Analysis of Airflow Patterns and Drag Force over a Streamlined Solar Car

Lisa Tang

May 12th, 2020

2.29 Numerical Fluid Mechanics

Power Efficiency Important for Solar Vehicles

MIT solar car receives 700 W power from panels

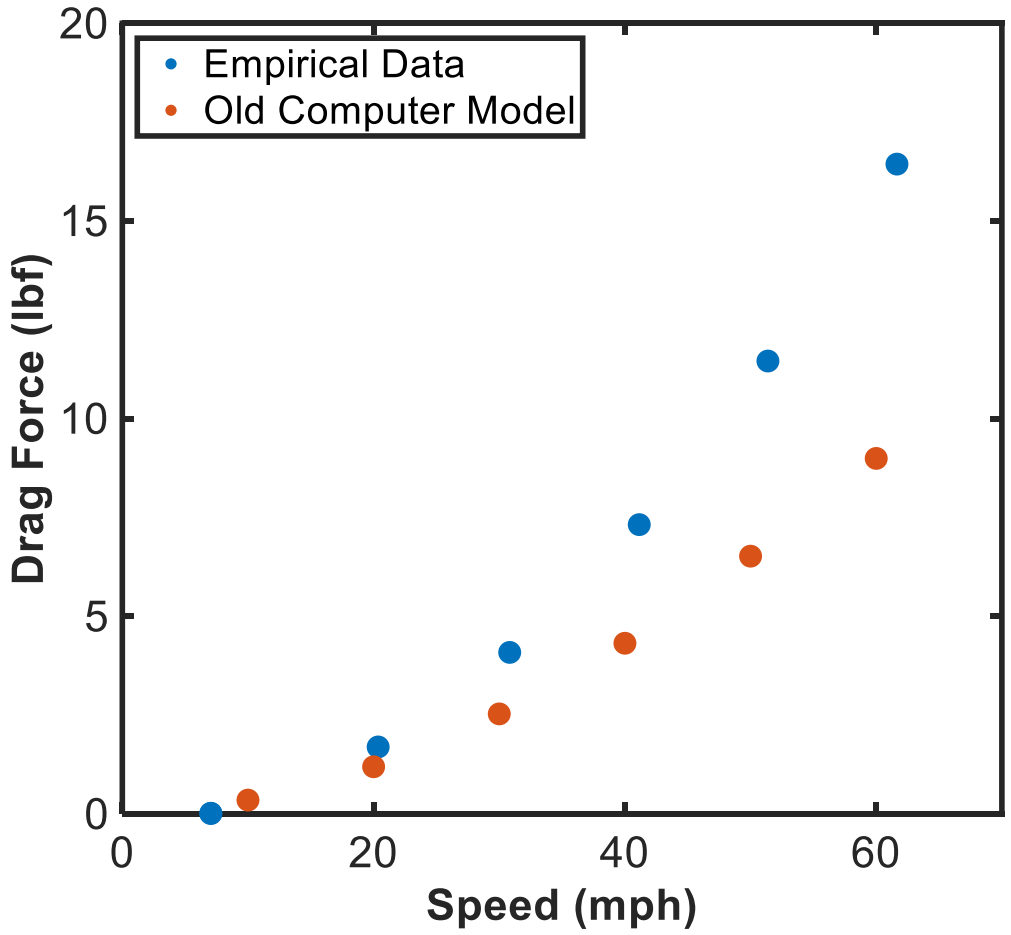
Streamlined airfoil shape reduces air drag

Wind Tunnel Testing to Assess Drag Force

Testing performed at Ford wind tunnel facilities

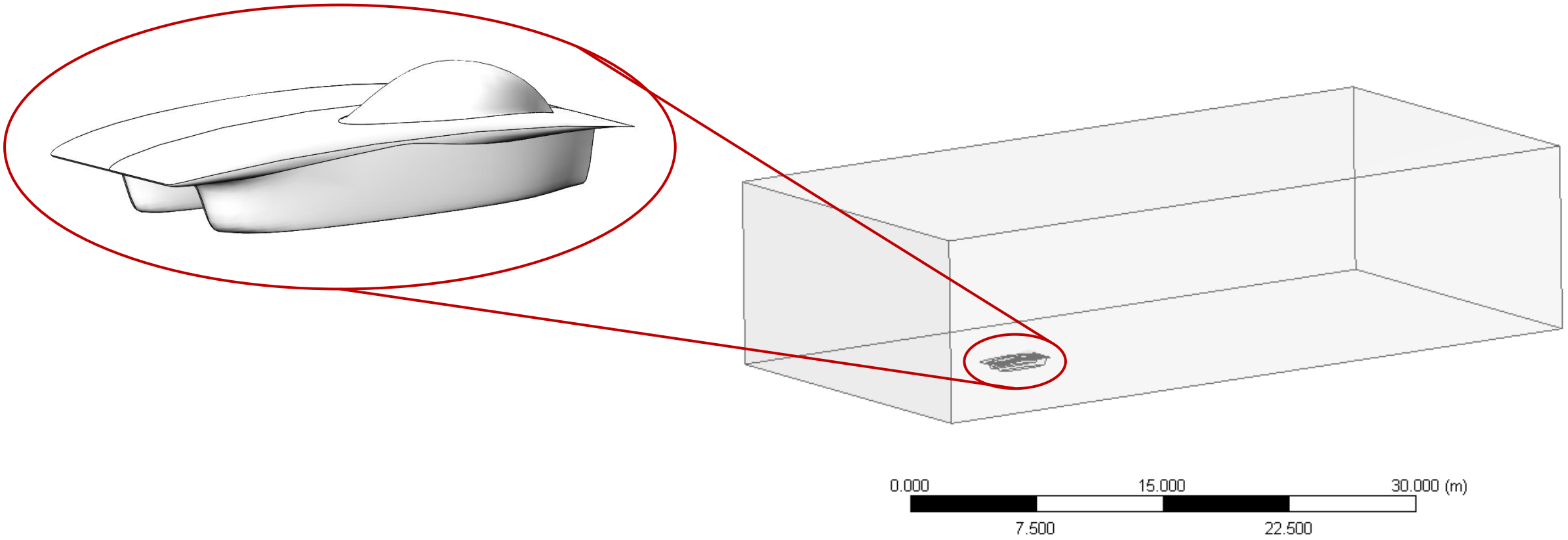
Data disagreed with previous computer simulations

Project goal: develop an airflow model matching empirical data



Geometry Setup in Ansys Fluent

Airflow volume constructed around CAD of solar car outer shell

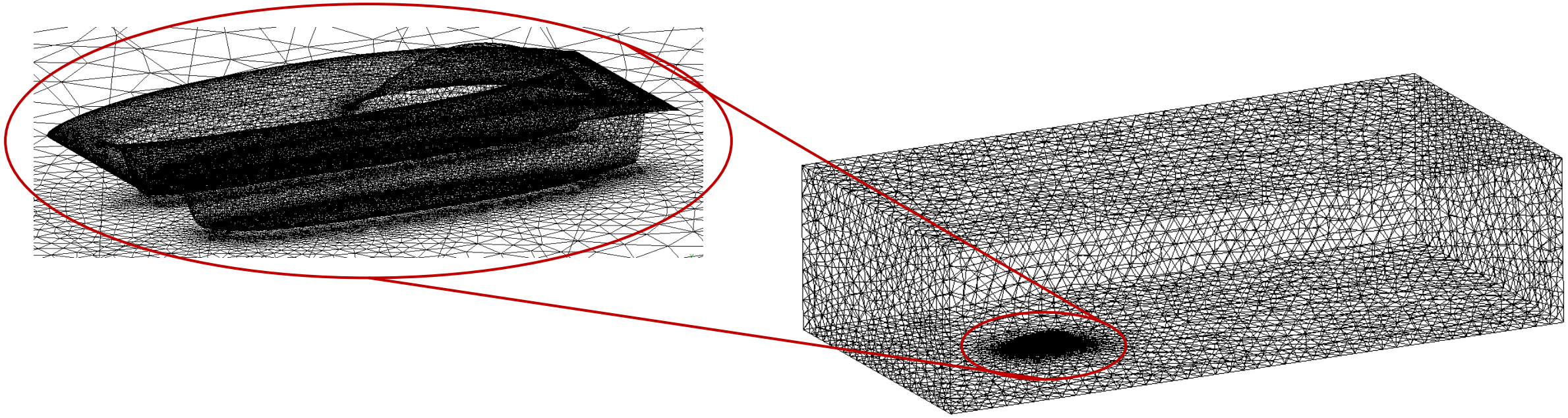


Mesh Generation in Ansys Fluent

Mesh more refined near the solar car surface

Maximum element size: 1.0 m

Minimum element size: 2.5 cm

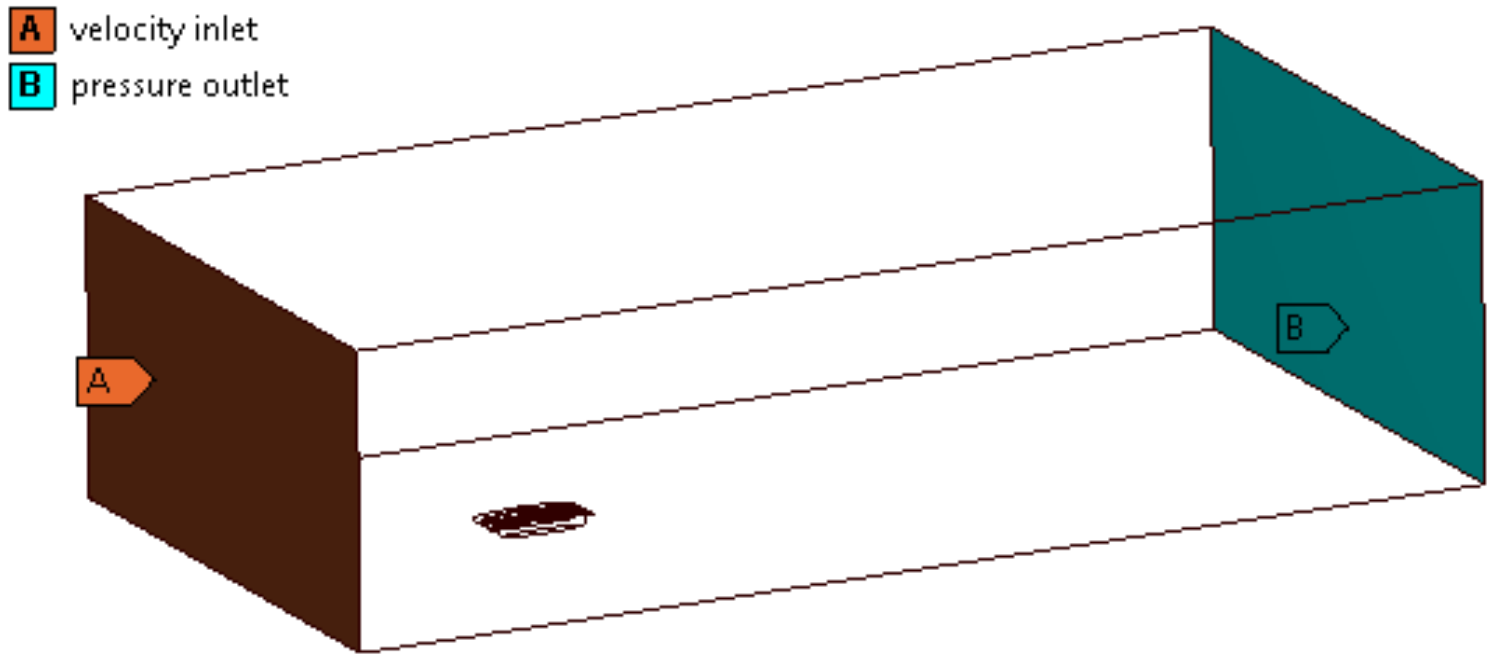


Boundary Conditions

Velocity set to car speed at inlet of bounding box

Atmospheric pressure at outlet of bounding box

No-slip walls



Experimentation with Simulation Settings

Models for turbulent flow:

- k-epsilon
- Reynolds stress model (RSM)
- Large eddy simulation (LES)

Pressure and velocity:

- Coupled vs. decoupled

Discretization methods:

- First order upwind
- Second order upwind
- QUICK

Best Fit Scheme for Empirical Data

k-epsilon model

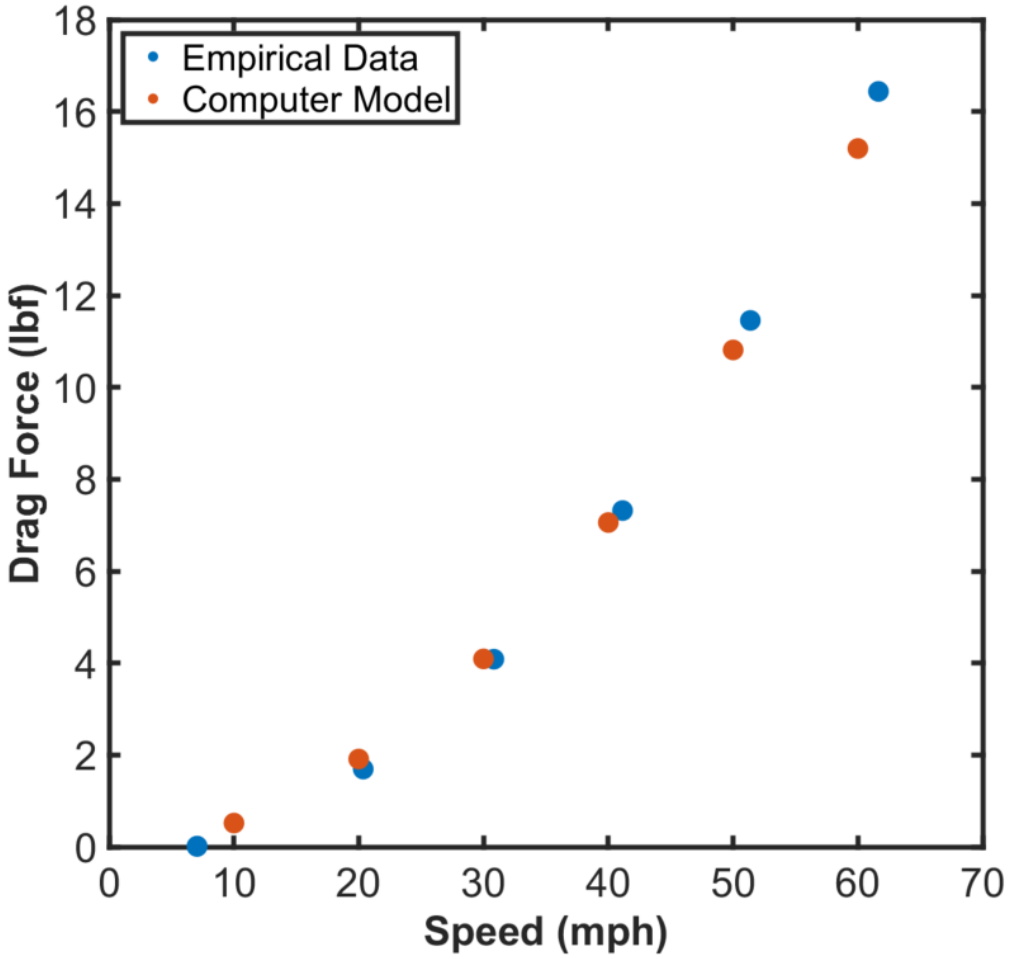
Pressure-velocity coupled

Least squares cell based discretization

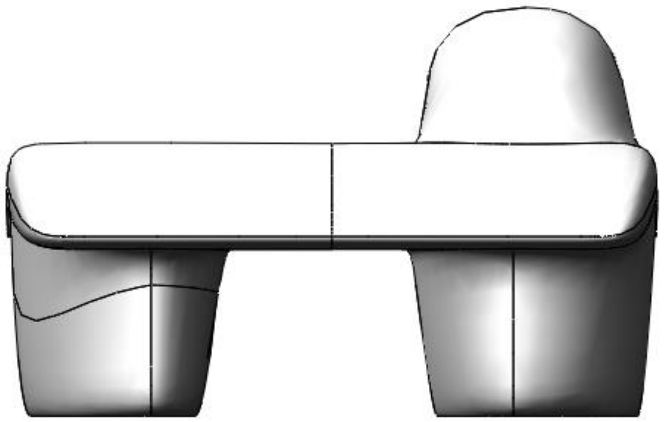
Second order pressure discretization

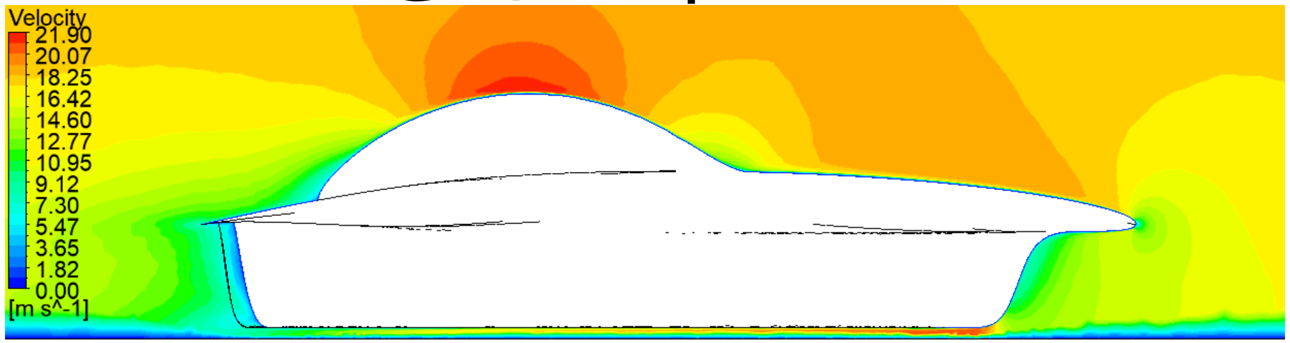
First order upwind momentum discretization

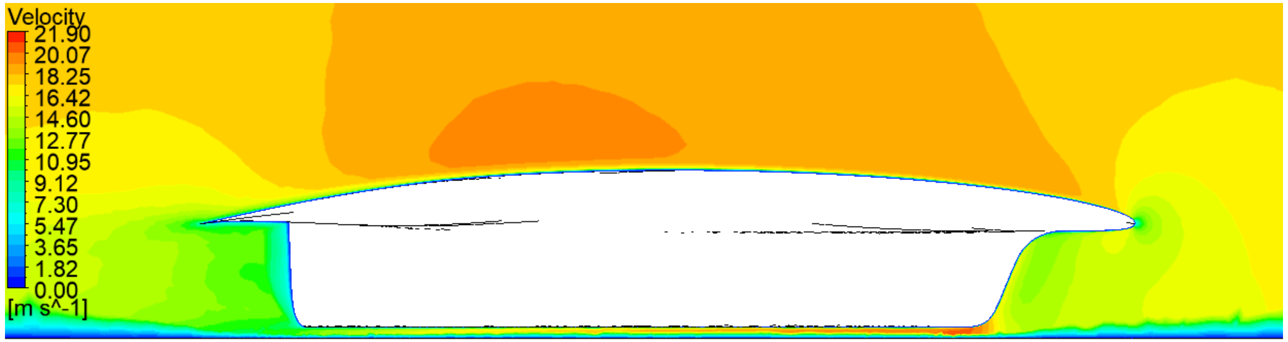
First order upwind discretization for turbulent kinetic energy (k) and dissipation rate (epsilon)



Solar Car Velocity Distribution @ 40 mph (18 m/s)







Solar Car Pressure Distribution @ 40 mph (18 m/s)