Aerodynamic Optimization of the Formula Car Using Ansys Fluent

Prerequisite

Software required:

Ansys Fluent – Request for Partnership

Learning Content: -

Basic: -

Introduction to Ansys Fluent | Ansys Innovation Courses

Geometry Prep for CFD using Ansys Discovery | Ansys Courses

Mesh Generation using Ansys Fluent Meshing | Ansys Courses

Physics Setup in Ansys Fluent | Ansys Innovation Courses

Customization in Ansys Fluent | Ansys Innovation Courses

Solution Setup in Ansys Fluent | Ansys Courses

Post Processing in Ansys Fluent | Ansys Innovation Courses

Best Practice Guidelines for CFD Simulations | Ansys Courses

Core: -

Aerodynamics of an FSAE Car | Ansys Innovation Courses

Problem Statement

In the highly competitive world of F1, the challenge of designing an aerodynamically efficient car is a crucial factor when it comes down to winning races. Teams must work to design a car that minimizes the force of drag, thereby allowing the car to have a higher top speed down the track.

The objective of this study is to analyse the aerodynamic performance of a Formula 1 car using Computational Fluid Dynamics (CFD). The focus will be on evaluating airflow behaviour, pressure distribution, drag and wake characteristics under different geometry configurations.

Design Challenge

- To evaluate key aerodynamic parameters below,
  - Drag coefficient (Cd)
  - Pressure and velocity distribution
  - Flow separation zones and wake effects

- To optimize aerodynamic components like front/rear wing profiles, front nose etc..

Geometry (CAD model)

Make sure to use your own vehicle geometry instead of a generic one

Reference generic geometry: To be used as reference only

Case 1 – Base geometry

Document below

1. **Grid Generation**
   - Minimum Size (Surface Mesh): -
   - Maximum Size (Surface Mesh: -
   - Type of Volume Mesh: -
   - Grid independence study

2. **Solver**
   - Solver type: -
   - Turbulence Model: -
   - Number of iterations: -

3. **Post Processing**
   - Velocity Contour at the center plane of the car
   - Pressure Contour at the center plane of the car
   - Calculate drag coefficient and drag force on the car

Case 2 – Modified geometry

Objective: Apply the engineering insights to refine your geometry, leading to a reduction in drag coefficient and drag force.

Geometry

Include the images of the modified geometry (CAD) – zoom into the specific location where you have made changes

Justification:

Provide a detailed explanation of the reasoning and justification behind the proposed change (not less than 100 words)

Document below

1. **Grid Generation**

- Minimum Size (Surface Mesh): -
- Maximum Size (Surface Mesh: -
- Type of Volume Mesh: -

2. **Solver**
   - Solver type: -
   - Turbulence Model: -
4. **Solver**
   - Solver type: -
   - Turbulence Model: -
   - Number of iterations: -

5. **Post Processing**
   - Velocity Contour at the center plane of the car
   - Pressure Contour at the center plane of the car
   - Calculate the new drag coefficient and drag force on the car
   - Percentage of Drag reduction in the car

Tips: -

- Modify the Frontal area (Changes in the design of the nose and wing).
- Look for flow visualization to assist you in changing the geometry at sharp edges.