CFD analysis of Flow around a Golf Ball

Rohan Gulve

BTech in Mechanical Engineering

Indian Institute of Technology Bombay, Mumbai, Maharashtra, India

Abstract

The objective of this project is to study aerodynamic characteristics of a Golf Ball by doing CFD analysis of the air flow around it, using open source CFD package OpenFOAM. Using the CFD analysis, the drag on the ball is to be determined for various Reynolds numbers and the drag is compared to drag acting on a smooth sphere of similar dimensions under similar conditions. For this an incompressible SimpleFoam solver is used for numerical analysis. 'komega SST' a Reynolds Averaged Simulation (RAS) model in OpenFoam is used to model the turbulence. The CAD of the golf ball is taken from grabcad. The mesh is generated using blockMesh and snappyHexMesh utility in openFoam.

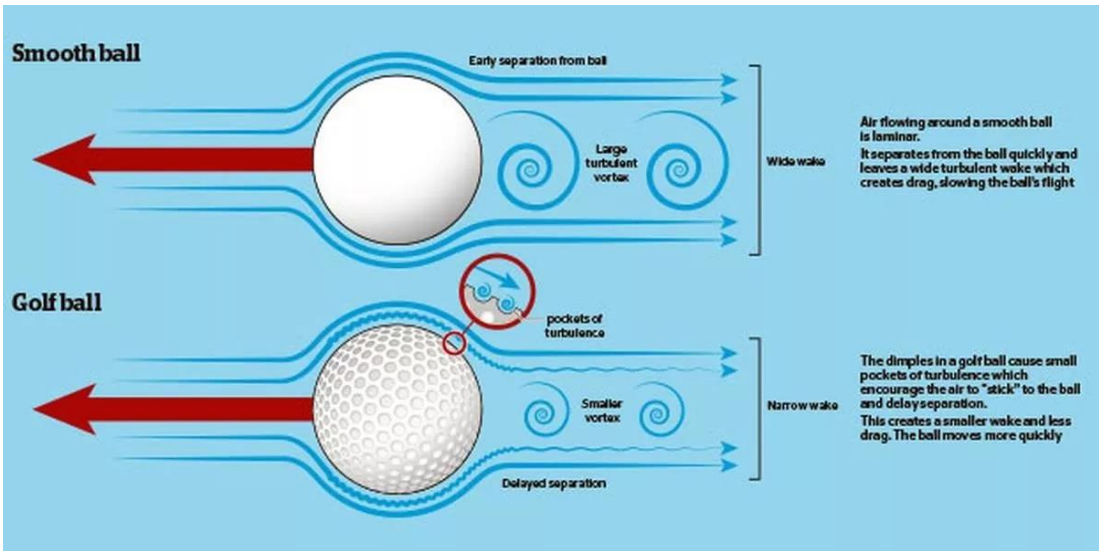


Fig1: Comparison of Flow over a smooth sphere and a golf ball

Introduction

An object moving in a fluid has a high-pressure zone on the front side known as the stagnation region. The fluid flows smoothly to the sides and somewhere in the back region the fluid boundary layer separates from the surface of the ball and thus cause a wake region at the back side of the object. The wake region formed has a lower pressure. Due to this difference in pressure at the front and the back, the object experiences drag.

An interesting thing is observed in plot of Cd Vs. Re for a sphere; when the Re increases above $2x10^5$ there is sudden drop in value of Cd as seen in Fig 2. This is caused as at Re = $2x10^5$ the flow around the sphere transitions from laminar to turbulent. Since the turbulent boundary layer is more resistant to flow separation than a laminar boundary layer, the turbulent layer stays attached to the object for longer distance and separates at a later stage. In the case of sphere, this reduces the wake region and thus the drag acting on the sphere.

This has inspired the design of a golf ball. The dimples on the golf ball cause the flow around he ball to become turbulent at a much earlier stage. Thus at playing conditions the golf ball has a lesser Cd as compared to a sphere of similar dimensions. In this project we try to verify this result using CFD simulations.

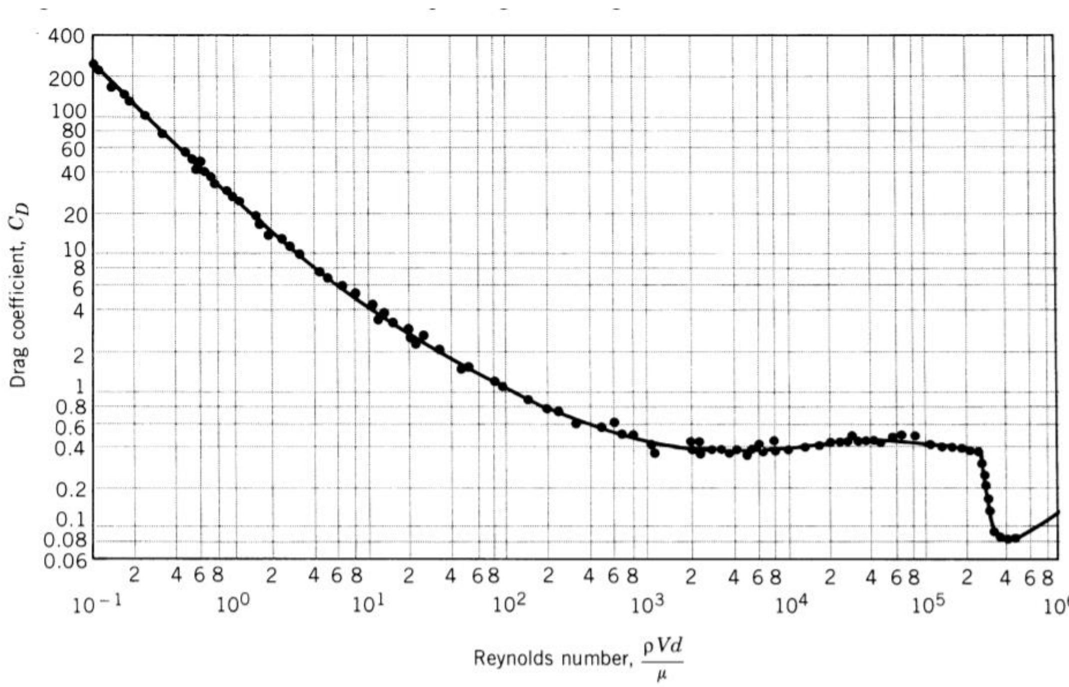


Fig 2: Effect of Reynolds number on drag coefficient of a smooth sphere

Problem Statement

The aim is to get Cd values of a golf ball for various Re. Plot and observe this variation and compare it to Cd Vs Re plot for a smooth sphere.

Governing Equations

The turbulence specific dissipation rate equation is given by:

$$\frac{D}{Dt}(\rho\omega) = \nabla \cdot (\rho D_\omega \nabla_\omega) + \frac{\rho\gamma G}{v} - \frac{2}{3}\rho\gamma\omega(\nabla \cdot u) - \rho\beta\omega^2 - \rho(F_1 - 1)CD_{k\omega} + S_\omega$$

and the turbulence kinetic energy by:

$$\frac{D}{Dt}(\rho k) = \nabla \cdot (\rho D_k \nabla k) + \rho G - \frac{2}{3}\rho k(\nabla \cdot u) - \rho\beta^*\omega k + S_k$$

The turbulence viscosity is obtained using:

$$v_t = \frac{a_1 k}{\max(a_1\omega_1 b_1 F_{23} S)}$$

Default model coefficients

α_{k1}	α_{k2}	$\alpha_{\omega1}$	$\alpha_{\omega2}$	β_1	β_2	γ_1	γ_2	β_*	a_1	b_1	c_1
0.85	1.0	0.5	0.856	0.075	0.0828	5/9	0.44	0.09	0.31	1.0	10.0

Initialisation

For isotropic turbulence, the turbulence kinetic energy can be estimated by:

$$k = \frac{3}{2}\left(I\left|u_{ref}\right|\right)^2$$

where I is the intensity, and uref a reference velocity. The turbulence specific dissipation rate follows as:

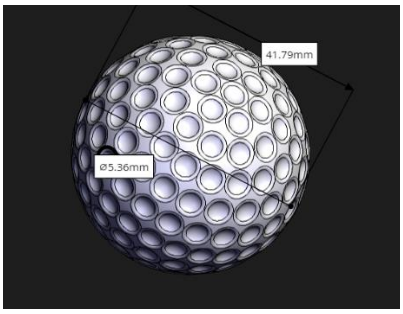
$$\omega = \frac{k^{0.5}}{C_\mu^{0.25} L}$$

where Cμ is a constant equal to 0.09, and L a reference length scale.

Simulation Procedure

Geometry and Mesh

The geometry is taken from GrabCAD



The mesh is created using blockMesh and snappyHexMesh utilities in openFoam

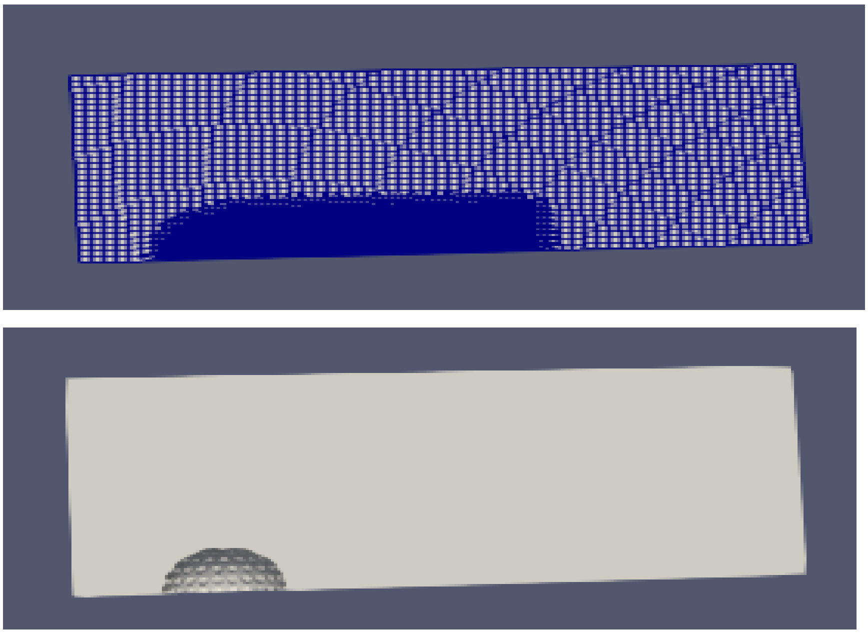


Fig3: Mesh for computation

While simulating only a quarter was taken into considerations due to symmetry and to reduce computational time.

An additional refinement is given in the wake region to better capture the effect of eddies formed in the wake region.

Initial and Boundary Conditions

	Velocity	**Pressure**	**k**	**nut**	**omega**
Inlet	Fixed Value	Zero Gradient	Fixed Value	Calculated	Fixed Value
Walls	Slip	Zero Gradient	kqrWall -Function	nutkWall -Function	omegaWall -Function
Symmetric Walls	symmetry	symmetry	symmetry	symmetry	symmetry
Outlet	Zero Gradient	Fixed Value (0 Pa)	Zero Gradient	calculated	Zero Gradient
Ball	No Slip	Zero Gradient	kqrWall -Function	nutkWall -Function	omegaWall -Function

Solver

Air can be considered incompressible for velocities less than 100 m/sec. And velocity of golf ball rarely does exceed 100m/s, it is ok to use incompressible solver. In this project incompressible simpleFoam solver is used. It is a steady state solver, this is sufficient as we are mainly interested in final drag coefficient.

Results and Discussions

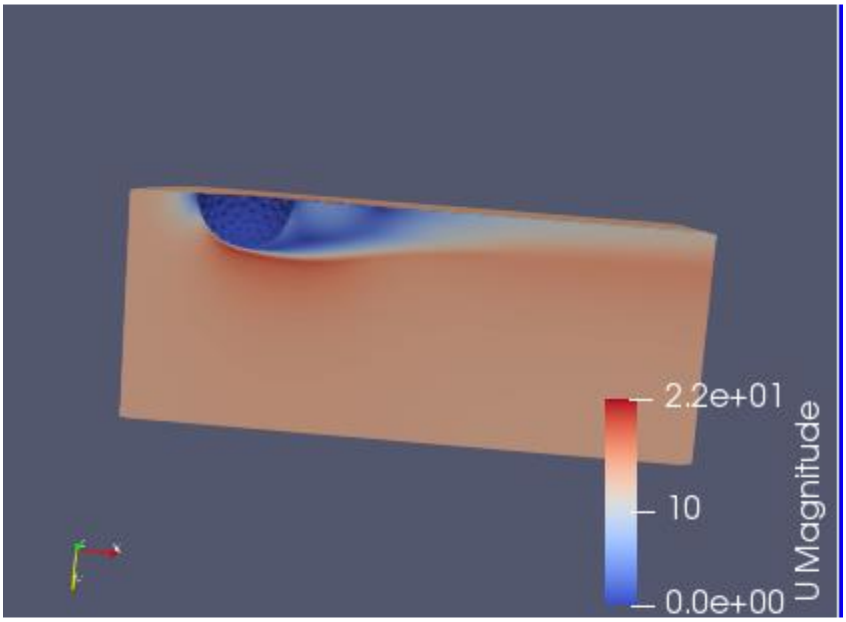


Fig 4: Velocity contour

Values of Cd Vs. Re in literature are as follows :

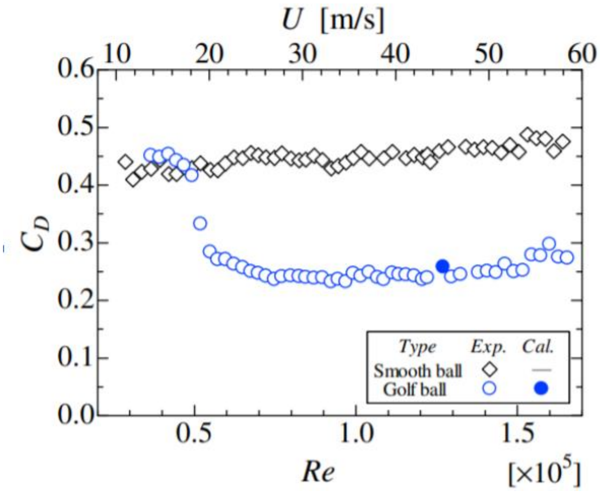


Fig 5: Cd Vs Re for sphere and golf ball

(Due to some slight errors in the settings of the case, the following result could not be verified!)

References

Fig1: https://encrypted-tbn0.gstatic.com/images?q=tbn:ANd9GcRz-h6aZoDeYoMaTI7hdWs2uTYozeohZtg4wqJ5KVzEIwOp4GjZAnbaqV0QuXH6JXJUCjQ&usqp=CAU

Fig. 2: https://www.me.psu.edu/cimbala/me325web_Spring_2012/Labs/Drag/intro.pdf

Fig. 5: https://core.ac.uk/download/pdf/82407114.pdf

Equations:

https://www.openfoam.com/documentation/guides/latest/doc/guide-turbulence-ras-k-omega-sst.html#sec-turbulence-ras-k-omega-sst-model-equations

Papers:

- https://link.springer.com/content/pdf/10.1007/s10494-017-9859-1.pdf
- https://core.ac.uk/download/pdf/82407114.pdf
- https://mycourses.aalto.fi/pluginfile.php/485040/course/section/90043/Tutorial_sphere_openfoam.pdf