SURGE 2023 End-Term Report

Flow analysis of a shuttlecock using Computational Fluid Dynamics

Mridul Nambiar, Raman Dhingra
Supervisor: Prof. Sanjay Mittal

1. Abstract

The flight of a shuttlecock is a complex phenomenon that is influenced by a variety of factors, including air resistance, aerodynamic forces, and deformation. The purpose of this research is to develop a computational model to investigate the flow around a spinning shuttlecock and to analyse its drag and auto-rotation speeds using Computational Fluid Dynamics (CFD). The air resistance, or drag, on a shuttlecock is caused by the friction between the air and the surface of the shuttlecock. The drag force opposes the motion of the shuttlecock, and it increases as the speed of the shuttlecock increases. The deformation of a shuttlecock can also affect its flight. When a shuttlecock is hit, it can deform slightly. This deformation can change the shape of the shuttlecock, which can affect the airflow around it and the aerodynamic forces on it. The computational model will be developed using the ANSYS Fluent software. The model will be validated against experimental data. The results of the CFD simulations will be used to improve the understanding of the flow around a spinning shuttlecock and to develop new strategies for improving the flight of shuttlecocks. The results of this research could have several applications. For example, the results could be used to design new shuttlecocks with improved flight characteristics. The results could also be used to develop new training methods for badminton players. The importance of the shuttlecock's shape and feathers in determining its flight characteristics, the effects of wind on the flight of a shuttlecock, and the challenges of modelling the flow around a spinning shuttlecock using CFD are all important topics that could be included in the expanded paragraph.

2. Objective

a. Generating a mesh designed for simulating a spinning shuttlecock

The first step in simulating the flow around a spinning shuttlecock is to generate a mesh. The mesh is a grid of points that represents the surface of the shuttlecock and the surrounding air. The mesh must be fine enough to capture the details of the flow, but it must also be coarse enough to be computationally feasible. There are a variety of approaches to generating and optimizing a mesh for a spinning shuttlecock. One approach is to use a commercial CFD software package. These packages typically have built-in meshing tools that can be used to generate a mesh for a spinning shuttlecock. Another approach is to use an open-source meshing tool. These tools are typically more flexible than commercial meshing tools, but they can be more difficult to use. Once the mesh has been generated, it must be optimized. The optimization process involves removing unnecessary points from the mesh and adjusting the size and shape of the points to improve the accuracy of the simulation.

b. Studying various approaches for solving a spinning shuttlecock

There are a variety of approaches for solving the flow around a spinning shuttlecock. Firstly, deciding the solver model is crucial. One approach is to use a steady-state solver. A steady-state solver assumes that the flow around the shuttlecock does not change over time. This is a simplification, but it can be a reasonable approximation for the flow

around a spinning shuttlecock at high Reynolds numbers. Another approach is to use an unsteady-state solver. An unsteady-state solver allows the flow around the shuttlecock to change over time. This is a more accurate approach, but it is also more computationally expensive. The choice of approach for solving the flow around a spinning shuttlecock depends on the specific goals of the simulation. If the goal is to predict the drag and autorotation speed of the shuttlecock, then a steady-state solver may be sufficient. However, if the goal is to understand the detailed flow structures around the shuttlecock, then an unsteady-state solver may be necessary. Now there are three methods which are generally used to simulate a rotating object, mesh motion, frame motion and boundary condition manipulation. Mesh motion is a relatively more computationally expensive method where a rotating zone is created around the object and the whole of the rotating zone mesh slides with respect to the stationary enclosure. In Frame motion however, the mesh remains stationary but the equations change to accommodate rotation, therefore making it more computationally efficient. Boundary Condition manipulation can only be used for objects with an axisymmetric geometry, which is not true in case of a shuttlecock.

c. Investigating the drag on the shuttlecock

The drag on a shuttlecock is the force that opposes its motion through the air. The drag force is caused by the friction between the air and the surface of the shuttlecock. The drag force increases as the speed of the shuttlecock increases. The drag on a shuttlecock can be investigated using CFD simulations. The drag force can be calculated from the results of the simulation. The drag force can also be compared to experimental data to validate the CFD simulation.

d. Investigating the auto-rotation speed of the shuttlecock

The autorotation speed of a shuttlecock is the speed at which the shuttlecock will spin without any external forces acting on it. The autorotation speed of a shuttlecock can be investigated using CFD simulations. The autorotation speed can be calculated from the results of the simulation. The autorotation speed can also be compared to experimental data to validate the CFD simulation

3. Methodology

a. This research used ANSYS Fluent software, which is a widely used CFD software, to simulate the flow around a spinning shuttlecock.

b. An existing model of a shuttlecock was optimized to decrease computational time, without affecting accuracy in results.

c. Techniques like mesh motion and frame motion were used to accurately simulate the rotation of the shuttlecock.

d. These techniques were validated by running simulations of flow across a 2D rotating cylinder.

e. The unsteady Reynolds-averaged Navier-Stokes (URANS) equations were solved for the fluid flow using the turbulence model k-e numerically.

f. The simulation results were analysed and used to calculate the drag coefficient, moments, and auto-rotation speed of the shuttlecock.

4. Results and Conclusion

Comparison of simulations run using Mesh motion and Boundary Condition manipulation on the case of flow across a 2D rotating cylinder with the results showcased in the paper "A Numerical Investigation into the Steady Flow past a Rotating Circular Cylinder at Low and Intermediate Reynolds Numbers" by D. B. Ingham and T. Tang Department of Applied Mathematical Studies, University of Leeds.
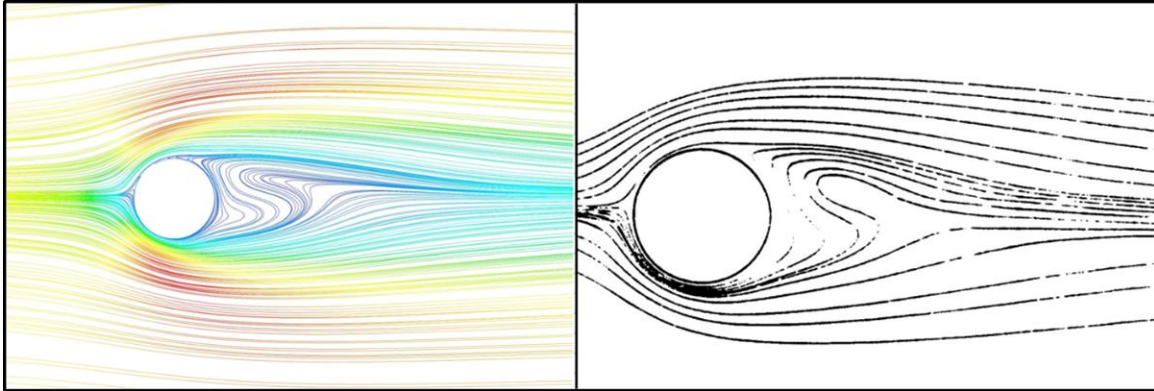
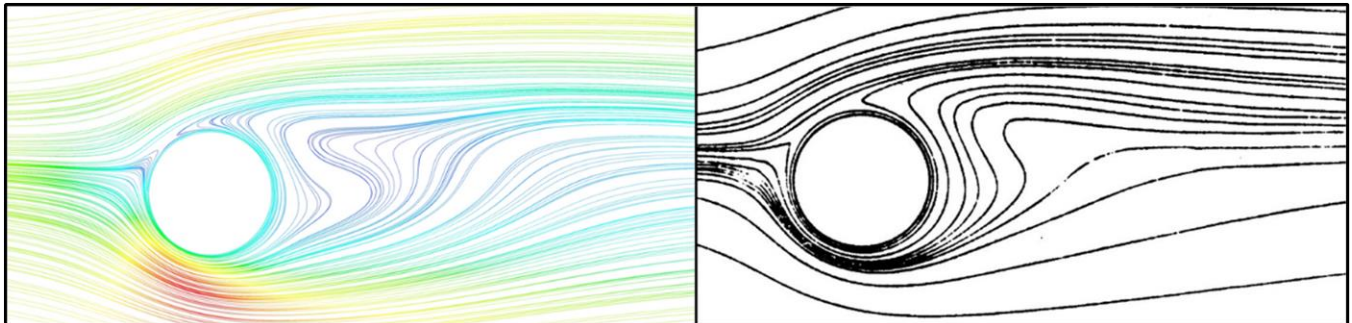### a. Re = 20 & α = 0.1

Figure 1

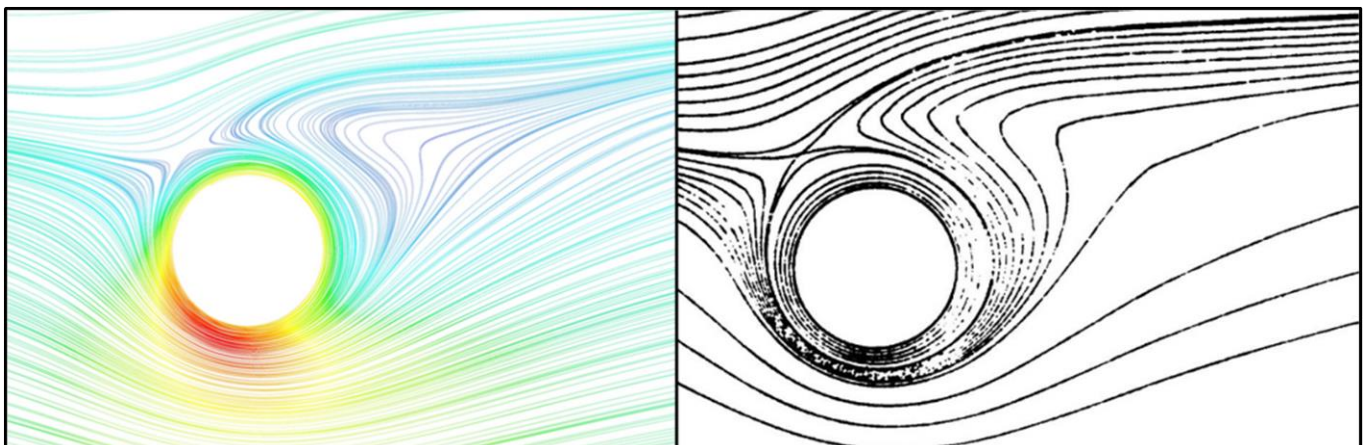### b. Re = 20 & α = 0.5

Figure 2

c. Re = 20 & α = 2

Figure 3

- From these results we concluded that our method works well and now simulations can be run to solve our actual problem statement.
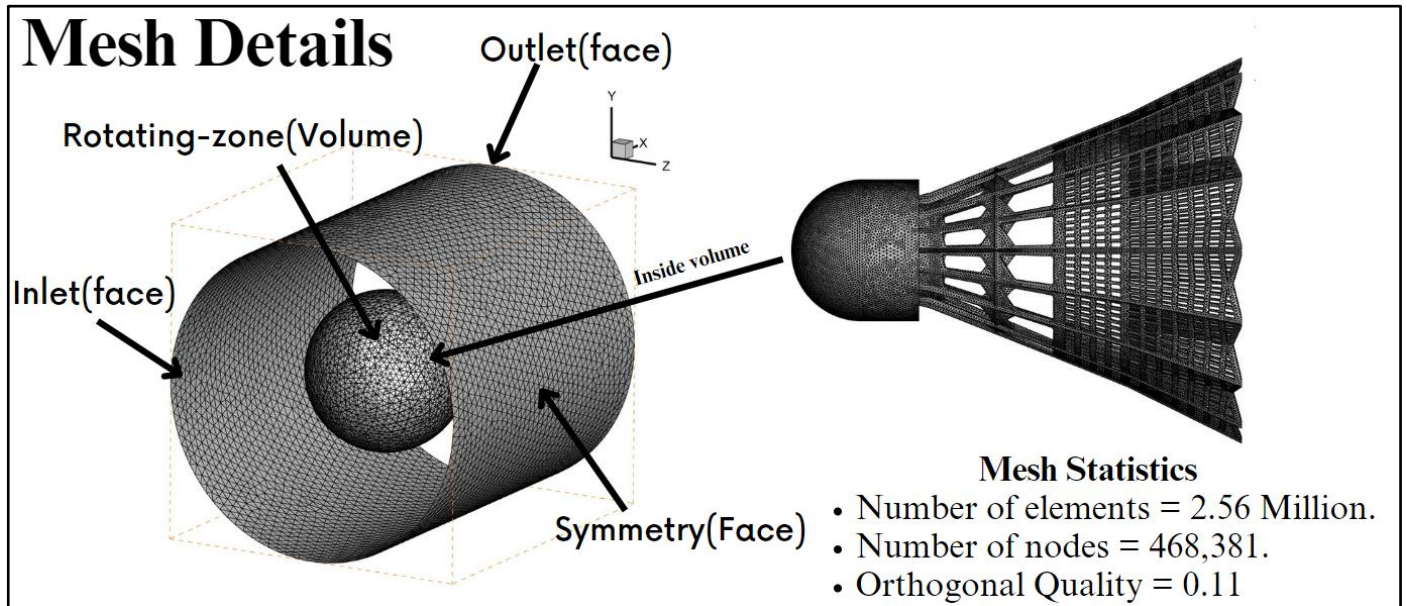
Mesh Details

Outlet(face)

Rotating-zone(Volume)

Inlet(face)

Inside volume

Symmetry(Face)

Mesh Statistics
- Number of elements = 2.56 Million.
- Number of nodes = 468,381.
- Orthogonal Quality = 0.11

Figure 4

d. Velocity profile plotted using Frame Motion

Velocity in Stn Frame
Streamline 1
1.217e+00

9.129e-01

6.086e-01

3.043e-01

8.951e-06
[m s^-1]

ANSYS
2021 R1

0   0.045   0.090 (m)
0.0225   0.067
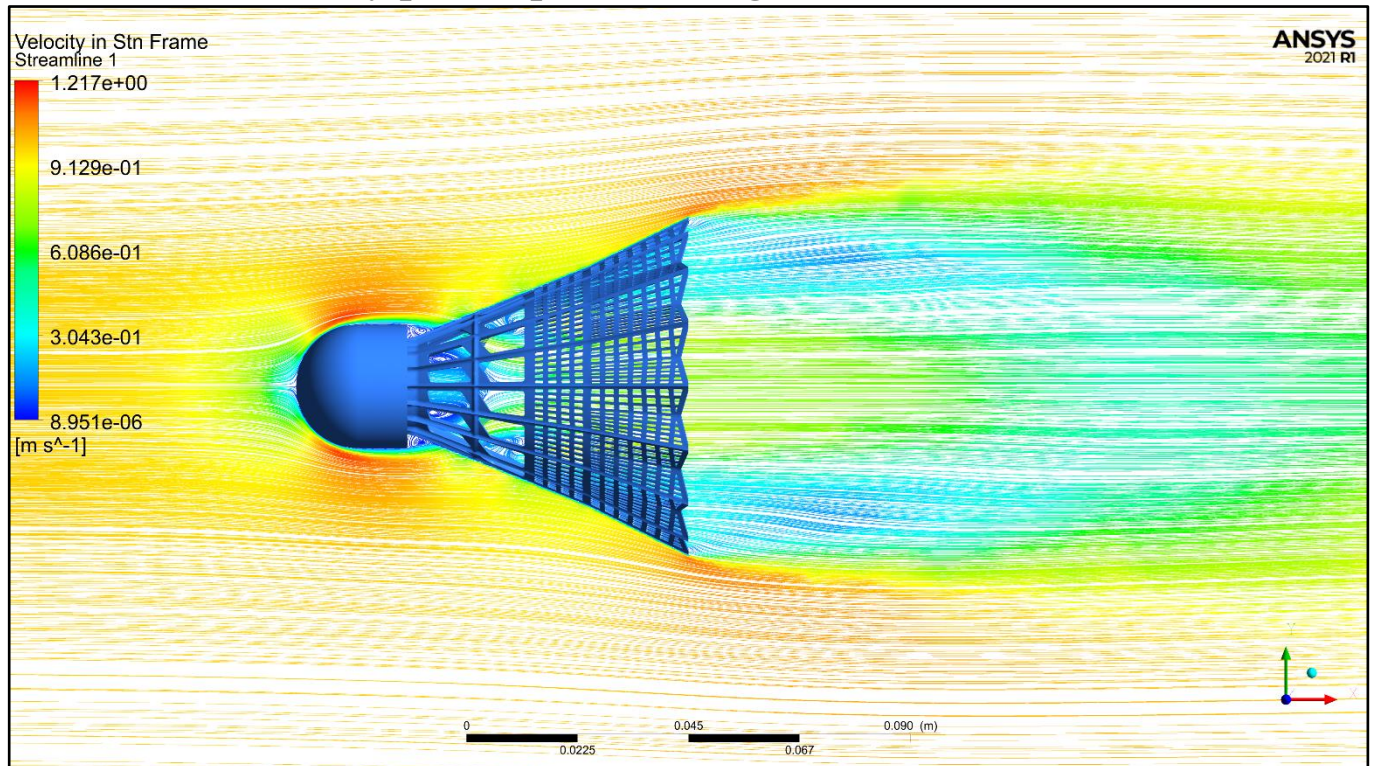
Figure 5

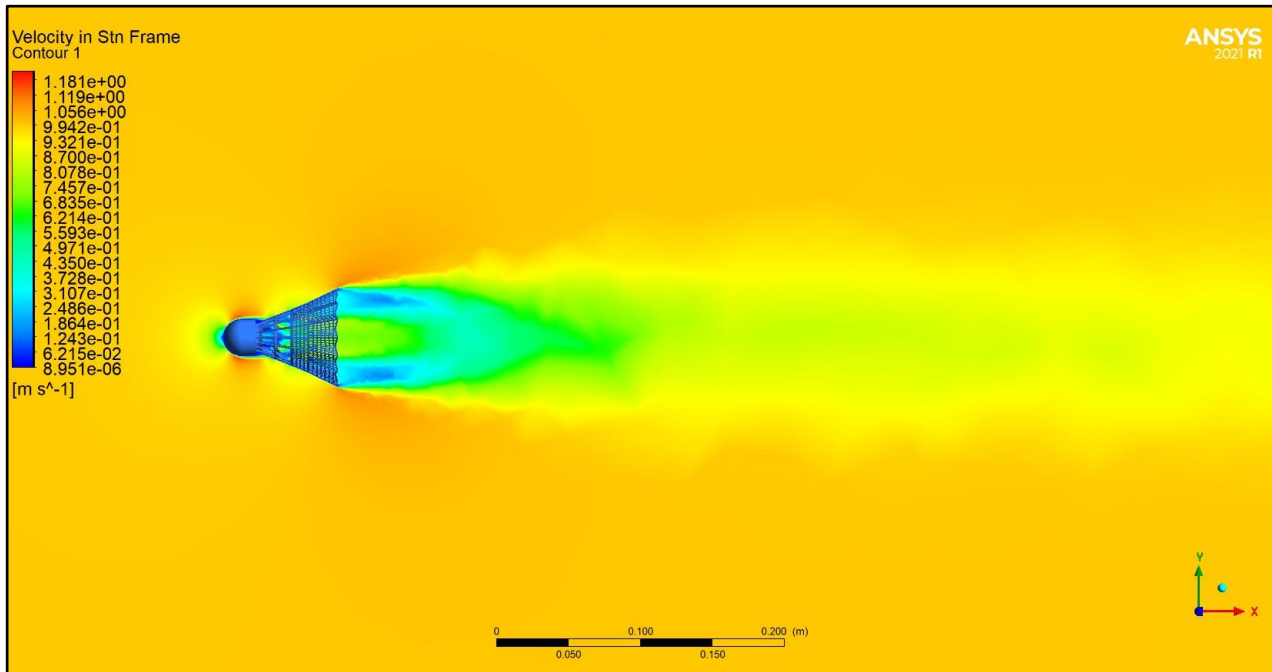## e. Wake Region Contours using Frame Motion

Figure 6

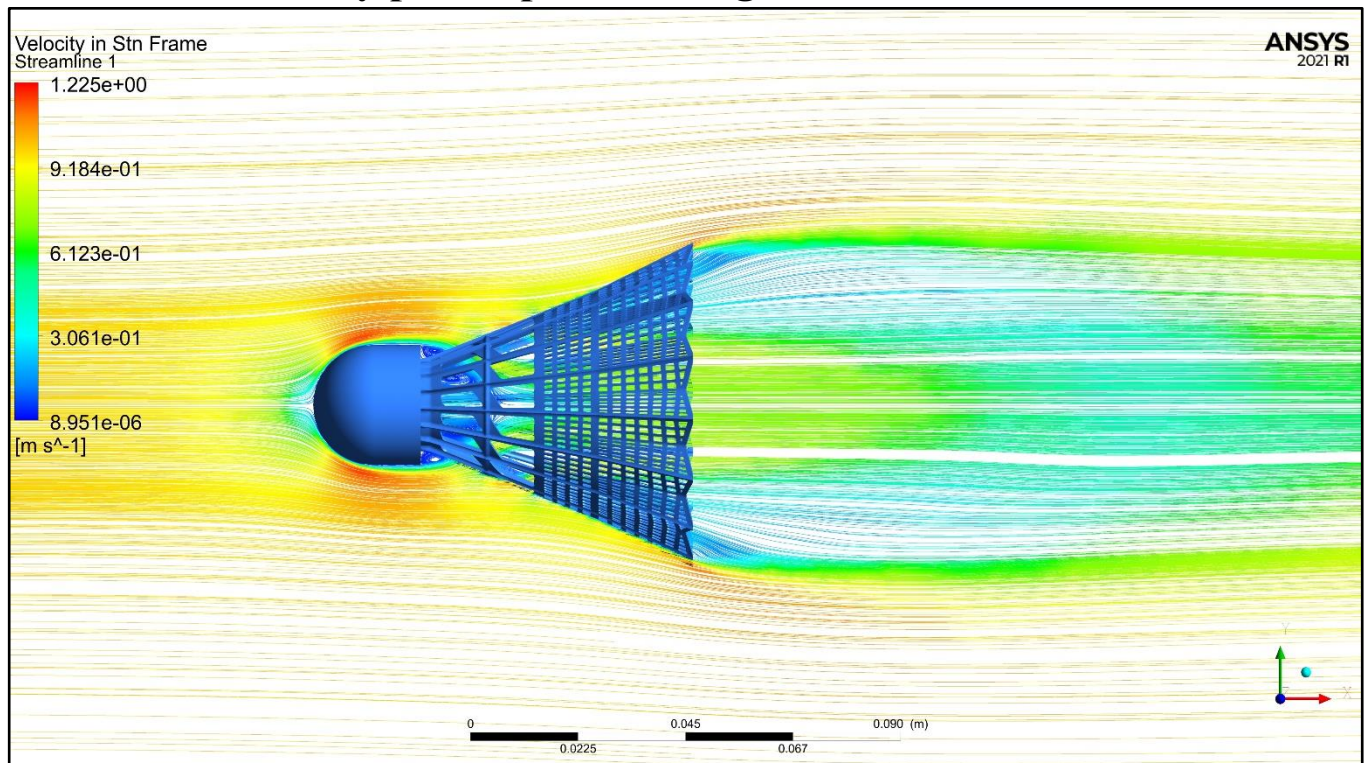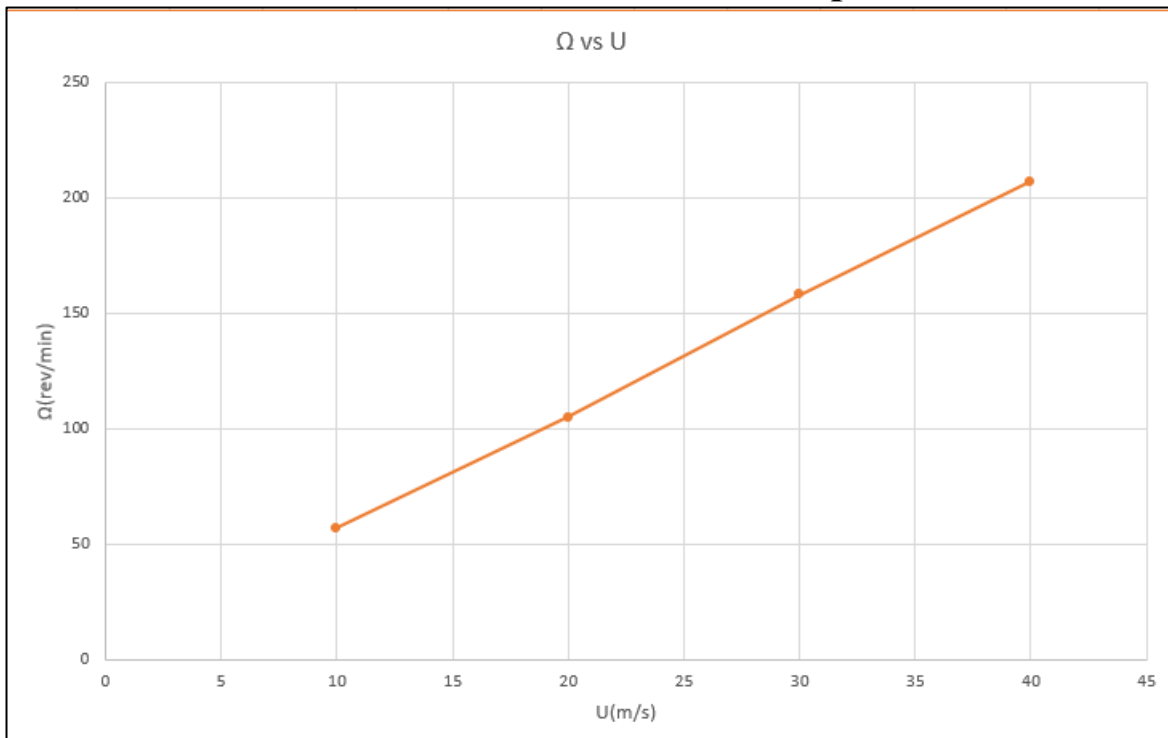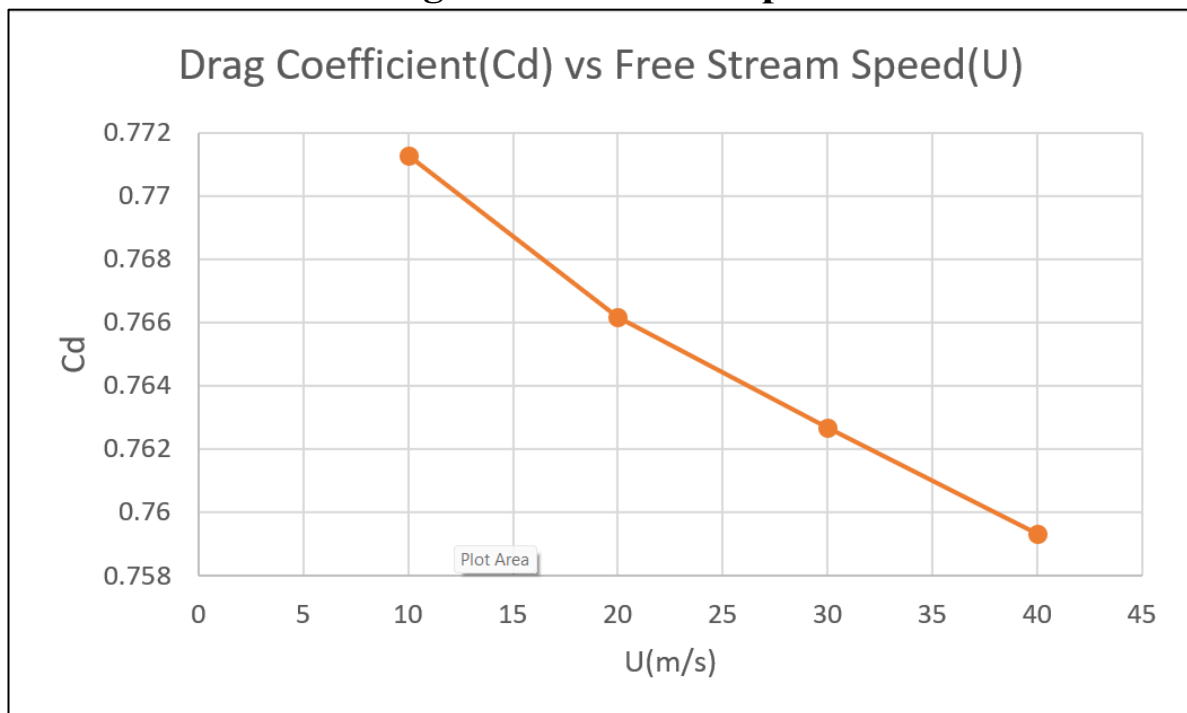## f. Velocity profile plotted using Mesh Motion

Figure 7

- We can see that both the methods are giving comparable results. However, as Frame motion is computationally less expensive all further simulations were run using Frame Motion.

Auto-rotation vs Free Stream Speed

Ω vs U

Drag vs Free Stream Speed

Drag Coefficient(Cd) vs Free Stream Speed(U)

- Increasing the upstream velocity in our simulations resulted in a consistent and approximately linear decrease in the drag coefficient of the rotating shuttlecock.
- Our simulations demonstrated that the autorotation speed of the rotating shuttlecock increased in a nearly linear fashion with increasing upstream velocity.