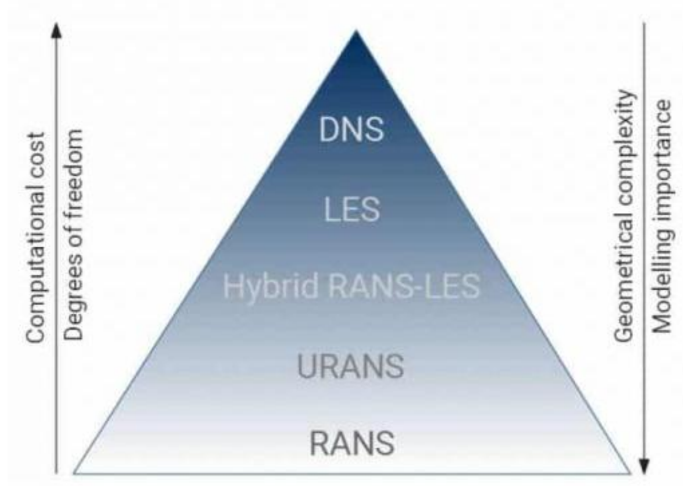
LES for Aerodynamic Analysis of a Formula SAE Car Incorporating Moving Ground and Rotating Wheels

Ting-Jung Kao, ChungGang Li*
High Performance Parallel Computing CFD Lab.
Department of Mechanical Engineering, National Cheng Kung University

Introduction

Vehicle aerodynamics involves complex three-dimensional, unsteady flow phenomena, including separation, vortex formation, and wake dynamics. These flow characteristics directly affect drag and lift forces, which in turn influence fuel efficiency and energy consumption, as well as vehicle stability and handling performance during high-speed cornering and braking. While wind tunnel and track testing can provide valuable experimental data, they are often limited by high costs, restricted measurement points, and the difficulty of controlling boundary conditions such as moving ground and rotating wheels .

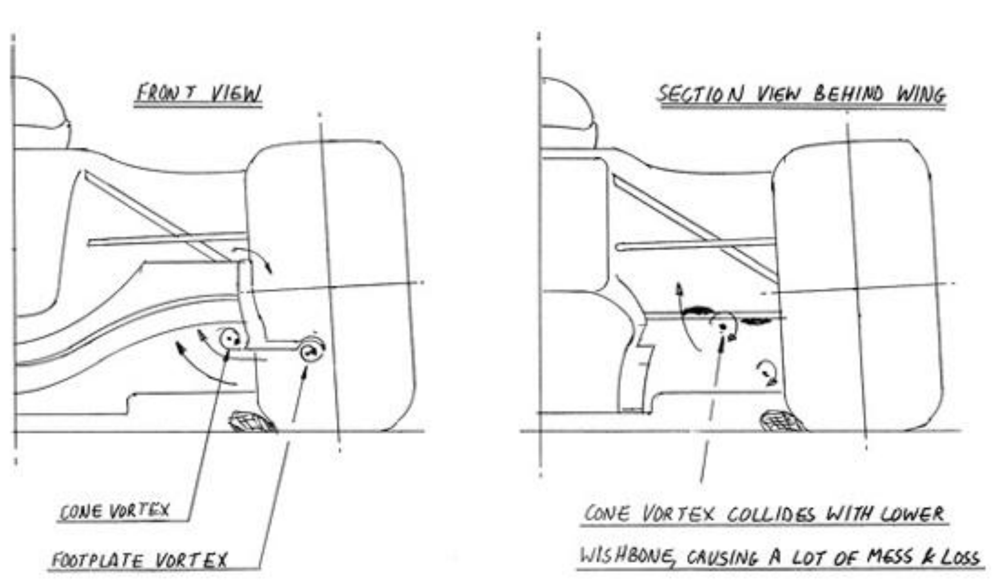
Computational Fluid Dynamics (CFD) provides a cost-effective and controllable approach to capture wake structures, pressure distributions, and turbulent energy evolution with high spatial and temporal resolution. In particular, Wall-Modeled Large Eddy Simulation (WMLES) enables realistic representation of large-scale turbulent structures and near-wall flows, allowing accurate evaluation of the impact of different geometries and aerodynamic configurations on drag and downforce.



Turbulence models in CFD from RANS to DNS

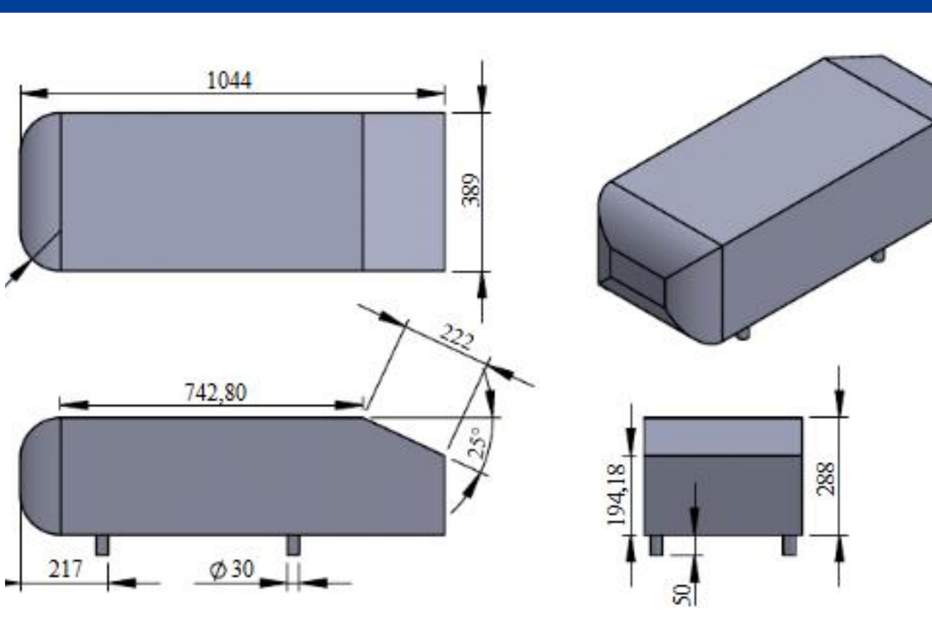
Abstract

This study first employs the classical Ahmed body geometry to perform WMLES, validating the reliability and accuracy of the adopted numerical methodology and meshing strategy. Subsequently, the CK06 vehicle model is used with WMLES to investigate the effects of a moving ground and rotating wheels on the surrounding flow field, including wake vortex structures, pressure distribution, and airflow attachment behavior. Finally, aerodynamic parameters such as drag coefficient, lift coefficient, and other relevant metrics are analyzed to compare the performance of the vehicle's aerodynamic package under different boundary conditions, providing valuable guidance for future race car design.



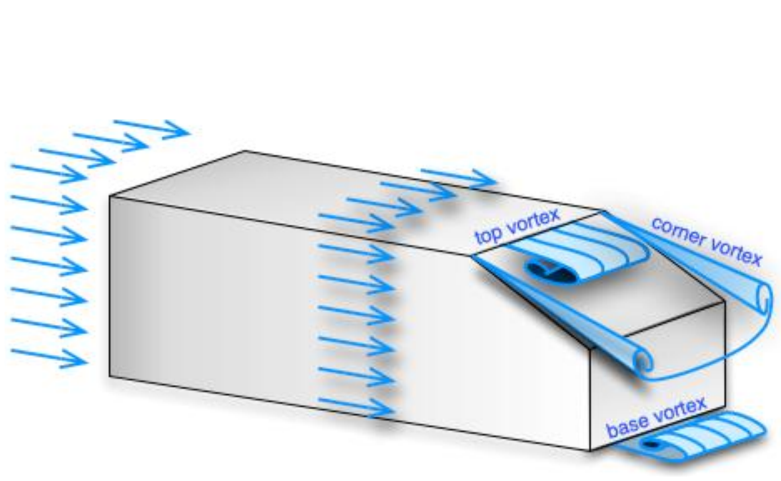
Control vortex structures and airflow by aerodynamic kits

Ahmed Body

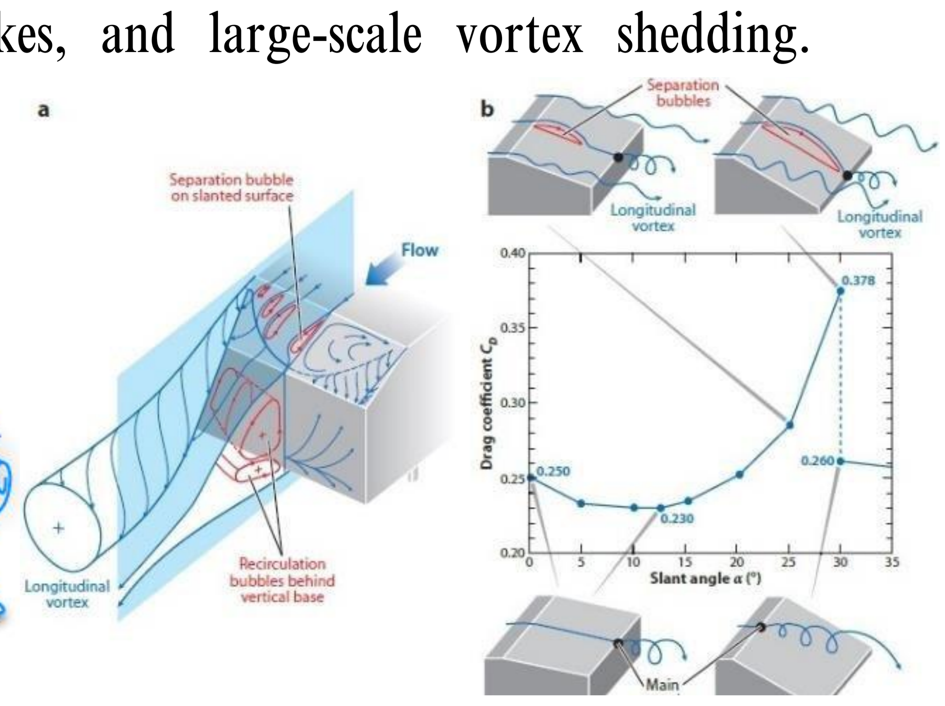


Ahmed Body with 25° Slant Angle

The Ahmed body, introduced by S.R. Ahmed in 1984, is a simplified vehicle model for fundamental automotive aerodynamics studies. Its bluff body with a slanted rear allows controlled variation of slant angle to examine flow separation and wake structures. Despite its simplicity, it reproduces key real-vehicle flow features such as boundary layer separation, recirculating wakes, and large-scale vortex shedding.



the Ahmed Body slant angle on flow separation and vortex structure



(a) Time-averaged 3D flow structures of the Ahmed body in the wake and (b) the variation in Cd for different slant angles

Physical Models

The aerodynamic behavior of the Ahmed body is governed by the compressible Navier-Stokes equations, solved using an LES implicit subgrid-scale model.

$$\frac{\partial \rho}{\partial t} + \frac{\partial u_j}{\partial x_j} = 0 \quad \textbf{(Continuity)}$$

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial \rho u_i u_j}{\partial x_j} = -\frac{\partial p}{\partial x_j}\delta_{ij} + \frac{\partial \tau_{ij}}{\partial x_j} \quad \textbf{(Momentum)}$$

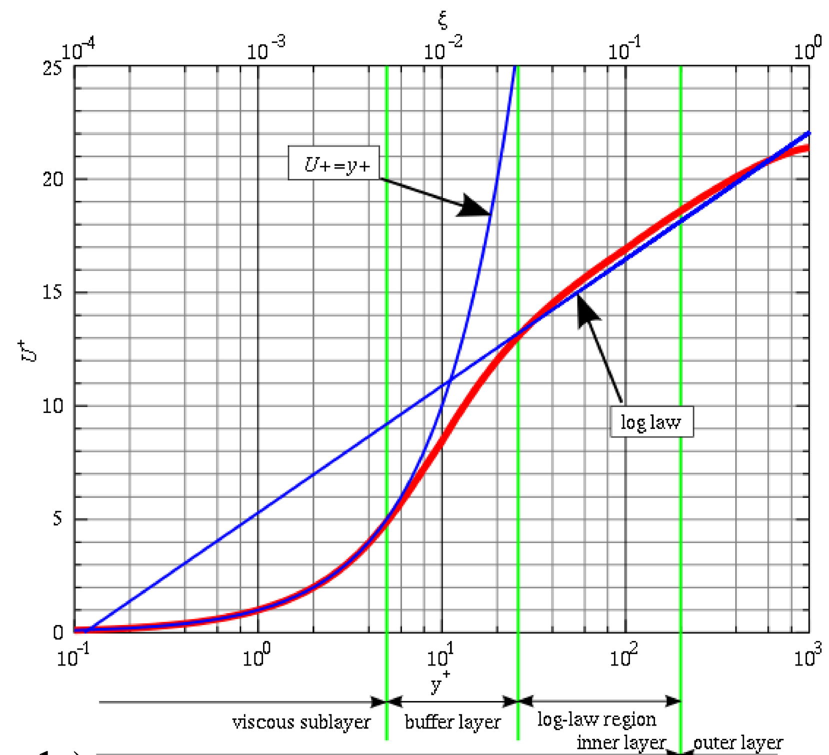
$$\frac{\partial \rho e_0}{\partial t} + \frac{\partial \rho u_j e_0}{\partial x_j} = -\frac{\partial u_j p}{\partial x_j} - \frac{\partial q_j}{\partial x_j} + \frac{\partial u_j \tau_{ij}}{\partial x_j} \quad \textbf{(Energy)}$$

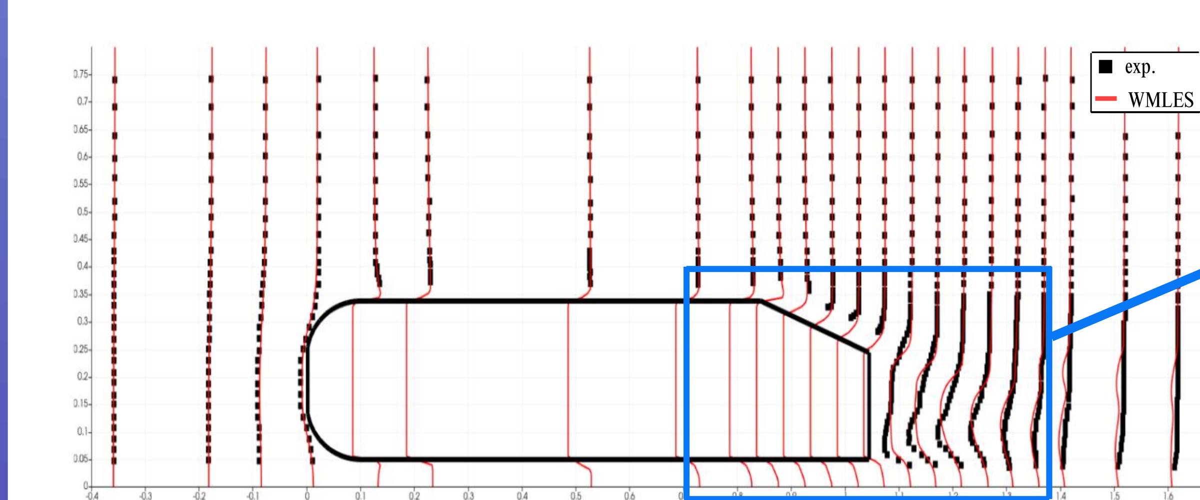
Results

Ahmed Body

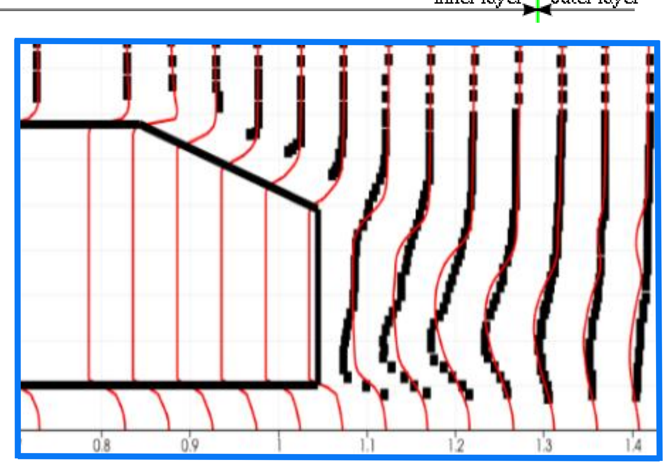
Set Up:

- Wind tunnel size: 24m×12m×6m
- Finest grid: 1.95 mm, 30 < y+ < 300
- Cubes: 22492
- U = 40 m/s, M = 0.1175468
- P = 101,300 Pa
- ρ = 1.184 kg/m^3
- μ = 1.86·10^(-6) Pa·s
- Re = UD/ν ≈ 768,000 (D is based on height of the body)

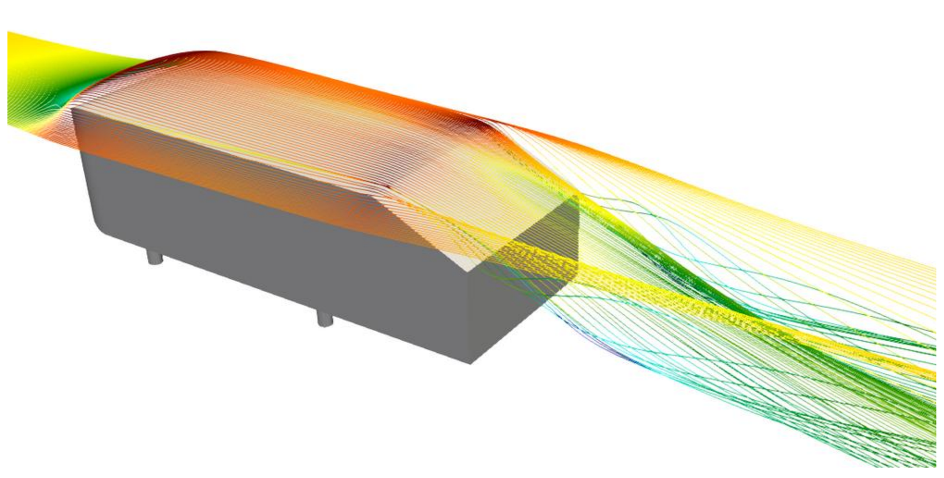




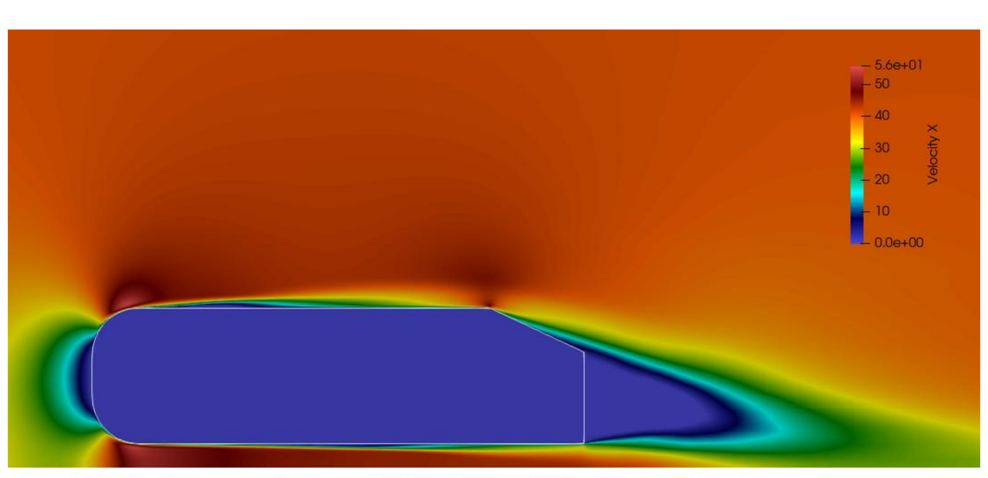
Profiles of the streamwise velocity component along the rear part of the Ahmed body for the slant angles ϕ = 25°



Detail streamwise velocity component at rear of the Ahmed body



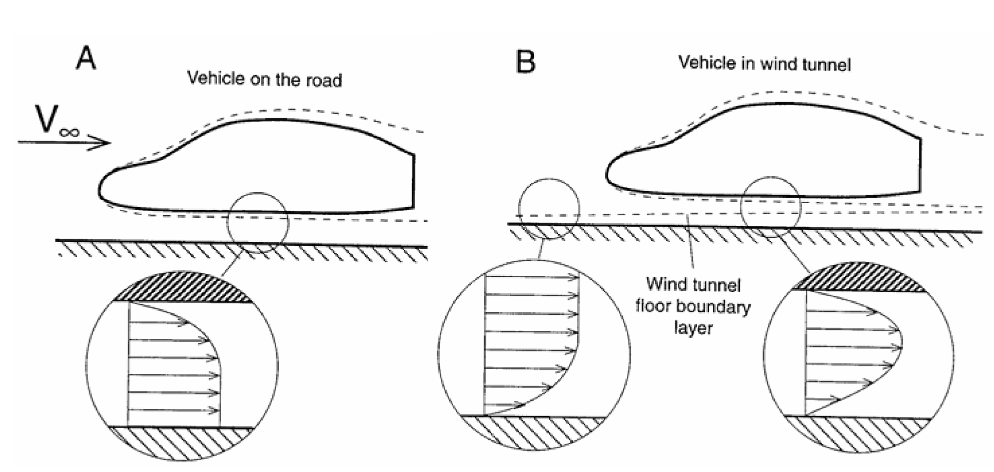
Streamline around the upper Ahmed body



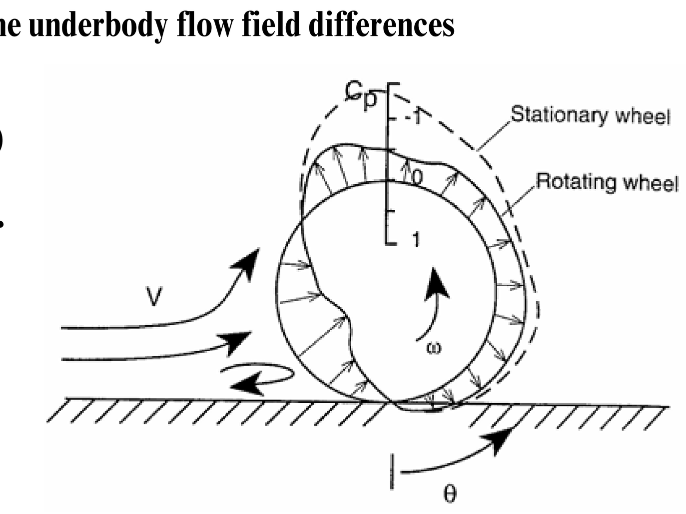
velocity contour at the symmetry plane

CK06

After validating the reliability and accuracy of the adopted numerical methodology and mesh -ing strategy using the Ahmed Body, the CK06 CAD model from NCKU Formula Student will be used to analyze the flow field of a Formula SAE race car with a moving ground and rotating tires.

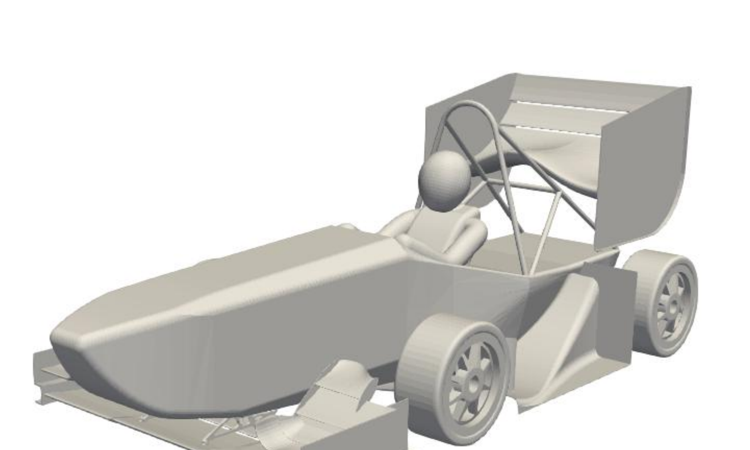


The impact of whether the ground moves relative to the vehicle on the underbody flow field differences



Pressure distribution around a rotating tire and a stationary tire

CK05 from NCKU Formula Student



CAD Model of CK06 from NCKU Formula Student