SHORT TERM COURSE ON SPORTS AERODYNAMICS

Dr. R. Ramakrishnan, ME,PhD,FIE, C.Eng(I)

Professor & Head

Vignesh S M, B.Tech (AERO)

Research Scholar

Department of Sports Technology
Tamil Nadu Physical Education and Sports University
Chennai

Computational Fluid Dynamics (CFD)

- Numerical Tool to solve the Fluid Dynamics and Heat Transfer Problem

Advantage

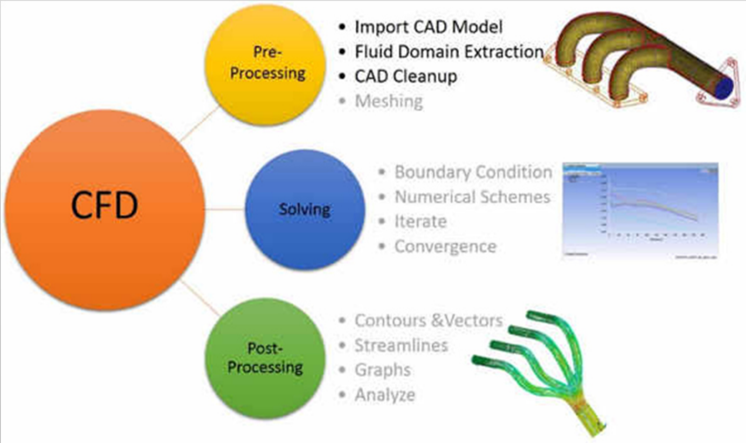
- Inexpensive and fast
- Provide complete information
- No similarity constraints

Disadvantage

- Accuracy and Sensible parameter
- Verification and Validation

Steps in CFD

- **Preprocessing -** Target variable, Selection of the physics of the flow, Geometry and Meshing
- **Solution** - Initialization, Setting algorithm and Convergence criteria
- **Post Processing -** Visualizing and analysing data

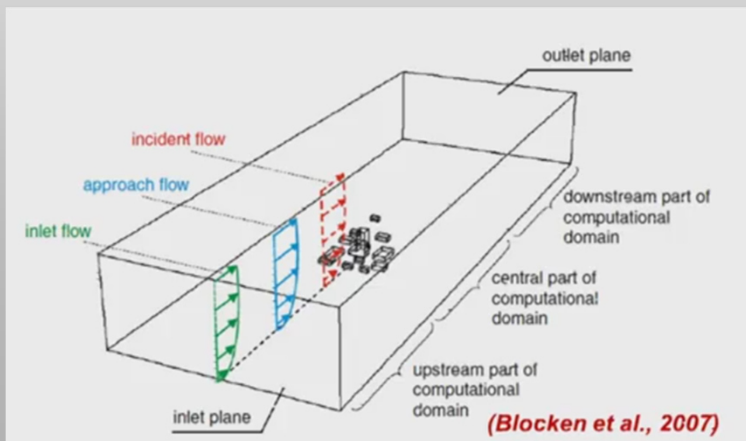


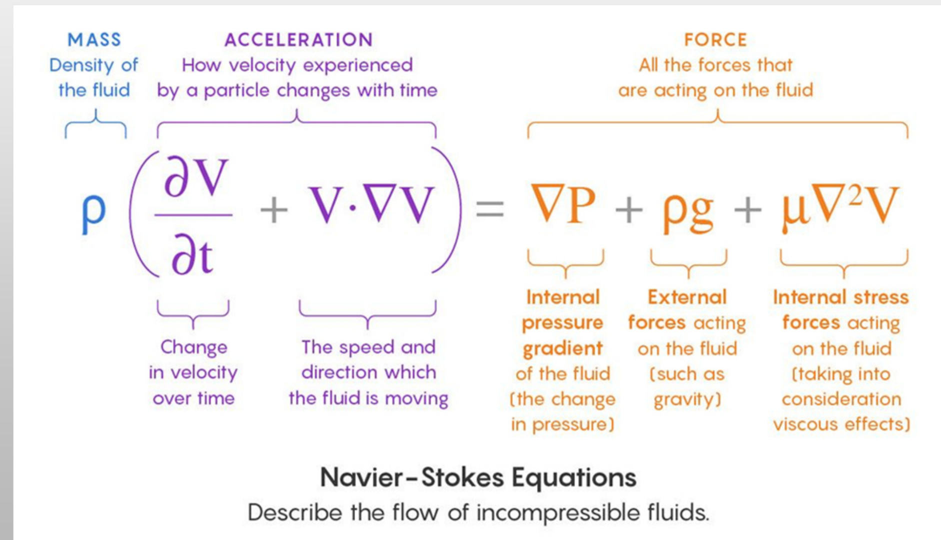
Equations

Continuity - 3 direction - Density Velocity

Momentum - 3 direction - Pressure

6 Unknown (U,P,Q,ρ,v,K,D) = 6 equation

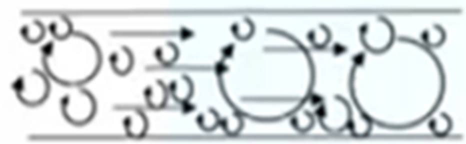




Simulation Method

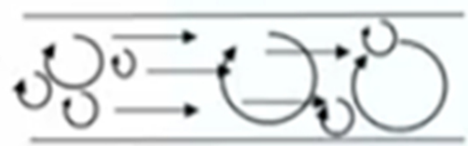
DNS: Direct Numerical Simulation

- Solve the exact Navier-Stokes equations completely
- All vortices/eddies are "solved", nothing "modeled"
- Very time-consuming, huge computational resources, only very simple geometries, huge amounts of data



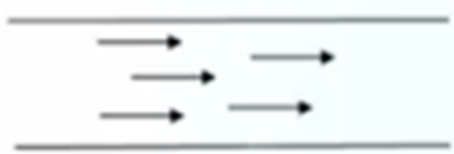
LES: Large Eddy Simulation

- Solve the "filtered" Navier-Stokes equations
- Only large eddies are "solved", small ones are "modeled"
- Not exact, but less computationally demanding



RANS: Reynolds Averaged Navier Stokes

- Solve the "averaged" Navier-Stokes equations
- Only the mean flow is "solved", all eddies are "modeled"
- Not exact, less accurate, but generally applicable

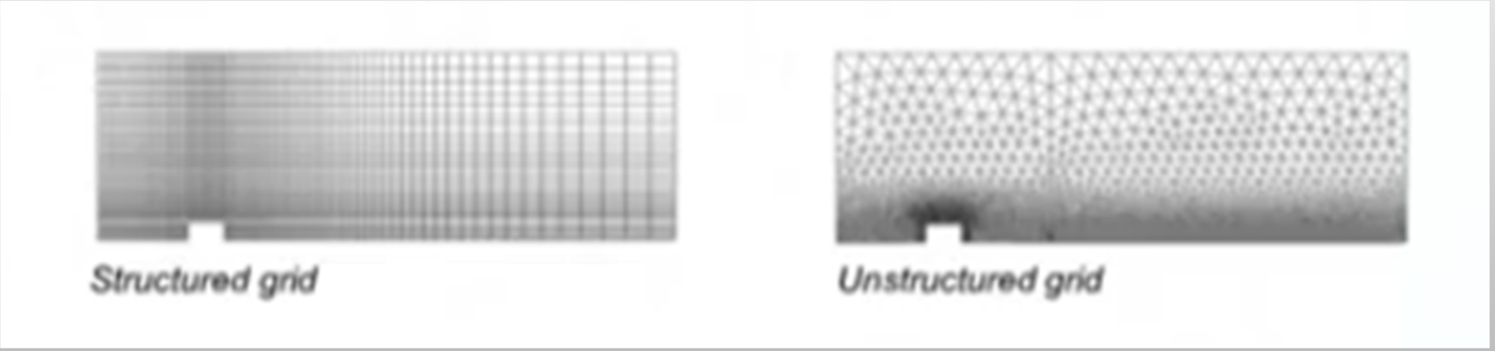


In the RANS approach, the "effect" of turbulence on the mean flow is modeled

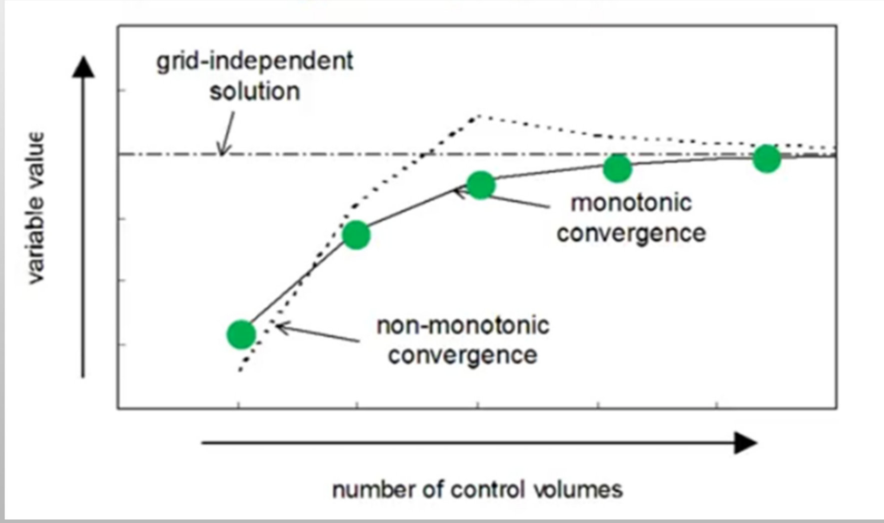
Approximation of NS equation

approach	solve	model (appr.)
DNS	All eddies	Nothing
LES	Large eddies	Small eddies
RANS	Average flow	All

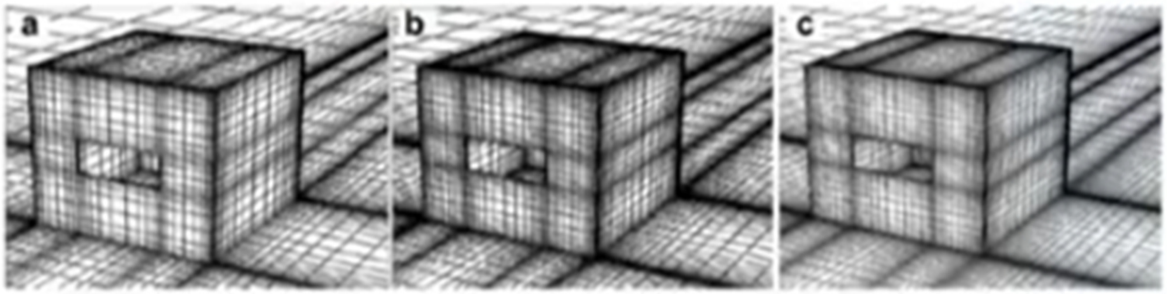
Grid Space



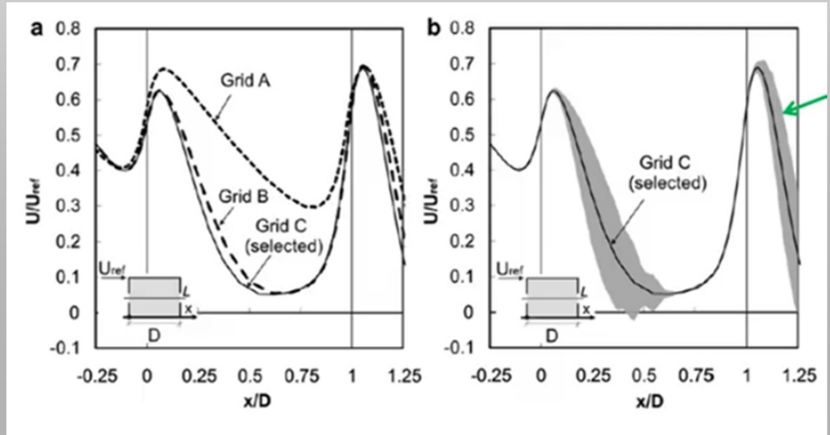
Grid Independence



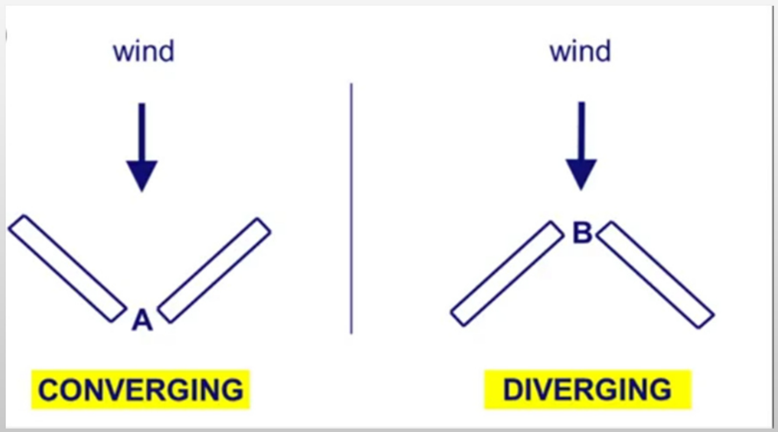
Example - Natural Ventilation study

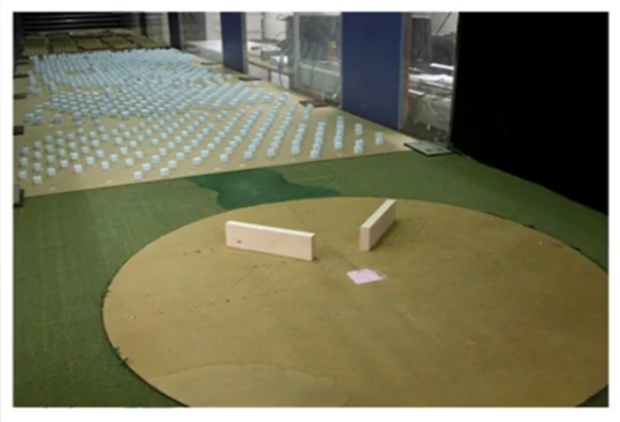


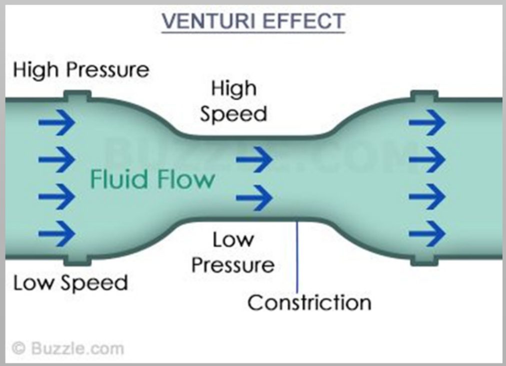
(a) Coarse grid A with 144,696 cells; (b) Middle grid B with 314,080 cells; (c) Fine grid C with 575,247 cells.



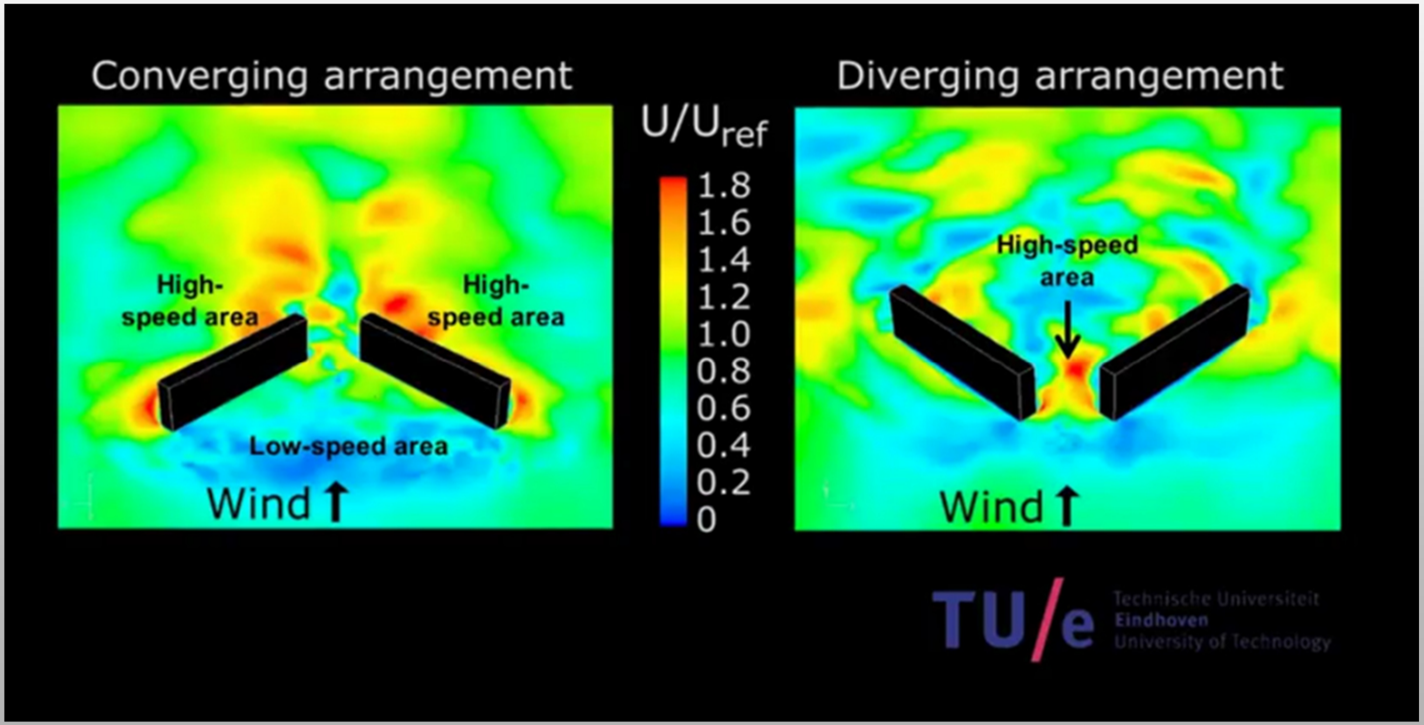
We can't judge fluid flow







Converging and Diverging Buildings



50 years of Computational Wind Engineering: Past, present and future

Accepted for publication in ***Journal of Wind Engineering and Industrial Aerodynamics***, March 13, 2014

50 years of Computational Wind Engineering: Past, present and future [1]

Bert Blocken

Building Physics and Services, Department of the Built Environment, Eindhoven University of Technology, P.O. box 513, 5600 MB Eindhoven, the Netherlands, b.j.e.blocken@tue.nl

Building Physics Section, Department of Civil Engineering, Leuven University, Kasteelpark Arenberg 40 - bus 2447, 3001 Heverlee, Belgium

Abstract

In the past 50 years, Computational Wind Engineering (CWE) has undergone a successful transition from an emerging field into an increasingly established field in wind engineering research, practice and education. This paper provides a perspective on the past, present and future of CWE. It addresses three key illustrations of the success of CWE: (1) the establishment of CWE as an individual research and application area in wind engineering with its own successful conference series under the umbrella of the International Association of Wind Engineering (IAWE); (2) the increasing range of topics covered in CWE; and (3) the history of overview and review papers in CWE. The paper also outlines some of the earliest achievements in CWE and the resulting development of best practice guidelines. It provides some views on the complementary relationship between reduced-scale wind-tunnel testing and CFD. It re-iterates some important quotes made by CWE and/or CFD researchers in the past, many of which are still equally valid today and which are provided without additional comments, to let the quotes speak for themselves. Next, as application examples to the foregoing sections, the paper provides a more detailed view on CFD simulation of pedestrian-level wind conditions

Wind Tunnel Technique

- Problem Statement
- Identify the input variables (Free stream Velocity, Building height etc..) and output variable (Pressure Coefficient, Temperature, Flow stream etc..)
- Build the scale down model
- Identify the suitable measurement technique (Manometers, PIV, Smoke Generator etc..)

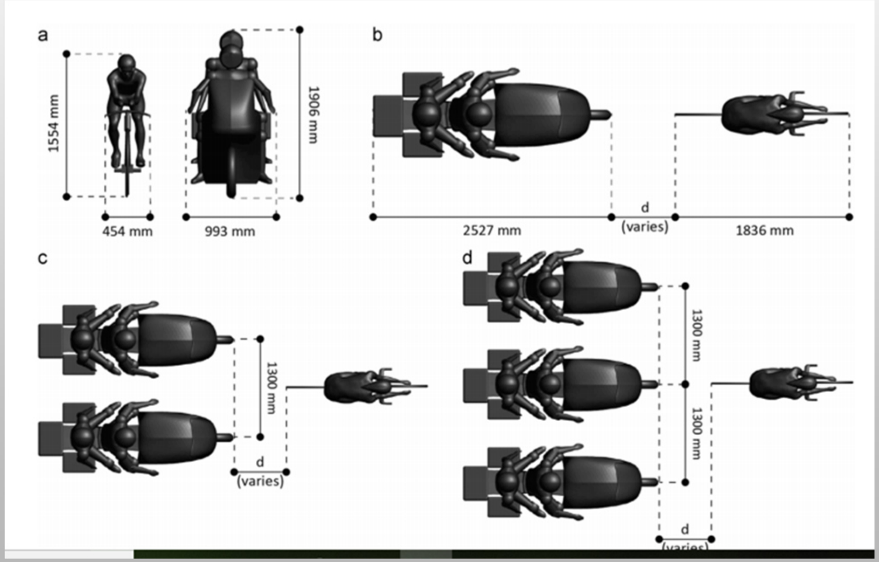
Wind Tunnel Technique

Problem Statement

Aerodynamic benefit for a cyclist by a following motorcycle

Problem Statement

- We need to find the Drag force acts on the Cyclist with 9 different distance (0.25 to 7.5) from the bike in 3 different cases (1,2 & 3 Bikes behind the cyclist)
- Total Number of cases is 9 X 3 = 27 case study



Required Variables

Output Variable

- Variation of **percentage of Drag force** acts on the cyclist in all case
- **Time advantage** in all cases

Input Variable

- Free Stream Velocity in the Wind Tunnel
- Dimension of the Cyclist and bikes model
- Distance between the Cyclist and Bike (Varying variable)

Wind Tunnel Test Section Size

Wind Tunnel Laboratory at the University of Liège in Belgium

The cross-section of the test section is W x H = 2 x 1.5 m²

Reynolds Number matching Model

- **Wind Tunnel Laboratory at the University of Liège in Belgium. The cross-section of the test section is W x H = 2 x 1.5 m²**
- **Scale down ¼, yielding a blockage ratio below 3.5% (Wind Tunnel Velocity 60 m/s Re full Scale reality 15 m/s**

Re= VL/U

Re- Reynolds Number

V- Velocity

L - Characteristic Length

(Frontal Height of the Cyclist

U- Viscosity

$$Re_{\text{Full Scale}} = Re_{\text{Reduced Scale}}$$

	Full Scale	**Reduced Scale**
L	1	1/4
U	$1.46X10^{-5}$	$1.46X10^{-5}$
V	15 m/s	**60 m/s**

Blockage Area

$BR = A_{model} / A_{Wind\ tunnel}$ BR = 0.105 m^2/3m^2 x 100 %= 3.5%

A_{model} - Cross - section Area of the Model 0.13 m X 1.2380 = 0.105

$A_{Wind\ tunnel}$ - Cross - section Area of the Wind Tunnel W x H = 2 m x 1.5 m = 3m^2

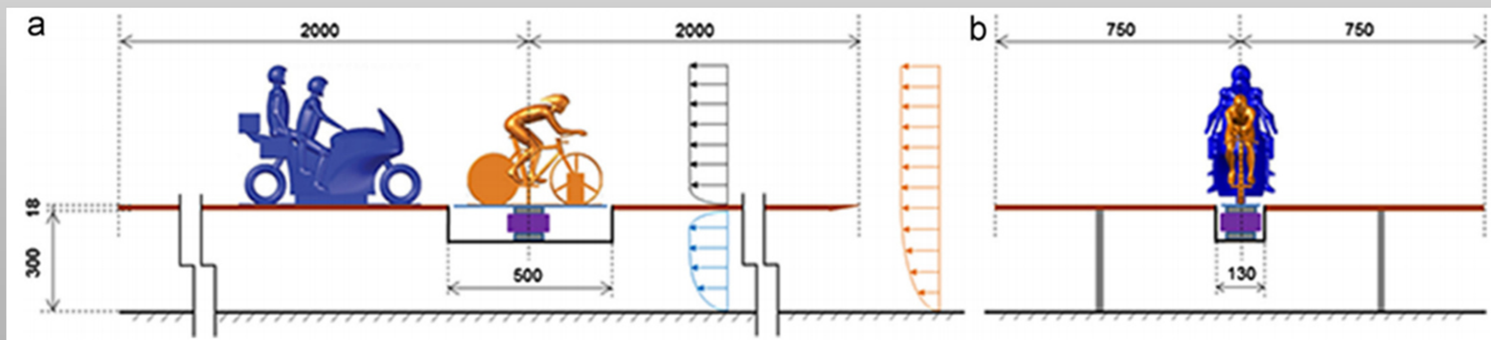


Fig. 3. Wind-tunnel set-up with models on elevated sharp-edged plate to reduce boundary-layer thickness. Dimensions in mm.

Finding Drag on the Cyclist

C_D in Full Scale = C_D in reduced Scale

$D = 0.5\ \rho U^2 S C_D$

Wind Tunnel Reduced Scale - $C_D = D / 0.5\ \rho U^2 S$

Full Scale Model

$D = 0.5\ \rho U^2 S C_D$

ρ= Density of air

U^2 = Velocity of Full scale model

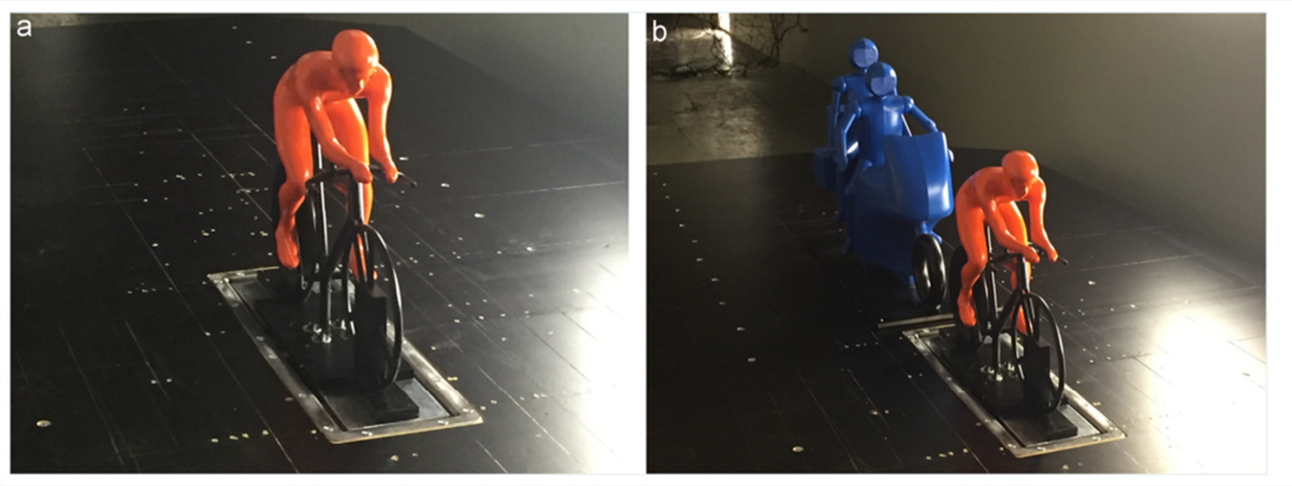
S = Frontal area of the full scale model

ρ= Density of air

U^2 = Wind Tunnel Velocity of reduced scale model

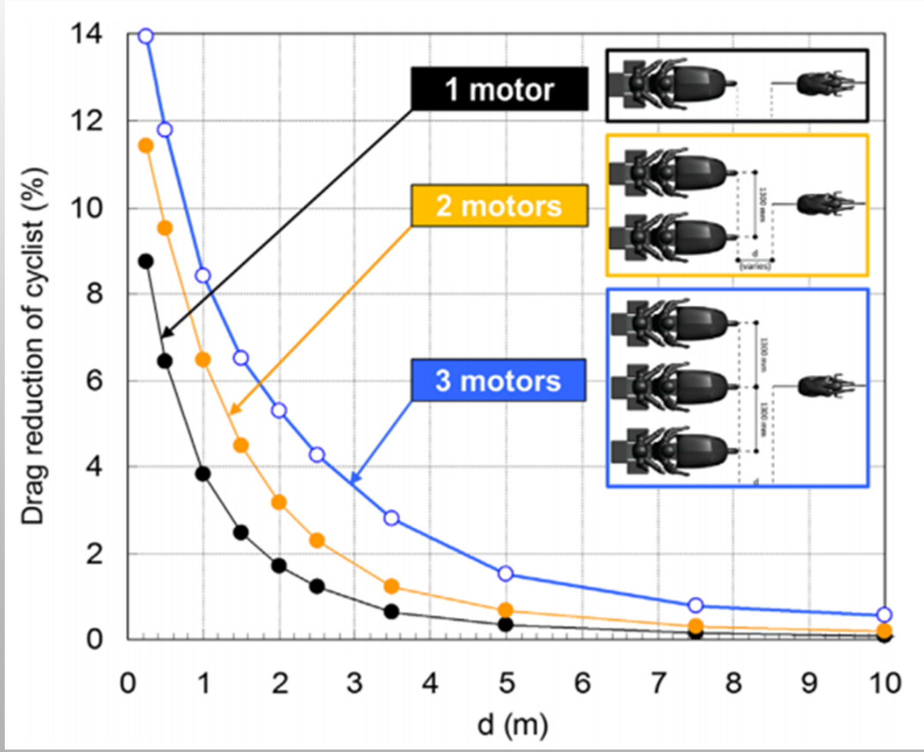
S = Frontal area of the reduced scale model

Fixing Model in the Force Balance



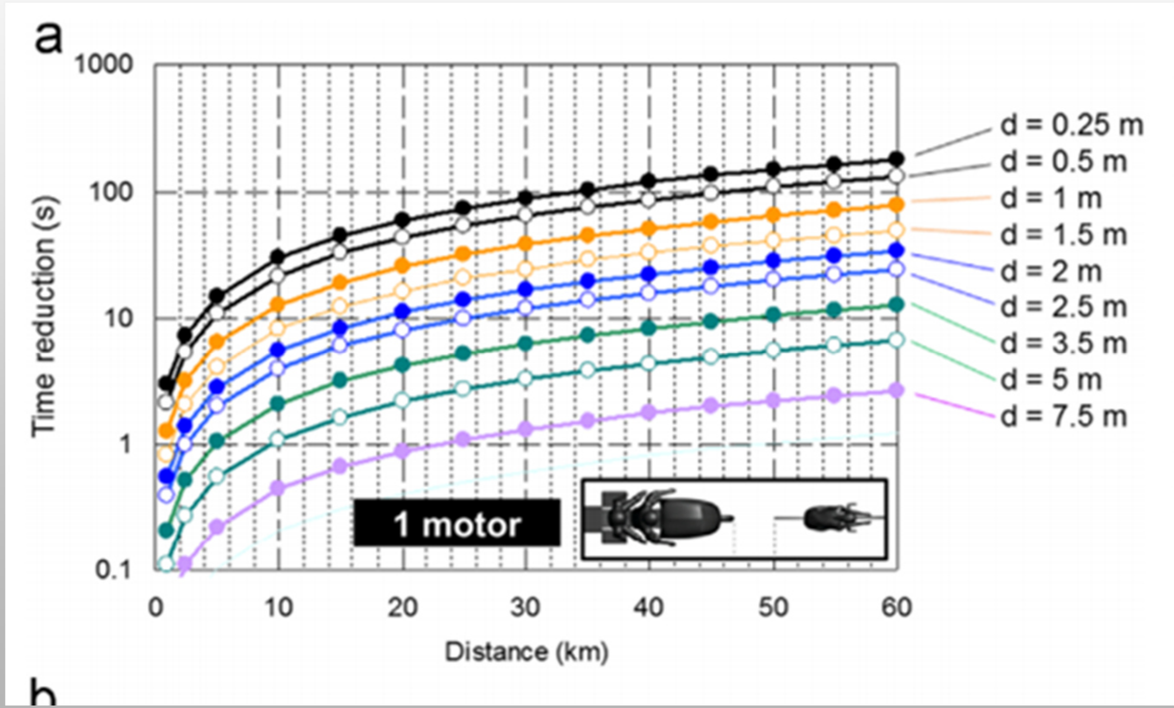
Photograph of models on elevated plate and embedded force balance in the wind tunnel

Results

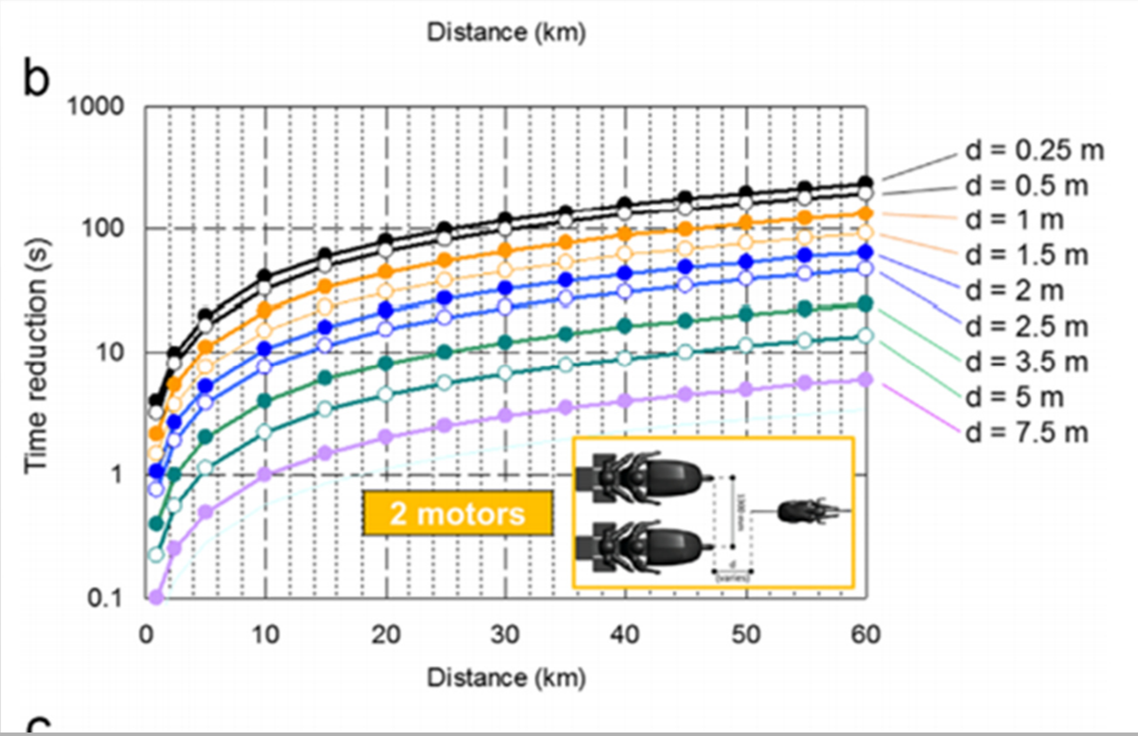


1 motor
2 motors
3 motors
Drag reduction of cyclist (%)
d (m)
0
2
4
6
8
10
12
14
0
1
2
3
4
5
6
7
8
9
10

Results

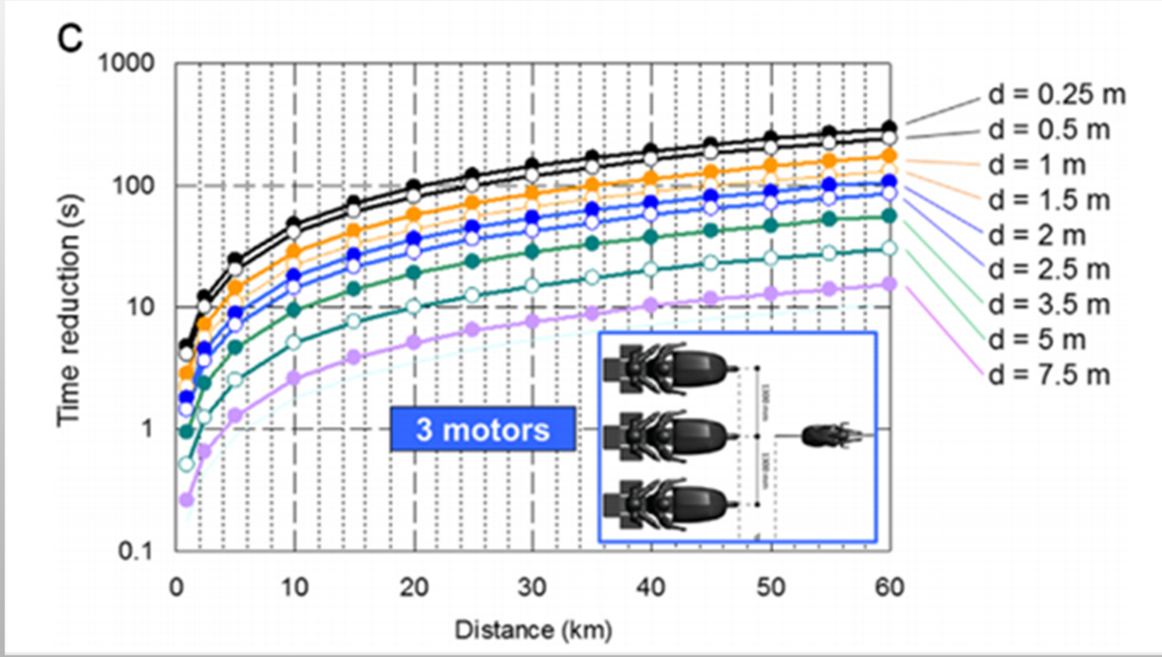

a
1000
100
10
1
0.1
Time reduction (s)
0
10
20
30
40
50
60
Distance (km)
d = 0.25 m
d = 0.5 m
d = 1 m
d = 1.5 m
d = 2 m
d = 2.5 m
d = 3.5 m
d = 5 m
d = 7.5 m
1 motor
b

Results



Distance (km)
b
1000
100
10
1
0.1
Time reduction (s)
0
10
20
30
40
50
60
Distance (km)
d = 0.25 m
d = 0.5 m
d = 1 m
d = 1.5 m
d = 2 m
d = 2.5 m
d = 3.5 m
d = 5 m
d = 7.5 m
2 motors

Results



C
Time reduction (s)
1000
100
10
1
0.1
0
10
20
30
40
50
60
Distance (km)
d = 0.25 m
d = 0.5 m
d = 1 m
d = 1.5 m
d = 2 m
d = 2.5 m
d = 3.5 m
d = 5 m
d = 7.5 m
3 motors

Assignment

Brief Review on following paper (Each Team Must select one paper)

1. Effects of spin on aerodynamic properties of tennis balls, In The Engineering of Sport
2. Effect of dimple structure on the flying characteristics and flow patterns of a golf ball, In The Engineering of Sport
3. An experimental study of cricket ball swing
4. Coupled urban wind flow and indoor natural ventilation modelling on a high-resolution grid: A case study for the Amsterdam ArenA stadium
5. Aerodynamics analysis of wheel configurations in Paralympic hand-cycling: a computational study
6. Aerodynamic drag in cycling pelotons: New insights by CFD simulation and wind tunnel testing

Study the reduction of drag in dimpled ball from smooth ball

Problem Statement

- **Design a golf ball and smooth surface ball with same diameter (Blockage Ratio less than 5%) in CATIA V5 (Design Tutorial available in YouTube)**
- **3D Print the ball in 3D Printer find the drag and separation point in the in both ball**
- **Plot Graph Drag VS Velocity for both ball**
- **Plot Percentage of Reduction Drag VS Velocity**
- **Using Projectile motion formula compare the flight of the smooth ball and the dimpled ball**

Note - Each Team can Vary the dimple depth and gementry (round, pentagon, hexagon) and compare your result