A detailed review of drag Force and lift force when fluid flow over rotating cylinders

G Sagar Kumar Achary, Asst.Prof Shakti Prasanna Khadanga

G Sagar Kumar Achary(Mechanical Engineering, College, GIET University)
Asst.Prof Shakti Prasanna Khadanga(Mechanical Engineering, Faculty, GIET University)
Corresponding author's email:20mthpte004.gsagarkumarachary@giet.edu,spkhadanga@giet.edu

Article Type: CFD MODELING OF NEWTONIAN FLOW OVER TWO ROTATING CYLINDERS IN SIDE BY SIDE ARRANGEMENTS

First Author's Full Name: G Sagar Kumar Achary
Highest Qualification: M.tech
Department: Mechanical Engineering
Post/Rank (If a student, provide course name and course year): M.tech ME, 2020-2022
Affiliation (College/University/Institute) with postal address: GIET University Gunupur, Rayagada, Odisha
email id:20mthpte004.gsagarkumarachary@giet.edu, sagarkumarchary@gmail.com
ORCID:

Second Author's Full Name: Shakti Prasanna Khadanga
Highest Qualification: PHD pursuing
Department: Mechanical Engineering
Post/Rank (If a student, provide course name and course year):
Affiliation (College/University/Institute) with postal address: GIET University Gunupur, Rayagada, Odisha
email id: spkhadanga@giet.edu
ORCID:

ABSTRACT

The thesis discusses how the hydrodynamics of flow are affected by the Reynolds number and rotational speed around bodies. The experiment considers the Newtonian fluid flow over two cylinders placed side by side, results obtained from the numerical simulation was carried out by combination of various parameters in the range of $0 \leq \alpha$ a(non-dimensional rotational velocity) ≤ 4, $10 \leq Re \leq 40$, G=4 (gap between cylinder / diameter). The domain size taken into account was 240m. The simulation is carried out on each of these variables using ANSYS 16.0. While Fluent is used for the remaining of the simulation, Workbench is used for the geometry-related tasks. At each of these combinations of Reynolds Number and angular velocity, the value of the drag coefficient and lift coefficient is determined.. The results of the analysis provide a clear understanding of the intricate interaction between the cylinders' drag and lift coefficients, rotational velocity, and Reynolds number.

Keywords: Reynolds number, dimensionless rotational velocity, Newtonian fluid ,Drag-lift Coefficient

1. Introduction

The problem statement - The current project involves a square domain with four sides measuring 240 meters each. Two meters on each side of the square's canter are set aside for two 1 m diameter cylinders. The viscosity and density of the fluid are kept constant while causing it to flow in the X direction at various Reynolds numbers. In this situation, the impact of gravity is not taken into account. The flowing fluid is supposed to have a density of 1000 kg/m3 and a viscosity of 0.001 kg/m-s. The fluid's Reynolds Number is changed from 10 to 40. The value of dimensionless rotational velocity is changed from 0 to 4 for each Reynolds Number value that is taken into consideration Workbench is used to generate the problem's geometry, which is then meshed. The remaining tasks, such as supplying beginning and boundary conditions, are completed in Fluent. As a result, ANSYS is used to determine the drag coefficient and lift coefficient for each value being examined. Tabulation is done and the values of the drag co-efficient and lift coefficient are observed to get an idea about the relation among the Reynolds Number, rotational velocity and the values of drag co-efficient and lift coefficient. The streamlines and vorticity magnitude is also plotted to visualize the fluid flow around the cylinders.

Newtonian fluid: - A Newtonian fluid is one whose stress vs. strain rate curve, when plotted, passes through the origin and is linear. It bears Sir Isaac Newton's name. It has a mathematical definition that is

Non-Newtonian fluid: - is a fluid whose flow characteristics are different from Newtonian fluids in any way. Most frequently, shear rate or shear rate history do not independently affect the viscosity (resistance to deformation or other forces) in non-Newtonian fluids. When dealing with such a fluid, the relationship between the shear stress and shear rate is unique and sometimes even time-dependent. Therefore, it is impossible to determine a constant coefficient of viscosity.

As a result, although viscosity is frequently used in fluid mechanics to characterize a fluid's shear properties, it may not be sufficient to describe non-Newtonian fluids. They can be investigated most effectively using a number of additional rheological properties that relate stress and strain rate tensors under a variety of flow situations, such as oscillatory shear or extensional flow, and which are measured using various instruments or rheometers. Tensor-valued constitutive equations can be used to study the properties more effectively. In the discipline of continuum mechanics, these equations are frequently used. Oobleck, flubber, cold caramel topping, silly putty, ketchup, and other items are examples.

Computational fluid dynamics :- It is known as computational fluid dynamics (CFD), and is a branch of fluid mechanics that uses numerical techniques and algorithms to solve and analyse fluid flow issues. Computers are used to do the computations necessary to model the fluid surface interaction as described by boundary conditions. Using high-speed supercomputers has the advantage of producing better solutions.

The Navier-Stokes equations serve as the essential foundation for practically all CFD issues. Almost any single-phase fluid flow is defined by these. By eliminating some of the factors that represent viscosity and result in the Euler equations, the Navier Stokes equations can be made simpler. More simplification can be achieved by eliminating the parameters that describe and produce all possible equations. The solutions to these equations can then be linearized to get the linearized potential equations when the simplifications have been completed.

ANSYS, Inc.:- It is a computer-aided engineering (or simply CAE) software company with its main office in Canonsburg, Pennsylvania, in the United States. Swanson Analysis Systems, Inc. was the initial name under which Dr. John A. Swanson launched the business in 1970.

ANSYS offers a comprehensive range of engineering simulation solution sets, providing access to virtually any field of engineering simulation that a design process requires.:-

- Simulation Technology Structural Mechanics Metaphysics Fluid Dynamics Explicit Dynamics Electromagnetic
- Workflow Technology ANSYS Workbench Platform High-Performance Computing Geometry Interfaces Simulation Process & Data Management

The current project work makes use of ANSYS 13.0, with Fluent 6.2.16 being utilised for further simulation and Workbench being used for geometry. At specific angular velocities for both the cylinder and the flowing liquid, the velocity and pressure profiles are visible. Thus, the drag coefficient is determined and then confirmed using the expected outcomes.

1.6 **Drag coefficient** :- The drag coefficient, often known as Cd, is a dimensionless variable used to measure an object's resistance to motion in a fluid environment like air or water. It is employed in fluid dynamics. A lower drag coefficient means the object will have less aerodynamic or hydrodynamic drag, which is used in the drag equation. A specific surface area is constantly linked to the drag coefficient..

Understanding the drag properties of objects in fluid movement is crucial, particularly for engineering design considerations. This understanding can be used to lessen the drag on vehicles, aircraft, and structures. The phenomenon of drag can be modeled and tested by simulating water flow over the item (using software such as ANSYS, Fluent, etc.).

1.7 Lift coefficient:- The lift coefficient, often known as CL, is a dimensionless variable that is used in fluid dynamics to measure the lift force that a body experiences as a result of the fluid density surrounding it. It is used in the lift equation, where a lower lift value indicates that, when the body is surrounded by fluid, there will be less lift force. The lift force is always acting perpendicular to the body.

Understanding the lift properties of objects in fluid flow is crucial, particularly for engineering design considerations. This knowledge can be used to decrease the lift for submarines, aircraft, and other vessels. Drag phenomena can be modeled (using programmes like ANSYS, Fluent, etc.

2. Research Methodology:

First the nature of the curve for the drag and lift coefficient is seen by varying the Reynolds number and keeping the dimensionless rotational velocity constant. The values are tabulated below and the graph is plotted

Table 4.1 – Values of Drag and lift Coefficient at α = 2 and different values of Re:-

	Upper cylinder		Lower Cylinder	
Re	C_D	C_L	C_D	C_L
10	2.76	-5.0	2.24	-6.85
20	1.86	-5.20	1.41	-6.62
30	1.47	-5.26	1.08	-6.42
40	1.24	-5.30	0.89	`-6.28

Table 4.2 values of Drag and lift Coefficient at α = 4 **and different values of Re:-**

	Upper Cylinder		Lower Cylinder	
Re	C_D	C_L	C_D	C_L

10	2.3	-11.7	1.07	-18.4
20	0.98	-13.47	0.57	-18.36
30	0.5	14.1	0.41	-18.28
40	0.40	-14.4	0.3	-17.9

Now these values of both the table are plotted and the graph obtained is as follows:-

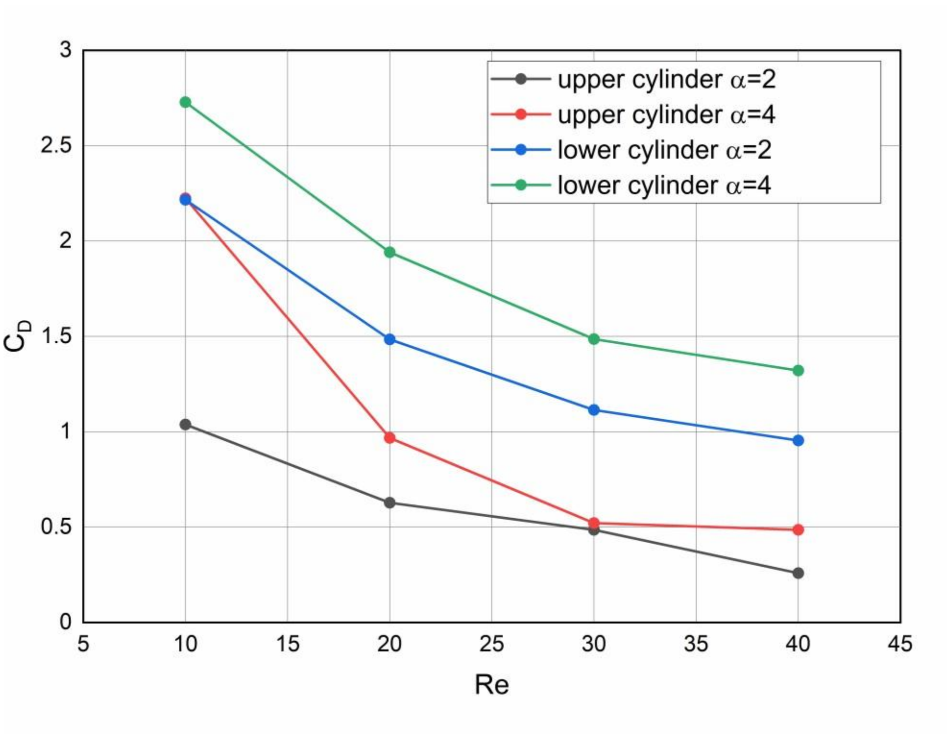

Fig. 4 C_D Vs Re graph

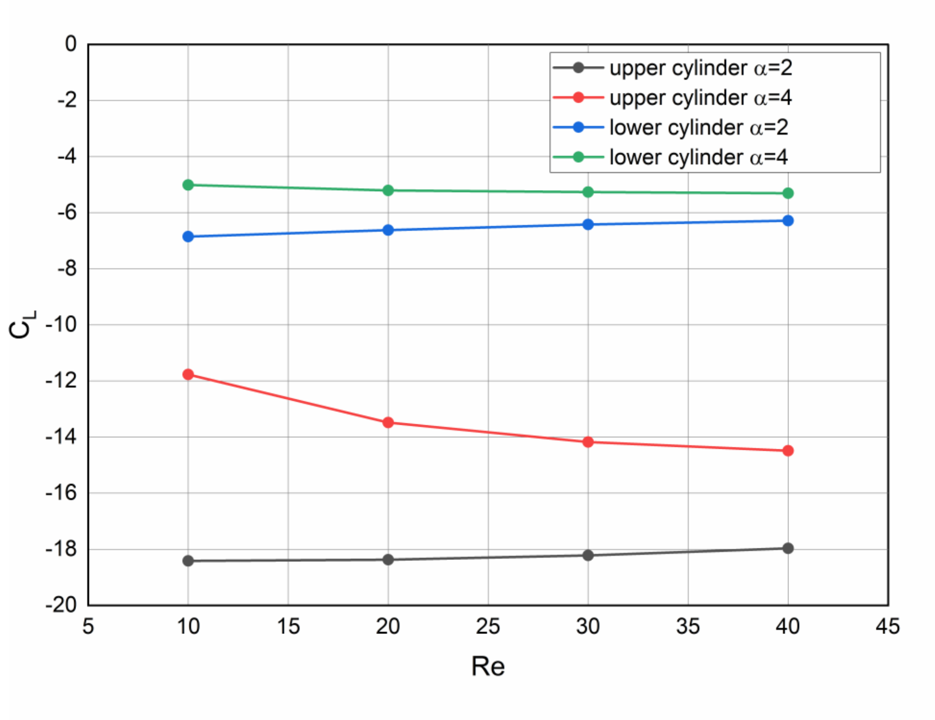

Fig. 5 C_L Vs Re graph

Following the plots, the Reynolds Number will remain constant, and we will then display how the angular velocity, i.e., raising the value of α affects the drag and lift coefficients (dimensionless rotational velocity).

Table 4.3:- Values of Drag and lift Coefficient and lift coefficient at Reynolds umber = 10 and different values of α:-

	Upper cylinder		Lower Cylinder	
A	C_D	C_L	C_D	C_L
0	2.9	0.32	2.97	-0.32
2	2.7	-5.01	2.24	-6.8
4	2.3	-11.7	1.07	-17.9

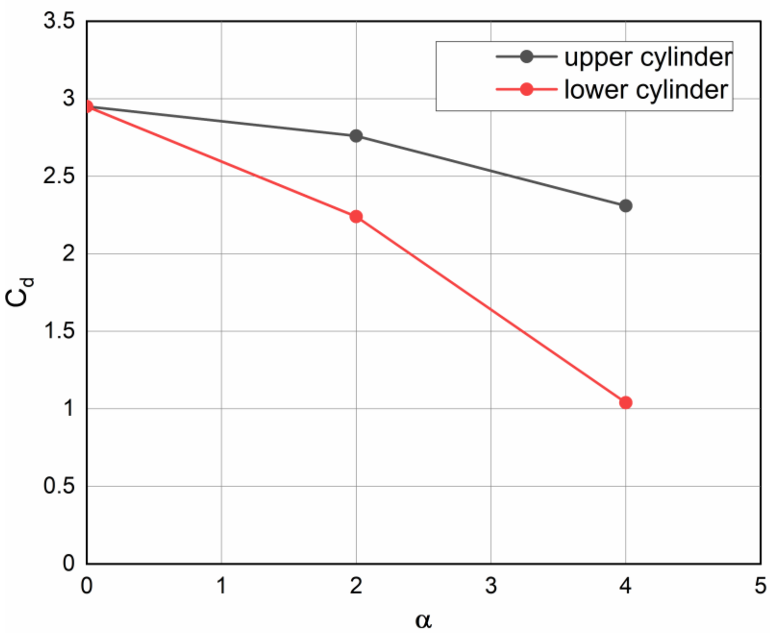

Fig. 6 C_d Vs dimensionless rotational velocity graph at Re = 10

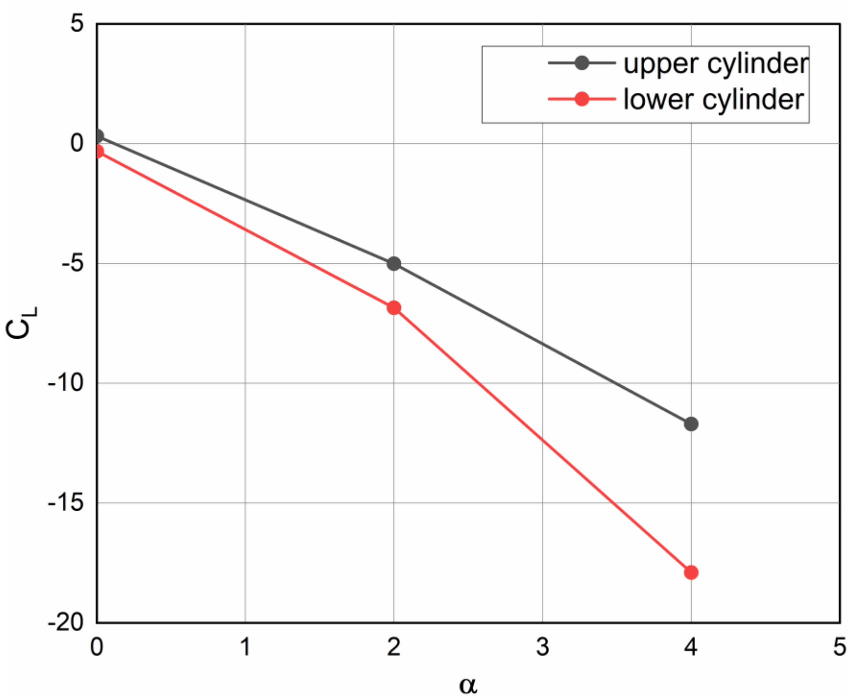

Fig.7 dimensionless rotational velocity graph at Re = 10

Now the value of Reynolds number is changed to 20 and the values of drag and lift coefficient are tabulated as follows:-

Table 4.4 – Values of Drag and lift CoefficieCnt and lift coefficient at Reynolds Number=20 and different values of α:-

	Upper cylinder		Lower Cylinder	
A	C_D	C_L	C_D	C_L
0	2.15	0.18	2.15	-0.18
2	1.86	-5.20	1.41	-6.62
4	0.98	-13.47	0.55	-18.30

The graph obtained for these values is plotted as follows:-

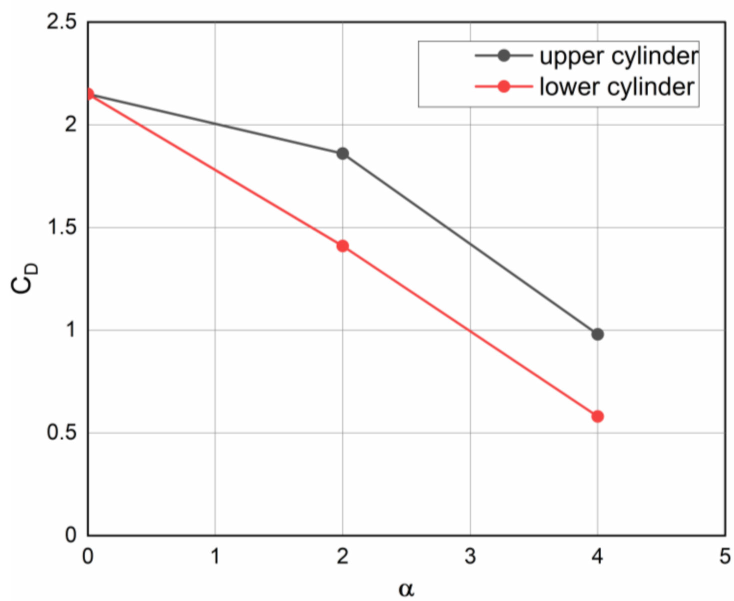

Fig. 8 C_d Vs dimensionless rotational velocity graph at Re = 20

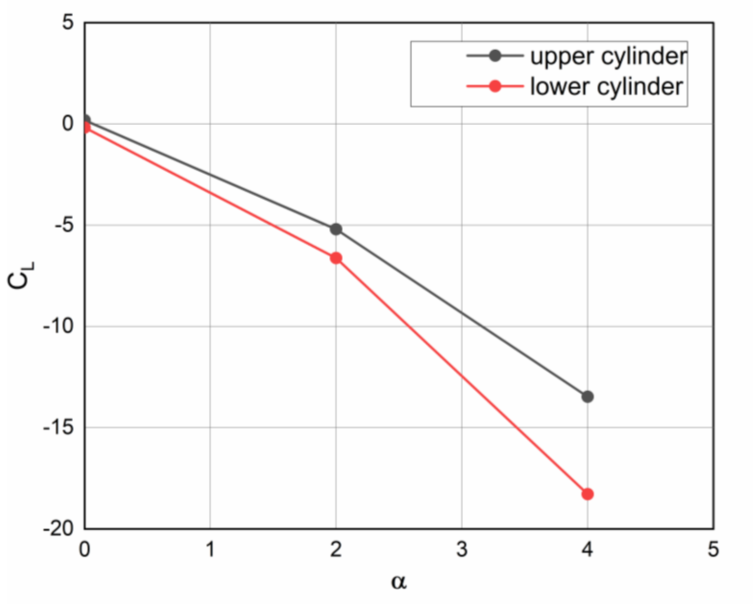

Fig.9 C_L vs dimensionless rotational velocity graph at Re=20

Now the value of Reynolds number is changed to 30 and the values of drag and lift coefficient are tabulated as follows:-

Table 4.5 – Values of Drag and lift Coefficient and lift coefficient at Reynolds Number =30 and different values of α :-

	Upper Cylinder		Lower Cylinder	
A	C_d	C_L	C_d	C_L
0	1.81	0.14	1.81	-0.14
2	1.47	-5.26	1.08	-6.48
4	0.56	-14.17	0.41	-18.3

The graph obtained for these values is plotted as follows

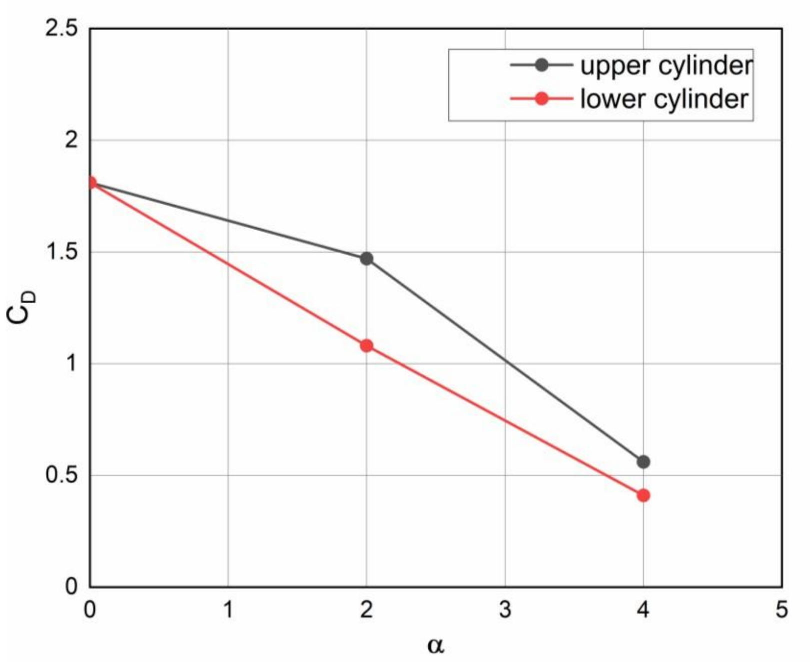

Fig. 10 C_d vs dimensionless rotational velocity graph a Re = 30

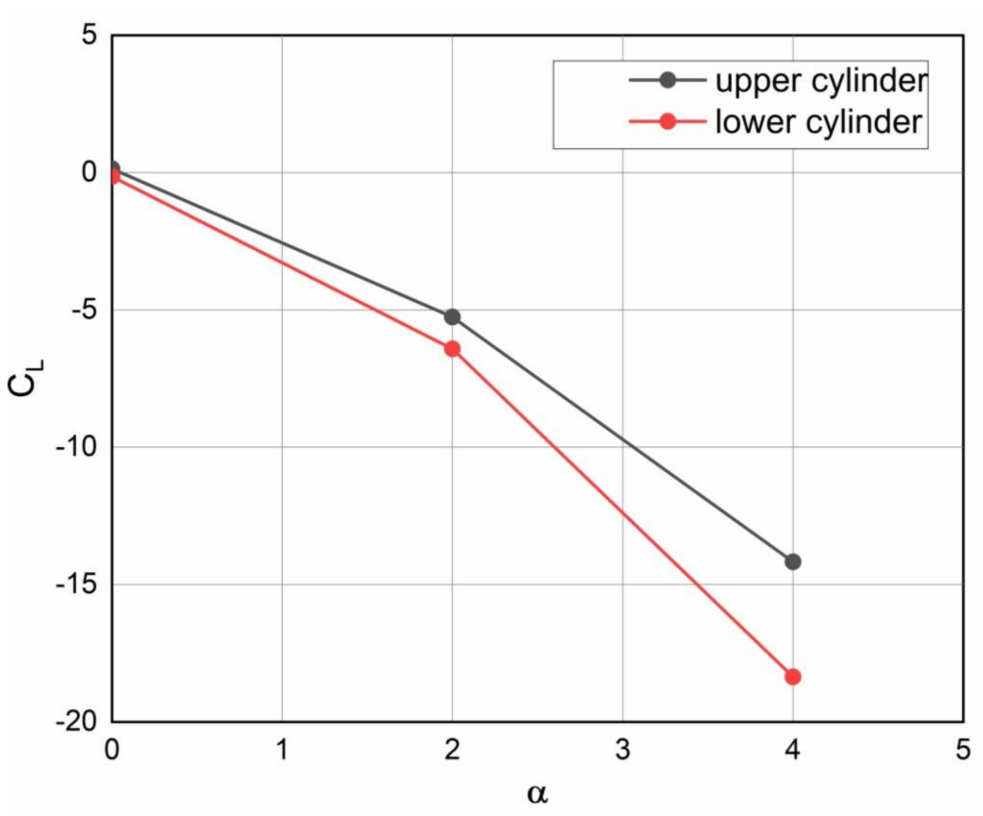

Fig.11 C_L Vs dimensionless rotational velocity graph at Re=30

Now the value of Reynolds number is changed to 30 and the values of drag and lift coefficient are tabulated as follows:-

Table 4.6 – Values of Drag and lift Coefficient at Reynolds Number = 40 and different values of α:-

	Upper Cylinder		Lower Cylinder	
A	C_D	C_L	C_D	C_L
0	1.62	0.12	1.62	-0.12
1	1.24	-5.30	0.89	-6.28
2	0.41	-14.49	0.3	-18.4

The graph obtained for these values is plotted as follows:-

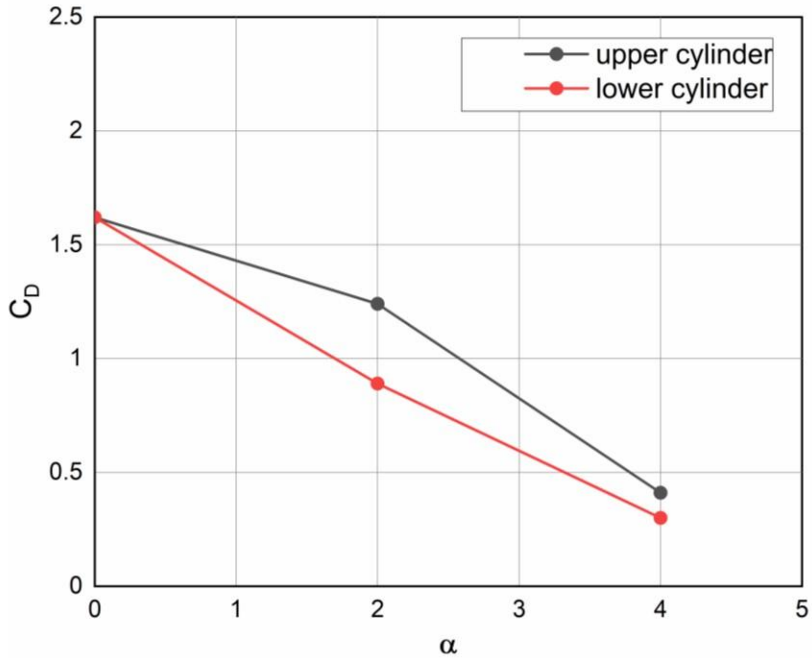

Fig.12 C_D Vs dimensionless rotational velocity graph at Re=40

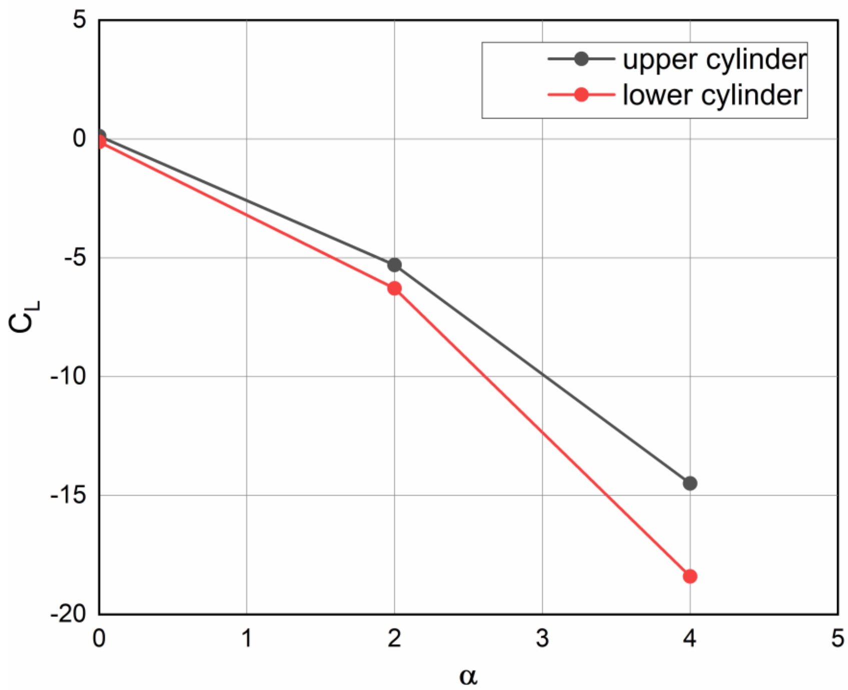

Fig. 13 C_L vs dimensionless rotational velocity graph at Re = 40

After analyzing the nature of the graphs against various parameters we get a clear knowledge on the effect of the speed of rotation on drag and lift coefficient of the cylinders.

CONCLUSION:

From the above simulation above the following conclusion can be drawn. With the increase in Reynolds Number from 10 to 40 and keeping the dimensionless rotational velocity constant, The drag coefficient of both the cylinders shows a constant decrease in its value. However the value of Reynolds Number increases, the value lift coefficient of upper cylinder decreases but the value of lift coefficient in lower cylinder slightly increases.

When the Reynolds number is kept constant and the value of dimensionless rotational velocity is increased both the drag and lift coefficient of both the cylinders shows a gradual decrease and the nature of the graph in all the cases is quite similar in nature. The reason behind this behavior of drag coefficient and lift coefficient is that Reynolds Number inversely on viscosity. More is the viscosity, more will be the area of contact between the fluid and the surface of the cylinder. Since the drag coefficient depends inversely on area of contact hence, more area results in lesser value of drag coefficient. Although the factor of velocity could also have been taken into concern as Reynolds Number depends directly on velocity and hence, depends directly on area of contact. But here, we have assumed our velocity and viscosity to be constant at. Hence a better relation between the drag coefficient, lift coefficient, area of contact, Reynolds Number and speed Of rotation could be understood.

The effect of rotational velocity is of much significant at all values, since it results in decrease of both drag and lift coefficient .The velocity profiles are found to be quite symmetric when the rotational velocity is at zero rpm i.e. the cylinder is static. Same is the case with the pressure profiles at zero rpm. More the value of angular velocity, more is the difference in fluid flow velocity created on the opposite sides of the cylinder. One of the sides gets its path smoother and easier, with the motion of the cylinder favoring its flow. While on the other side, the motion gets a bit hindered and hence velocity decreases. Thus, it can be concluded that with the increase in the angular velocity, the symmetry in the profile is lost.

From the streamlines figure it is concluded that at greater Reynolds number there is Significant wake formation. And as the rotational velocity is increased the wakes formed around the cylinders is decreased significantly.

References:

1. Mccormick, Barnes W. (1979): Aerodynamics, Aeronautics, and Flight Mechanics, John
2. Wiley & sons, Inc., New York, ISBN 0-471-03032-5, p. 24,

   Milne-Thomson, L.M. (1973). Theoretical Aerodynamics. Dover Publications. ISBN 0-48661980-x.

3. Batchelor G• K• An Intoduction to Fluid Dynamics. The Press Syndicate of the University of Cambridge , United Kingdom (1967)
4. Chaitanya, V., Parimal N., Ravindranath, G., Satish, K, B., Albert, K. Basics of computational fluid dynamics analysis (2003)
5. http://etheSiS•nirl.ac.in/2336/I/CFD Analysis of three phase fluidized bed.
6. http: /en.wikipedia.orgwiki/Computational fluid_dynamics
7. https://confluence.cornell.edu/display/SIMULATION/ANSYS+Taylor-Couette+Flow+between+Rotating+Cylinders+-+Physics+Setup
8. https://www.cfd-online.com/Forums/ansys-meshing/108833-rotating-cylinders-ansys.html
9. Reynolds Number, http:// engineeringtoolbox.com
10. Reynolds, Osborne. “An experimental investigation of the circumstances which determine whether the motion of water shall be direct or sinuous and of the law of resistance in parallel channels”. Philosophical Transactions of the Royal Society, Vol.174 (0) (1883), pp. 935-82.