AERODYNAMIC ANALYSIS OF AN FSAE CAR -using Ansys Fluent

Geometry Preparation for CFD Analysis in FSAE Car Design

Computational Fluid Dynamics (CFD) analysis plays a crucial role in the design and optimization of Formula SAE (FSAE) cars. A well-prepared geometry is essential for accurate simulations and efficient computational performance. The process of geometry preparation involves two key aspects: **simplification** and **repair**, both of which ensure that the model is suitable for meshing and further analysis.

1. Geometry Simplification

FSAE car geometries often contain intricate details that may not significantly impact the overall aerodynamic performance but can drastically increase computational cost. To optimize the simulation process, the following simplifications are commonly applied:

- **Removing Small Features:** Eliminating unnecessary fillets, bolt holes, and other minor details that do not affect aerodynamic behavior.

- **Merging and Combining Surfaces:** Simplifying complex surface transitions to create a cleaner, more structured model.

- **Streamlining Curves:** Reducing unnecessary complexity in curved surfaces while maintaining their aerodynamic relevance.

These simplifications help reduce mesh complexity, improve solver stability, and lower computational time while preserving the accuracy of aerodynamic predictions.

2. Geometry Repair

To ensure the geometry is topologically valid and mesh-ready, it is crucial to identify and fix issues that may interfere with mesh generation. The key aspects of geometry repair include:

- **Closing Gaps:** Ensuring that all surfaces are properly connected to create a continuous, watertight model.

- **Fixing Overlaps and Intersections:** Removing self-intersecting surfaces and overlapping elements that could lead to errors in meshing.

- **Addressing Non-Manifold Edges:** Eliminating singularities where more than two surfaces meet at a single edge.

- **Smoothing and Healing:** Refining surface transitions to avoid abrupt changes that may cause meshing difficulties.

By applying these repair techniques, we can ensure that the model is robust and free from defects that might affect the quality of the mesh and the accuracy of CFD results.

3. Practical Implementation Using Ansys Discovery

Through this three-part lesson, we will explore the **simplification** and **repair** of geometry using **Ansys Discovery**. This software provides powerful tools to refine, heal, and prepare CAD models for CFD simulations, ensuring a seamless transition from geometry creation to analysis.

By mastering these techniques, we can enhance the accuracy of CFD simulations, improve computational efficiency, and optimize the aerodynamic performance of FSAE cars.
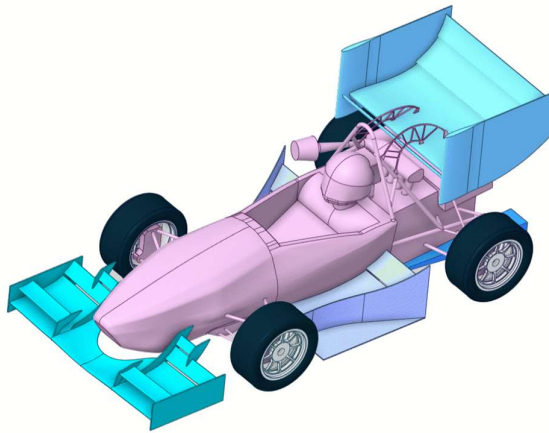
Meshing for CFD Simulation of FSAE Cars:

Computational Fluid Dynamics (CFD) simulations play a vital role in the aerodynamic design and optimization of Formula SAE (FSAE) cars. To ensure accurate simulation results, it is essential to generate a high-quality, error-free mesh. Meshing is the process of dividing the computational domain into small sub-volumes or cells, where the governing equations of fluid dynamics are solved to predict airflow behavior around the vehicle. Given the complexity of FSAE car geometries, special attention must be given to refining the mesh in critical regions to enhance accuracy.

Meshing Workflow Using Ansys Fluent Meshing

In this study, we utilize the **Ansys Fluent Meshing Watertight Geometry Workflow** to create a simulation-ready mesh for the FSAE car. The structured approach ensures an optimized mesh that accurately captures flow physics while maintaining computational efficiency. The key steps in this process include:

1. **Importing CAD Geometry**

   o   Loading and preparing the car geometry for meshing.

   o   Checking for any inconsistencies or defects that may impact mesh generation.

2. **Creating Local Refinement Regions**

   o Identifying high-gradient regions such as wing surfaces, suspension components, and wake zones.

   o Refining the mesh in these regions to accurately capture flow features like separation, vortices, and turbulence.

3. **Applying Local Sizing for Complex Curvatures**

   o Ensuring that sharp edges, curved surfaces, and intricate geometrical details are adequately resolved.

   o Controlling element size variations to maintain mesh quality and prevent excessive skewness.

4. **Generating Boundary Layers**

   o Adding inflation layers near solid surfaces to resolve near-wall flow behavior accurately.

   o Ensuring proper y+ values for turbulence modeling, which is crucial for aerodynamic analysis.

5. **Creating and Improving Volume Mesh**

   o Generating a high-quality volume mesh with appropriate element types (tetrahedral, hexahedral, polyhedral).

   o Refining mesh density in critical aerodynamic regions while keeping computational cost manageable.

6. **Mesh Quality Checks and Error Resolution**

   o Verifying mesh quality parameters such as skewness, aspect ratio, and orthogonality.

   o Identifying and resolving common meshing warnings or errors to ensure solver stability and accuracy.

7. **Understanding General Meshing Best Practices**

   o Discussing key meshing decisions and their impact on simulation accuracy and computational efficiency.

   o Optimizing the mesh to balance accuracy and solver performance.

This structured meshing approach ensures that the CFD simulation of the FSAE car provides reliable aerodynamic insights, leading to improved vehicle performance. By following best

practices in meshing, we can achieve a well-refined computational domain that accurately represents real-world aerodynamic conditions.
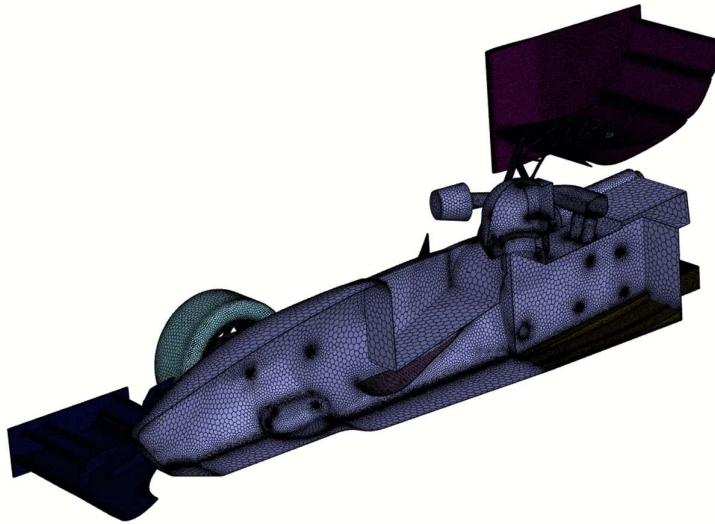
CFD Simulation Setup for Aerodynamic Analysis of an FSAE Car:

To obtain accurate and reliable results in a Computational Fluid Dynamics (CFD) simulation, careful attention must be given to the solver setup. The simulation setup involves defining numerical methods, solution controls, boundary conditions, and other critical parameters that influence the accuracy of the results. This study focuses on configuring a CFD simulation for the aerodynamic analysis of an **FSAE car** using **Ansys Fluent**.

Key Aspects of Simulation Setup

1. **Defining Boundary Conditions**

   o   Establishing appropriate inlet and outlet conditions for airflow over the FSAE car at **35 mph**.

   o   Setting wall boundary conditions for the vehicle surface to capture aerodynamic interactions accurately.

   o   Defining symmetry planes to reduce computational effort while maintaining realistic flow behavior.

2. **Modeling rotating Components Using MRF Zones**

- o Implementing **Multiple Reference Frame (MRF) zones** to accurately simulate airflow around **wheel rims**.

- o Capturing the rotational effects of the wheels to improve the accuracy of wake and turbulence predictions.

3. **Selecting the Turbulence Model**

- o Utilizing the **GEKO (Generalized k-omega) turbulence model**, which provides a robust and flexible approach to solving **complex flow fields** around the FSAE car.

- o Understanding turbulence model settings and calibrating them to match experimental or real-world conditions.

4. **Solution Controls and Numerical Methods**

- o Setting appropriate discretization schemes for **pressure-velocity coupling** and **turbulence equations** to enhance numerical stability and accuracy.

- o Configuring relaxation factors and solution methods to optimize convergence behavior.

5. **Monitoring Convergence and Solution Accuracy**

- o Tracking **residuals** of flow variables to ensure numerical stability and solution convergence.

- o Setting up **report definitions** to monitor aerodynamic parameters such as **drag force, lift force, and pressure distribution** during the simulation.

- o Ensuring a sufficient number of iterations for the solution to reach a steady-state or transient convergence.

By following this structured simulation setup approach, the CFD analysis of the FSAE car will provide **accurate aerodynamic insights**, leading to optimized vehicle performance. The integration of **boundary conditions, MRF zones, turbulence modeling, and convergence monitoring** ensures a well-validated simulation that closely represents real-world aerodynamic behavior.

Post-Processing in CFD Simulations for FSAE Car Aerodynamics:

Post-processing is a critical phase in **Computational Fluid Dynamics (CFD) simulations**, enabling the extraction and analysis of key flow characteristics from simulation data. It plays a crucial role in evaluating whether the mesh accurately captures all relevant flow features and in understanding the aerodynamic behavior of the **FSAE car**. Through post-processing, essential insights into **pressure distribution, velocity fields, turbulence structures, and aerodynamic forces** can be obtained, aiding in performance optimization.

Quantitative and Qualitative Analysis

This study involves a **two-part post-processing approach** using Ansys Fluent to analyze the simulation results effectively:

1. **Quantitative Analysis**

   o Extracting **aerodynamic forces** such as **drag and lift coefficients** to assess vehicle performance.

   o Analyzing **pressure distribution** on the car's surface to understand high and low-pressure regions affecting stability.

   o Evaluating **velocity profiles** at key locations to examine airflow characteristics around the body and wheels.

   o Investigating **turbulence parameters** to identify flow separation, wake regions, and vortex formations.

2. **Qualitative Analysis**

   o Visualizing **streamlines** to track airflow behavior around the car and detect recirculation zones.

   o Generating **contour plots** of velocity and pressure fields to identify critical aerodynamic features.

   o Creating **vector plots** to observe localized flow directions and turbulence interactions.

   o Using **iso-surfaces and vortex structures** (e.g., Q-criterion) to detect and analyze aerodynamic vortices affecting car stability.

By employing both **quantitative and qualitative post-processing techniques**, this study provides a **comprehensive evaluation of the aerodynamic performance** of the FSAE car. The insights gained will guide design modifications aimed at enhancing **efficiency, stability, and overall vehicle performance**.

pathlines-2
Particle ID

4.11e+04
3.70e+04
3.29e+04
2.87e+04
2.46e+04
2.05e+04
1.64e+04
1.23e+04
8.21e+03
4.11e+03
0.00e+00