

# CFD Analysis of an F1-Style Vehicle Using OpenFOAM

**Author:** Sulaiman Mahmood

---

## Overview

This document summarizes an independent Computational Fluid Dynamics (CFD) project focused on analyzing the aerodynamics of an F1-style open-wheel vehicle using OpenFOAM. The project was carried out to develop and demonstrate practical CFD capability, with particular attention paid to solver setup, mesh quality, and interpretation of flow physics rather than purely visual outcomes.

The simulation is based on the numerical solution of the Navier-Stokes equations under turbulent flow conditions and reflects workflows commonly used in motorsport and automotive aerodynamics.

---

## Context and Motivation

Aerodynamic performance in Formula 1 is dominated by complex, three-dimensional, turbulent flow structures. Accurately capturing these effects requires:

- careful meshing of complex geometries,
- appropriate turbulence modelling,
- numerical stability when solving the Navier–Stokes equations,
- and engineering judgement when interpreting results.

This project was undertaken as a self-directed exercise to explore these challenges using an open-source CFD framework.

---

## Methodology

### Geometry and Setup

An F1-style vehicle geometry was imported as an STL and scaled correctly to SI units. The geometry included major aerodynamic features such as the front wing, underfloor, diffuser, and rear wing.

### Meshing

The computational mesh was generated using `snappyHexMesh`, with local refinement applied to regions of expected high gradients, including:

- the front wing and nose region,
- the underfloor and diffuser,
- the near-wake region downstream of the car.

Boundary layer refinement was applied to ensure adequate near-wall resolution, targeting  $y^+$  values below 5 where feasible.

### Solver and Physics

The incompressible **Reynolds-Averaged Navier–Stokes (RANS)** equations were solved using `simpleFoam`. Turbulence was modelled using the **k- $\omega$  SST** model due to its robustness in separated flow regions and near-wall behaviour.

---

## Results and Observations

The simulation converged to a stable solution with physically consistent flow features. Key observations included:

- a low-pressure region beneath the vehicle contributing to downforce,
- accelerated flow through the diffuser,
- coherent wake structures forming behind the rear wing and diffuser.

The predicted drag force was on the order of **94 N**, which is realistic for the scale and assumptions of the model. Flow fields were analysed in **ParaView**, focusing on velocity magnitude, pressure distribution, and wake behaviour.

---

## Engineering Judgement and Challenges

Several practical challenges were encountered, including:

- initial instability due to mesh quality near sharp edges,
- correcting geometry scaling and surface normals,
- balancing mesh resolution against computational cost.

These issues were addressed iteratively, guided by expected physical behaviour rather than blind numerical convergence.

---

## Significance

This project demonstrates an applied understanding of:

- numerical solution of the Navier–Stokes equations,
- turbulence modelling in external aerodynamics,
- mesh design and boundary layer treatment,
- Interpretation of CFD results in an engineering context.

The work reflects independent problem-solving and mirrors real-world CFD workflows rather than purely academic exercises.

---

## **Availability**

The full case setup, configuration files, and documentation are publicly available on GitHub.