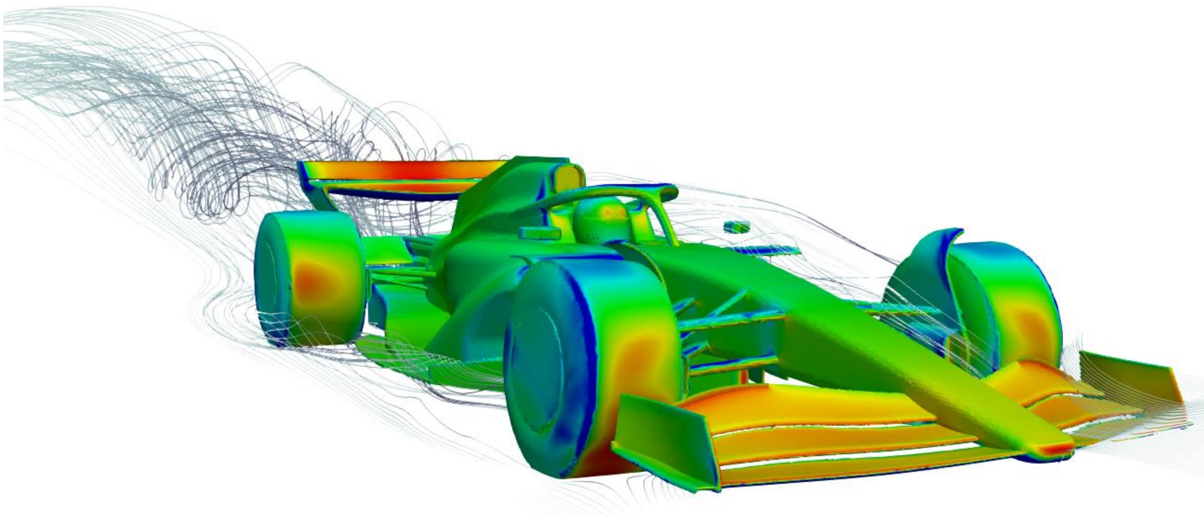
ISIEINDIA Private Limited
E – 210, 2nd Floor, Sector – 63, Noida – 201301
www.isieindia.com | 0120 – 4538457



Program: EV Design, Simulation & Component Selection
Project: CFD Analysis of Different 4 Wheelers

Contents

Background:

A spoiler is a device commonly used in cars to reduce lift, increase downforce, and improve stability at high speeds. When a car is moving, the air flows over the surface of the car and creates an area of low pressure above the car. This low-pressure zone creates lift, which reduces the amount of traction between the tires and the road surface.

A spoiler is designed to disrupt the airflow over the car and create a turbulence behind it. This turbulence increases the air pressure above the rear of the car, which reduces the low-pressure zone and, thus, decreases lift force. Additionally, a spoiler can redirect airflow to create downforce, which increases the amount of grip and stability on the road surface, allowing the car to corner faster and more safely at high speeds.

Spoilers are especially useful in high-performance cars because they typically generate more lift due to their aerodynamic design. By reducing lift and increasing downforce, spoilers can improve the car's handling and stability, which can help the driver maintain control of the vehicle, particularly in high-speed situations or during sudden maneuvers.

Objectives:

Computational Fluid Dynamics (CFD) is a method used to simulate the behavior of fluid flows, such as air flow over a car. CFD analysis of a car's aerodynamics can be used to optimize the design of the car's body and components, such as the shape of the body, wing mirrors, and air intakes.

CFD project on a car involve understanding the objectives of the project, which includes improving the car's performance, reducing drag, or enhancing fuel efficiency. The project involves use of software tools ANSYS Fluent to simulate the flow of air over the car, considering factors such as the car's shape, speed, and orientation. The project begins with the creation of a CAD model of the car, which would be used as the basis for the CFD simulations. The model would need to be carefully prepared, with a high level of detail and accuracy, to ensure that the simulations are as realistic and accurate as possible.

CFD project on a car would require a deep understanding of fluid mechanics, as well as expertise in software tools and simulation techniques. It would also require a thorough understanding of the specific objectives of the project, and the ability to translate those objectives into appropriate simulation parameters and design changes.

The project begins with the creation of a CAD model of the car, which would be used as the basis for the CFD simulations. The model would need to be carefully prepared, with a high level of detail and accuracy, to ensure that the simulations are as realistic and accurate as possible.

CFD project on a car would require a deep understanding of fluid mechanics, as well as expertise in software tools and simulation techniques. It would also require a thorough understanding of the specific objectives of the project, and the ability to translate those objectives into appropriate simulation parameters and design changes.

Methodology:

The project will be divided into the following phases:

1. Define the Objectives
   - The objective is reducing the lift force acting on a vehicle by adding spoiler in the car.

2. Develop the CAD Model
   - Develop a car model using CAD modeling tools.
   - Make a projection of the car in front plane to find the projected area.
   - Import the CAD model in Ansys Space-Claim to prepare the geometry.
   - Use Volume Extract command to create a wind tunnel and find the effected volume of the fluid.
   - Create required name selections.

Note- Computational domain can be divided into 2 symmetrical domains to save computational time

3. Mesh Generation
   - Use Fluent with Meshing to generate volumetric mesh of the system.

4. Define Boundary Conditions

- Velocity – 60 km/h
- Environment pressure
- Define reference value (Projected area, velocity, etc)

5. Run the Simulation
   - Simulate it in Ansys Fluent to find the following
     - Drag Coefficient
     - Drag force
     - Lift Coefficient
     - Lift force

6. Analyze Results
   - Analyze the results to find the required change in the design.

7. Optimize the Design
   - To reduce the lift force and increase the vehicle stability, design a spoiler for the car and repeat the above steps.

Expected Outcomes:

The following outcomes are expected from this project:
A project on simulation and optimization of a car using a spoiler has the potential to yield several outcomes, including:

- Improved understanding of the aerodynamic principles behind car spoilers: By simulating the airflow around a car with and without a spoiler, researchers can gain insight into how spoilers affect the car's drag and lift. This understanding can inform future designs of car spoilers.
- Identification of optimal spoiler design parameters: Through simulation and optimization, researchers can identify the best spoiler design parameters to achieve specific goals, such as minimizing drag or maximizing downforce.
- Validation of simulation and optimization techniques: This project will also help validate simulation and optimization techniques for car design, which can be applied to other areas of vehicle design and engineering.

Conclusion:

This project aims towards optimization of vehicle dynamics with the help of spoiler and give a better understanding to students with Computational Fluid Dynamics