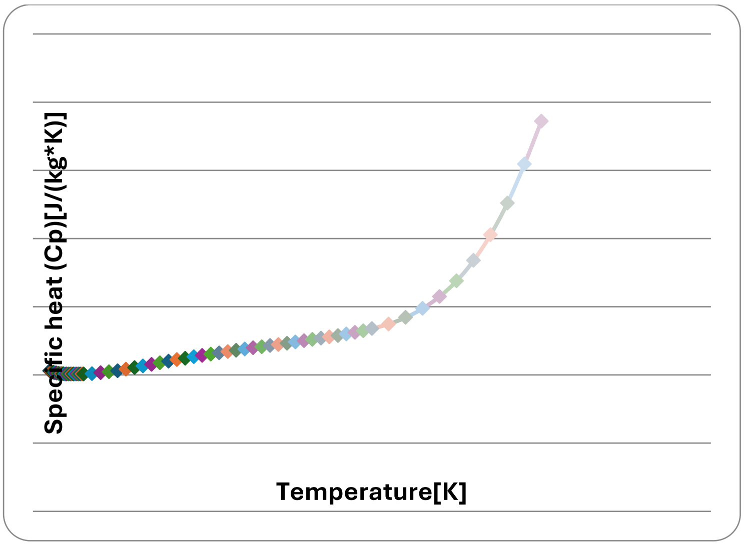
Global Min-Max-Table

Name	Minimum	Maximum
Density (Fluid) [kg/m^3]	1.17	1.20
Pressure [Pa]	100373.01	102834.20
Temperature [K]	297.62	298.61
Temperature (Fluid) [K]	297.62	298.61
Velocity [m/s]	0	41.122
Velocity (X) [m/s]	-14.593	39.851
Velocity (Y) [m/s]	-20.001	21.276
Velocity (Z) [m/s]	-31.506	30.992
Mach Number [ ]	0	0.12
Velocity RRF [m/s]	0	41.122
Velocity RRF (X) [m/s]	-14.593	39.851
Velocity RRF (Y) [m/s]	-20.001	21.276
Velocity RRF (Z) [m/s]	-31.506	30.992
Vorticity [1/s]	0	130309.76
Relative Pressure [Pa]	-951.99	1509.20

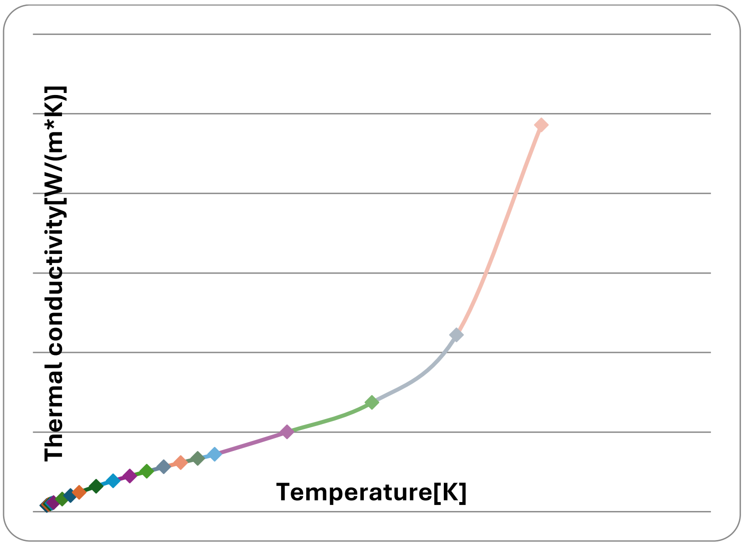
Shear Stress [Pa]	0	26.03
Bottleneck Number [ ]	0	1.0000000
Heat Transfer Coefficient [W/m^2/K]	0	0
ShortCut Number [ ]	0	1.0000000
Surface Heat Flux [W/m^2]	0	0
Surface Heat Flux (Convective) [W/m^2]	0	0
Total Enthalpy Flux [W/m^2]	-1.080e+07	1.078e+07
Acoustic Power [W/m^3]	0	0.003
Acoustic Power Level [dB]	0	94.94



Specific heat (Cp)



Thermal conductivity



F1 CAR MODEL 02

Analysis Time

Calculation Time: 1459 s

Number of Iterations: 474

Warnings:

Results

Analysis Goals

Name	Unit	Value	Progress	Criteria	Delta	Use in convergence
GG Maximum Static Pressure 1	Pa	102508.68	100	979.260622	57.9126831	On
GG Maximum Total Pressure 2	Pa	102975.22	100	962.97686	1.17623899	On
GG Maximum Velocity 3	m/s	43.405	100	0.679891657	0.0213563775	On
GG Maximum Velocity (X) 4	m/s	42.111	100	0.559243982	0.017543992	On
GG Maximum Velocity (Y) 5	m/s	23.190	100	0.577926447	0.293782155	On
GG Maximum Velocity (Z) 6	m/s	33.186	100	0.651096836	0.0377192774	On

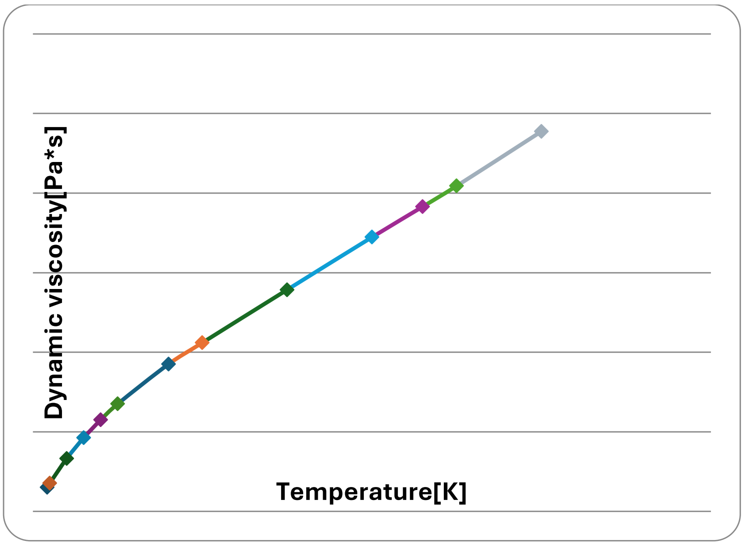
GG Force 7	N	1.529	100	0.242602101	0.00997234716	On
GG Force (X) 8	N	1.527	100	0.241590191	0.00995854385	On
GG Force (Y) 9	N	0.074	100	0.0349351296	0.00449001328	On
GG Force (Z) 10	N	0.002	100	0.00130694118	0.00114529678	On
GG Friction Force 11	N	0.260	100	0.0128173294	0.00108851639	On
GG Friction Force (X) 12	N	0.260	100	0.0128104267	0.0010911547	On
GG Friction Force (Y) 13	N	0.009	100	0.00044962112	7.21458032e-05	On
GG Friction Force (Z) 14	N	8.137e-05	100	5.61702315e-05	5.48974788e-05	On
Equation Goal 1	N	0.791	100	0.125200717	0.00516087522	On

Global Min-Max-Table

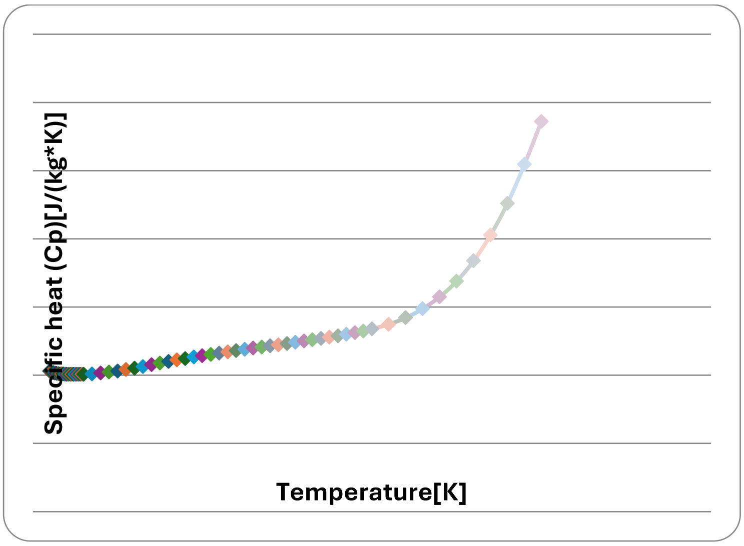
Name	Minimum	Maximum
Density (Fluid) [kg/m^3]	1.17	1.20
Pressure [Pa]	100423.94	102508.68

Temperature [K]	297.65	298.61
Temperature (Fluid) [K]	297.65	298.61
Velocity [m/s]	0	40.824
Velocity (X) [m/s]	-15.866	40.329
Velocity (Y) [m/s]	-20.867	22.875
Velocity (Z) [m/s]	-29.110	28.861
Mach Number [ ]	0	0.12
Velocity RRF [m/s]	0	40.824
Velocity RRF (X) [m/s]	-15.866	40.329
Velocity RRF (Y) [m/s]	-20.867	22.875
Velocity RRF (Z) [m/s]	-29.110	28.861
Vorticity [1/s]	4.91e-03	147008.60
Relative Pressure [Pa]	-901.06	1183.68
Shear Stress [Pa]	0	44.28
Bottleneck Number [ ]	5.5564842e-17	1.0000000
Heat Transfer Coefficient [W/m^2/K]	0	0
ShortCut Number [ ]	6.8538651e-18	1.0000000
Surface Heat Flux [W/m^2]	0	0
Surface Heat Flux (Convective) [W/m^2]	0	0
Total Enthalpy Flux [W/m^2]	-1.079e+07	1.078e+07
Acoustic Power [W/m^3]	0	0.249
Acoustic Power Level [dB]	0	113.96

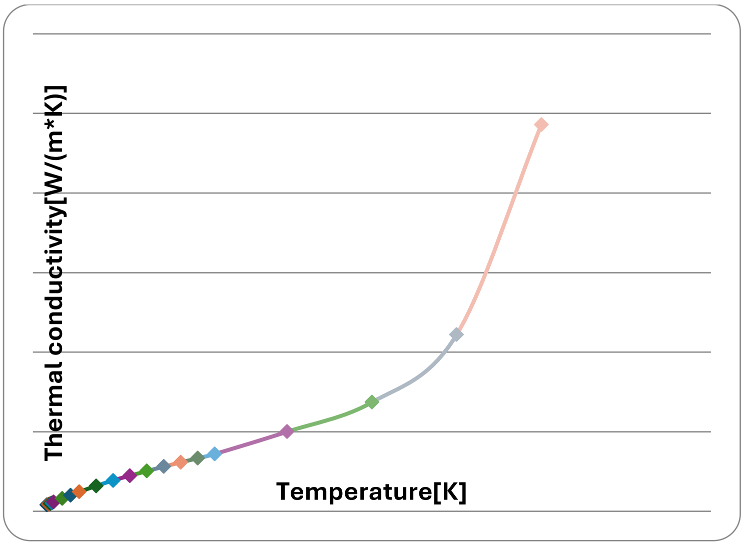
Dynamic viscosity



Specific heat (Cp)



Thermal conductivity



F1 CAR MODEL 03

Analysis Time

Calculation Time: 2578 s

Number of Iterations: 830

Warnings:

Results

Analysis Goals

Name	Unit	Value	Progress	Criteria	Delta	Use in convergence
GG Maximum Static Pressure 1	Pa	102299.24	100	977.427448	4.58275984	On
GG Maximum Total Pressure 2	Pa	103128.54	100	952.03006	0.899169961	On

GG Maximum Velocity 3	m/s	45.042	100	0.689429062	0.00930589251	On
GG Maximum Velocity (X) 4	m/s	43.518	100	0.536945562	0.0126500235	On
GG Maximum Velocity (Y) 5	m/s	28.057	100	0.495233363	0.124319636	On
GG Maximum Velocity (Z) 6	m/s	36.419	100	0.587763658	0.137597512	On
GG Force 7	N	1.492	100	0.235042786	0.0024207953	On
GG Force (X) 8	N	1.492	100	0.234063499	0.00237196688	On
GG Force (Y) 9	N	0.020	100	0.0300506101	0.00198987645	On
GG Force (Z) 10	N	-0.017	18	0.00180975985	0.00992861151	On
GG Friction Force 11	N	0.262	100	0.012939534	0.000794198253	On
GG Friction	N	0.262	100	0.0129328204	0.000794962416	On

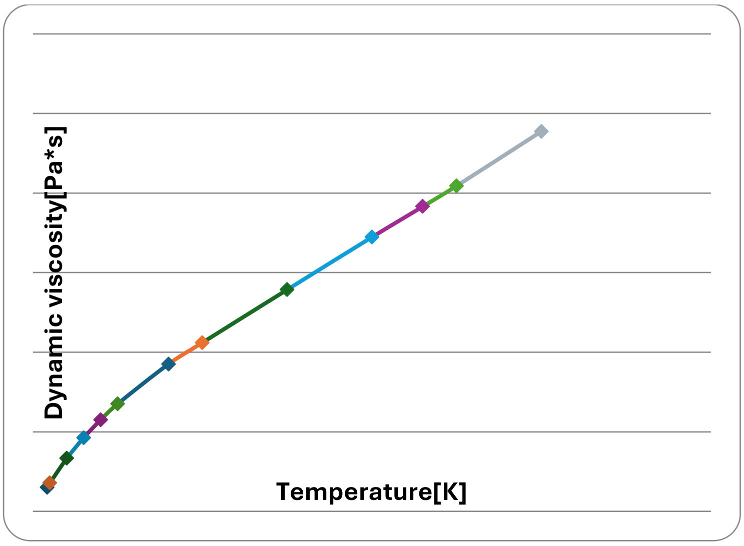
Force (X) 12						
GG Friction Force (Y) 13	N	0.009	100	0.000464731837	0.000113862654	On
GG Friction Force (Z) 14	N	0.001	8	7.82290432e-05	0.000965209131	On
Equation Goal 1	N	0.773	100	0.121300115	0.00122923846	On

Global Min-Max-Table

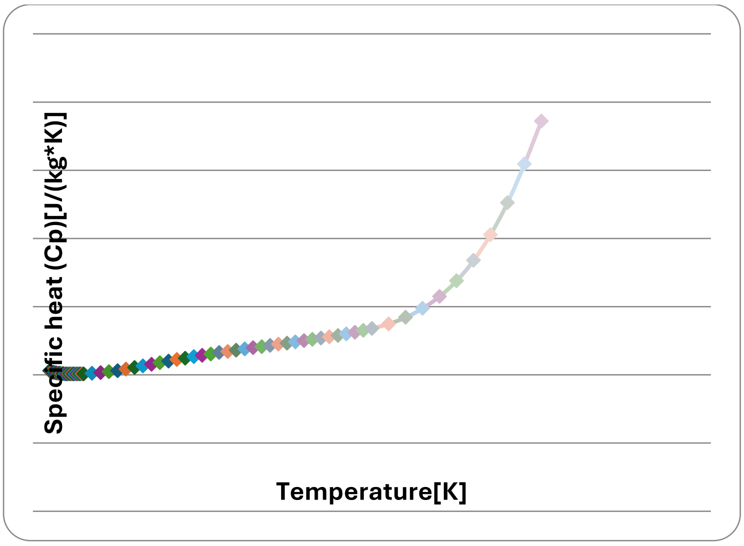
Name	Minimum	Maximum
Density (Fluid) [kg/m^3]	1.17	1.20
Pressure [Pa]	100472.71	102299.24
Temperature [K]	297.57	298.61
Temperature (Fluid) [K]	297.57	298.61
Velocity [m/s]	0	41.856
Velocity (X) [m/s]	-14.135	40.809
Velocity (Y) [m/s]	-21.041	22.138
Velocity (Z) [m/s]	-30.491	31.347
Mach Number [ ]	0	0.12
Velocity RRF [m/s]	0	41.856
Velocity RRF (X) [m/s]	-14.135	40.809
Velocity RRF (Y) [m/s]	-21.041	22.138
Velocity RRF (Z) [m/s]	-30.491	31.347
Vorticity [1/s]	5.20e-03	97674.87

Relative Pressure [Pa]	-852.29	974.24
Shear Stress [Pa]	0	19.45
Bottleneck Number [ ]	3.7949018e-18	1.0000000
Heat Transfer Coefficient [W/m^2/K]	0	0
ShortCut Number [ ]	6.6954582e-17	1.0000000
Surface Heat Flux [W/m^2]	0	0
Surface Heat Flux (Convective) [W/m^2]	0	0
Total Enthalpy Flux [W/m^2]	-1.079e+07	1.078e+07
Acoustic Power [W/m^3]	0	6.411e-04
Acoustic Power Level [dB]	0	88.07

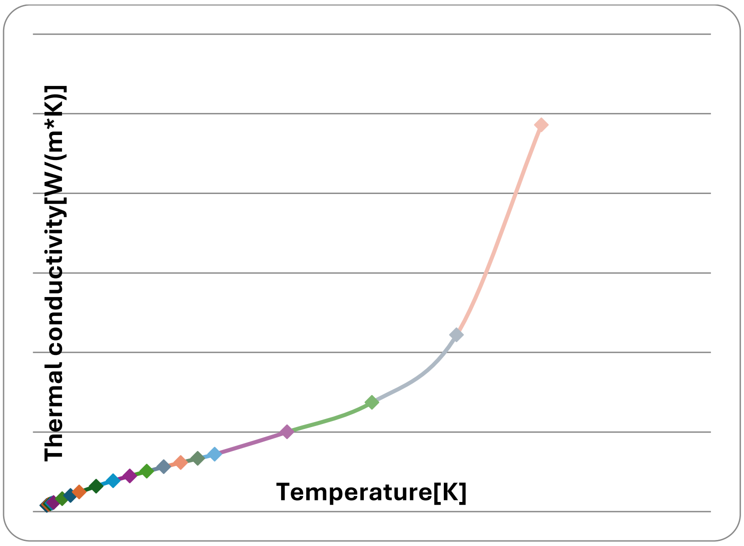
Dynamic viscosity



Specific heat (Cp)



Thermal conductivity



DISCUSSION

The CFD flow simulation results for the three F1 car models provide valuable insights into the effects of aerodynamic modifications such as the Drag Reduction System (DRS) and updated front splitter. Below is a comparative analysis of the key parameters, including static pressure, total pressure, velocity, force, friction force and drag coefficient.

1. Static Pressure

Static pressure measures the force exerted by air when it is at rest relative to the car.

- Model 01: The base model shows a maximum static pressure of 102834.20 Pa, representing the typical pressure distribution on an F1 car during a race.
- Model 02: With DRS activated, the maximum static pressure decreases to 102508.68 Pa. This reduction occurs as DRS lowers the resistance on the rear wing, allowing smoother airflow and reducing pressure build-up.
- Model 03: The addition of an updated front splitter in Model 03 further reduces static pressure to 102299.24 Pa, indicating improved airflow management over the car's surface.

2. Total Pressure

Total pressure combines static pressure and dynamic pressure due to airflow velocity.

- Model 01: The maximum total pressure is 103150.87 Pa, reflecting the combined aerodynamic forces acting on the base car.
- Model 02: DRS reduces the total pressure to 102975.22 Pa, showing a decrease in aerodynamic resistance.
- Model 03: The updated front splitter optimizes airflow further, increasing the total pressure to 103128.54 Pa, signifying a more efficient conversion of airflow into aerodynamic forces.

3. Velocity

Velocity analysis highlights airflow speed around the car.

- Model 01: The maximum velocity is 43.864 m/s, typical for a standard F1 car.

- Model 02: With DRS activated, velocity slightly decreases to 43.405 m/s. This indicates reduced turbulence and drag at the rear wing.
- -Model 03: The updated splitter increases velocity to 45.042 m/s, suggesting better airflow acceleration due to improved aerodynamic efficiency.

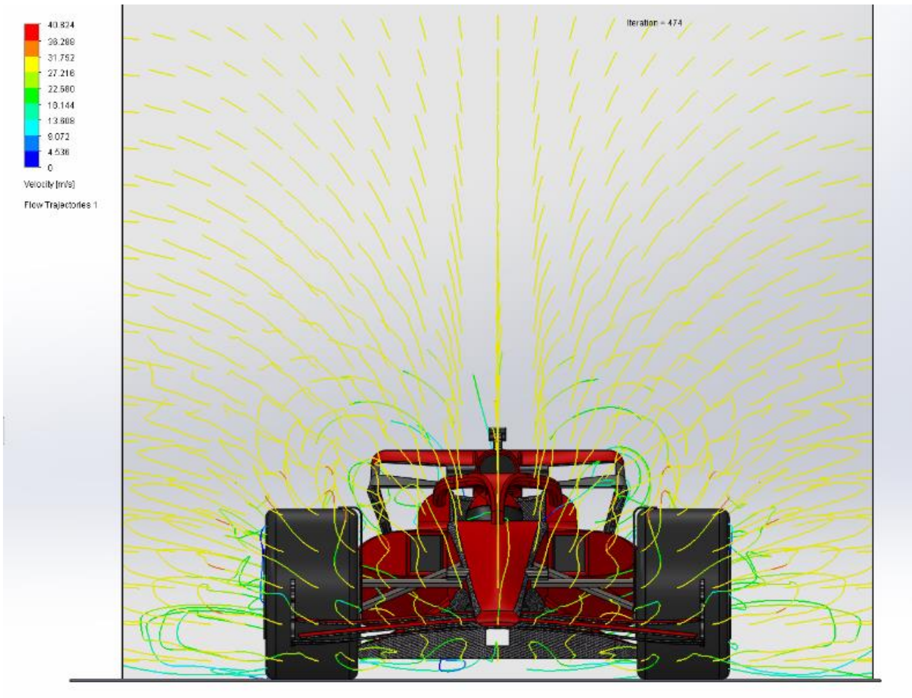


Figure 15 Velocity distribution of the F1 car model with the active Drag Reduction System (DRS) at the front

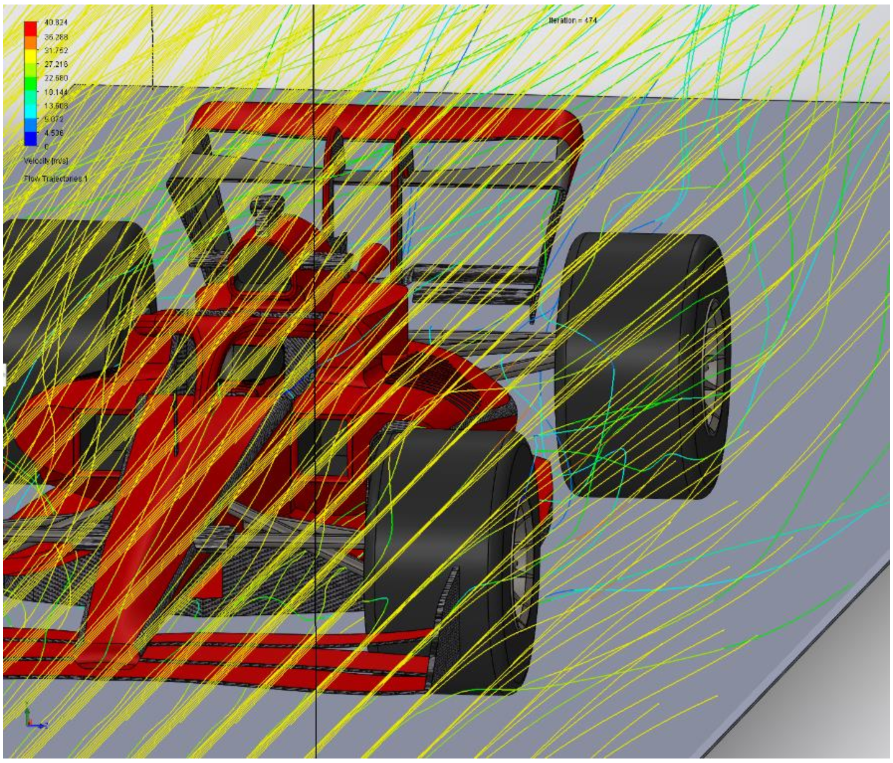


Figure 14 Velocity distribution of the F1 car model with the active Drag Reduction System (DRS) at the front and the side

4. Force

The aerodynamic forces acting on the car directly impact its performance and stability.

- Model 01: The total force on the base model is 1.595 N.
- Model 02: With DRS, the force reduces to 1.529 N, as DRS reduces drag and the force exerted on the car.
- Model 03: The updated splitter further reduces the force to 1.492 N, demonstrating enhanced aerodynamic control.

5. Friction Force

The friction force is the resistive force due to airflow interacting with the car's surface.

- Model 01: The base model experiences a friction force of 0.263 N.
- Model 02: The friction force decreases slightly to 0.260 N with DRS, as smoother airflow reduces skin friction.
- Model 03: The updated splitter maintains a similar friction force of 0.262 N, indicating that while the airflow is optimized, the car's surface interaction remains consistent.

6. Drag Coefficient

The drag coefficient (Cd) is a dimensionless number representing the car's aerodynamic drag.

- **Model 01**: The base model has a Cd of **0.826**, typical for F1 cars that balance drag reduction and downforce needs.
- **Model 02**: DRS reduces the Cd to **0.791**, reflecting decreased aerodynamic resistance and improved straight-line speed.
- **Model 03**: The updated splitter reduces the Cd further to **0.773**, highlighting the cumulative effect of DRS and splitter optimization.

Although these key parameters of static pressure, total pressure, velocity, force, friction force and drag coefficient changed due to different F1 car models, there were a few unfortunate simulation errors during the CFD Flow Simulation.

Since the F1 car model is mated to the platform by using the tyres, those tyres were not rotating at the velocity of 30 m/s and made the below simulation error in which the velocity distribution near the tyre was not clearly simulated.

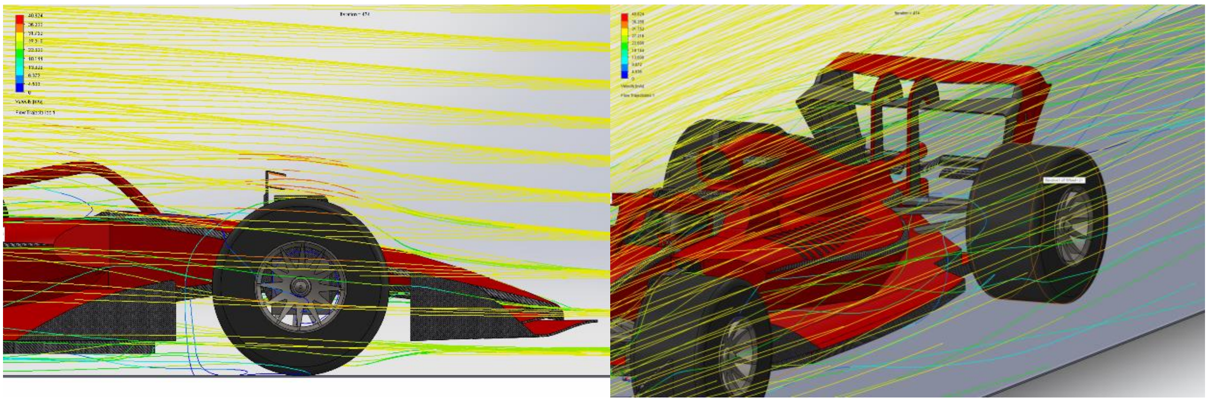


Figure 16 Figure 16 CFD Simulation error inside and the outside the front and the rear tyres of the F1 model car

CONCLUSION

The analysis of the three F1 car models highlights the impact of aerodynamic modifications on key performance parameters, even though the model is not accurate in scale and design, the drag coefficient may not fully reflect real-world conditions. The results demonstrate significant changes in static pressure, total pressure, force, and friction force while velocity changes were less pronounced.

- **Model 01 (Base Model)** provides a baseline representation of an F1 car, showcasing balanced aerodynamic characteristics essential for typical track conditions.
- **Model 02 (Base + DRS)** effectively reduces drag and force through the activation of the Drag Reduction System, allowing for smoother airflow and improved straight-line efficiency.
- **Model 03 (Base + DRS + Updated Splitter)** achieves the most aerodynamic efficiency, optimizing airflow with reduced drag and improved pressure distribution through the combined effects of DRS and splitter updates.

While this simulation model has limitations in scale and exact drag coefficient representation, it aligns with the unique aerodynamic characteristics of F1 cars, where the drag coefficient is intentionally kept higher between 0.7–1.2 than regular vehicles to balance drag reduction with downforce for stability and control to effectively demonstrates the relative impact of aerodynamic design changes.

These insights reinforce the importance of fine-tuning aerodynamic elements to enhance performance in F1 cars, with a clear emphasis on reducing drag while maintaining stability and control.

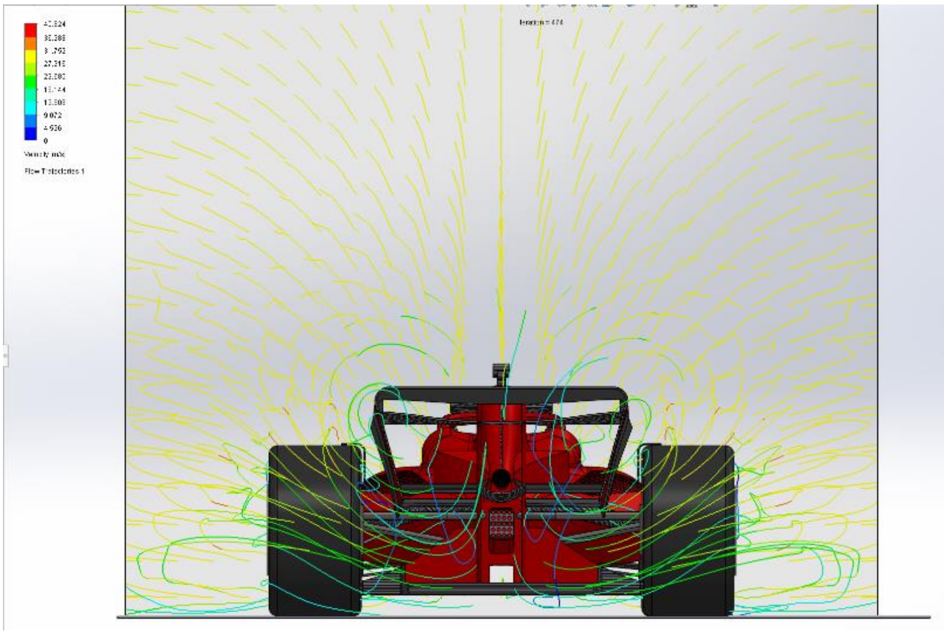


Figure 18 Velocity distribution of the F1 car model with the active Drag Reduction System (DRS) and Updated front Splitter at the back

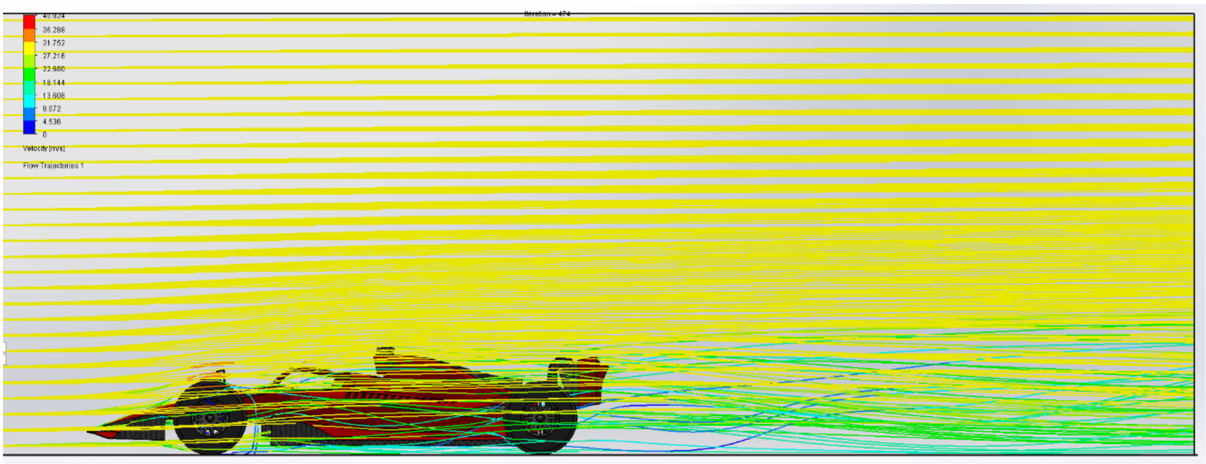


Figure 17 Velocity distribution of the F1 car model with the active Drag Reduction System (DRS) from side view

PRECAUTIONS AND SAFETY MEASURES

These precautions and safety measures help ensure the project proceeds efficiently, safely and without technical or human errors.

- Ensure Accurate Dimensions: Double check all dimensions and parameters in the CAD model to avoid design errors. But during the making of the F1 model some of the parts could not made correctly and precisely.

Figure 20 Driverless F1 car model due to lack of skill to put the F1 driver inside the car

Figure 19 Impurities of the front splitter of the F1 car due to the lack of Surfacing skill in SolidWorks CAD Modeling skills

- Save Work Regularly: Save the CAD model frequently to prevent data loss due to software crashes or power failures.
- Avoid Overloading the System: Use optimized model details to prevent SolidWorks from lagging or crashing.
- Follow Proper Simulation Settings: Set accurate boundary conditions, such as velocity and temperature, in the CFD analysis to ensure reliable results.

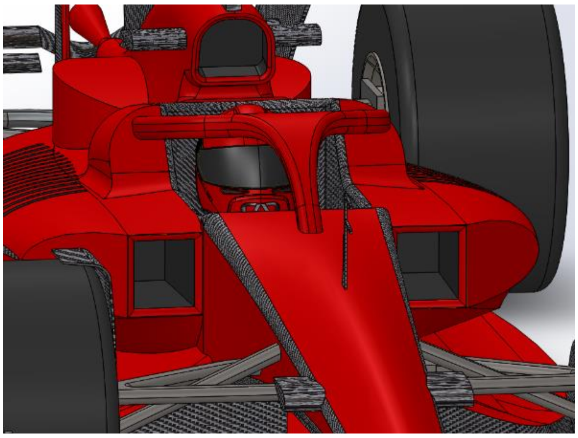


Figure 21 Fake S duct at the side of the open cockpit of the F1 car model due to lack of SolidWorks CAD Modeling skills

- Verify Assumptions: Confirm that assumptions like ideal gas behaviour and steady-state flow are valid for the analysis.
- Maintain System Requirements: Use a computer with the recommended hardware specifications to run SolidWorks and CFD simulations smoothly.
- Wear Eye Protection: Protect your eyes from screen glare by using anti-glare glasses if working for extended periods.
- Practice Ergonomics: Sit in a comfortable position with proper posture to avoid strain during long design sessions.
- Handle Software Licenses Carefully: Use authorized software licenses to avoid legal or functionality issues.
- Use Correct Units: Ensure all measurements and values are in consistent units to prevent calculation errors.

REFERENCES

- *What is F1?* (no date). https://www.formula1.com/en/page/what-is-f1.
- PlanetF1 (2022a) 'What is F1's 'DRS' and how does it work?,' *PlanetF1*, 15 July. https://www.planetf1.com/features/what-is-f1s-drs-and-how-does-it-work.
- *F1 3D airflow - the new era is upon us! (2/2) - Formula 1 Videos* (no date). https://www.motorsport.com/f1/video/f1-3d-airflow-the-new-era-is-upon-us-2-2-94936/154176/.
- Ahmmed, A. (2023) *F1 Aerodynamics for Beginners*. https://www.linkedin.com/pulse/f1-aerodynamics-beginners-alvi-ahmmed-cswp/.
- Admin (2024) *Jobs In F1: How To Become A Formula 1 Aerodynamicist*. https://motorsportengineer.net/jobs-in-f1-how-to-become-a-formula-1-aerodynamicist/.
- Massobrio, A. (2024) *F1 Aerodynamics — Shape Predictions & Optimization With NCS*. https://www.neuralconcept.com/post/formula-1-multiple-connected-components-and-long-range-aerodynamic-correlations.