To study Magnus Effect on Flettner Rotor

Mandar Gadkari[1], Varun Deshpande[2], Sumant Mahulkar[3], Vishal Khushalani[4], Shubham Pardhi[5], Dr. A.P. Kedar[6]

[1-5] (Final year B.E Mechanical Engineering Student, DBACER)
[6](Professor, Department of Mechanical Engineering, DBACER)
Dr. Babasaheb Ambedkar College
of Engineering & Research, Nagpur-441110

Abstract - *Magnus effect is used in many fields. This paper describes the approach taken to create a software model for the wind propulsion device flettner rotors (FR). For simulation purpose FR are considered as 2-D cylinders. Utilizing the advanced computer aided software ANSYS fluent to determine the fluid flow characteristics of cylinder's magnus effect. The behavioral study of the fluid flow on rotating cylinders is conducted. The simulation here computed as purely laminar boundary layer flow. The flettner rotor devices have been characterized in terms of lift and drag coefficient & this data is compared with the experimental observations available in literature.*

***Key Words*: Magnus effect, Cylinder, CFD, Coefficient of Lift & Drag.**

I.INTRODUCTION

FLETTNER ROTORS

Flettner rotors (FR's) are a form of a wind based propulsion system that utilizes the magnus effect shown by a spinning body in a fluid flow incident upon it. A FR typically comprises of a cylinder with an end plate affixed to it on the top. The rotating cylinder in an airstream generates a lift and a drag force that contributes to the propulsive need of the ship. The rotation to the FR is generally given by an electric motor.

Wind is a renewable energy source that is freely available on the world's oceans. As shipping faces the challenge of reducing its reliance on fossil fuels and cutting its carbon emissions this paper seeks to explore the potential of utilization of ocean wind energy for shipping.

Shipping faces an enormous challenge its fossil fuel consumption and ensuring CO2 emissions have grown over time but in the future they must be controlled and reduced [1]. For the past decades, however, international shipping emissions have

While there is scope for some improvement and optimization in operations, e.g. where market incentives lead to inefficient practices & it seems likely that technology will have to play a key role in amending the current shipping scenario. Wind power technology is certainly one attractive preference, wind is a free and renewable energy source that is available in oceans, and furthermore, it can be used in union with all other low carbon fuels. The focus of this paper is to assess the potential for wind-assisted shipping. A few studies have considered a fuel saving potential of flettner rotors (FR's) with the help of integrated software's using real time data increases the efficiency up to 8% per rotor [2].

MAGNUS EFFECT- Magnus effect is the phenomenon whereby a spinning body in a fluid creates a perpendicular force called lift and an opposing force drag.

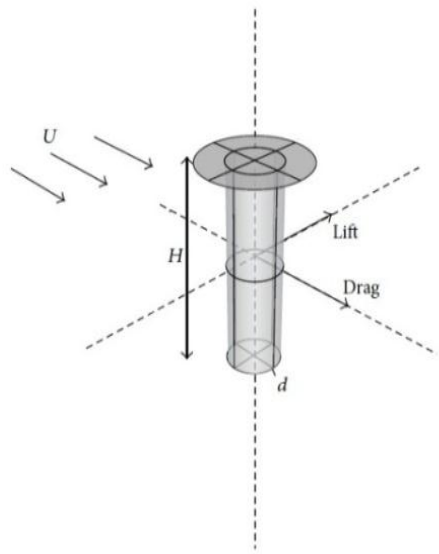


Fig.1 Forces acting on Flettner Rotor

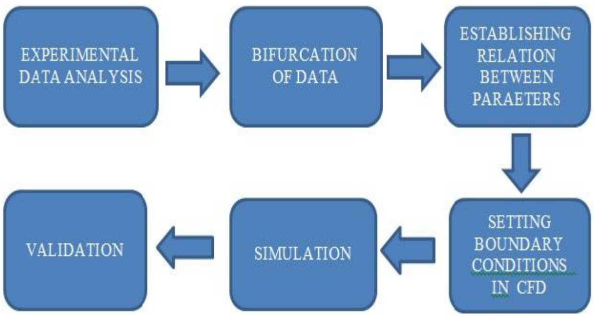
This forces acting on the cylinder provides the propulsion required to sail the ship in the ocean.The lift force vector direction is dependent on the rotor rotation direction [3]. By adjusting the rotation speed, the amounts of drag and lift can be varied.

II. NOMECLATURE

Symbol	Units
F_d	Drag Force (N)
F_l	Lift Force(N)
C_l	Coefficient of lift
C_d	Coefficient of Drag
α	Velocity ratio
U	Peripheral Velocity(rad/s)
V	Wind speed(m/s)
ρ	Density of Fluid(kg/m^3)
A	Area(m^2)
N	Revolutions per minute(rpm)
D	Diameter of Cylinder(m)

III. METHODOLOGY

1. **Experimental data Analysis**
2. **Bifurcation of Data**
3. **Establishing relation between Parameters**
4. **Setting boundary conditions in CFD**
5. **Simulation**
6. **Validation**



1.EXPERIMENTAL DATA ANALYSIS

The capability of infinite length rotating cylinders to produce aerodynamic forces was studied for the first time at the Langley NACA Laboratory by Reid, it was found that in particular conditions such simple devices are capable of developing very high values of the lift coefficient and of the aerodynamic efficiency (i.e., the lift-to-drag ratio) [4].

Extensive study done at Langley memorial on action of the rotating cylinders with axis perpendicular to the direction of flow helped us to procure the results of the wind tunnel test of rotating cylinders. In this experiment wind at various velocities (15, 10, 7, 5 m/s) are incident upon a cylinder rotating at wide range of revolutions per minute (RPM) thus giving coefficient of lift & drag. The study is based on the previous work done by the physicist Kutta-Joukowski, Prandlt .The sample observations are tabularized as follows:-

TABLE 1

RPM	Coefficient of Drag(C_d)	Coefficient of Lift(C_l)	Wind velocity(V)
25	0.925	-0.008	15
500	0.925	-0.008	15
900	0.835	-0.016	15
1020	0.766	-0.018	15
1115	0.693	-0.006	15
1240	0.632	0.002	15
1300	0.614	0.014	15
1400	0.602	0.122	15
1500	0.605	0.230	15
1600	0.605	0.326	15
1700	0.618	0.495	15
1780	0.611	0.537	15
1800	0.614	0.548	15
1900	0.616	0.650	15
2000	0.618	0.710	15
2080	0.622	0.706	15
2100	0.622	0.811	15
2200	0.622	0.873	15
2220	0.640	0.942	15
2300	0.628	0.967	15
2420	0.614	1.040	15
2500	0.604	1.089	15
2600	0.593	1.194	15
2620	0.589	1.060	15
2700	0.578	1.284	15

The governing equation for lift & drag forces are as follows [6]:-

$$F_d = \frac{V^2 \rho A c_d}{2} \quad (1)$$

$$F_l = \frac{V^2 \rho A c}{2} \quad (2)$$

2.BIFURCATION OF DATA & 3.ESTABLISHING RELATION BETWEEN PARAMETERS

It is seen in the above observations that more than two parameters are effecting C_d & C_l . Therefore in order to generalize and establish a direct relation between governing parameters velocity ratio is used (α) which is ratio of peripheral velocity (U) to wind velocity (V).

$$\alpha = \frac{U}{V} \quad (3)$$

TABLE 2:- represent the direct relation between velocity ratio(α), and coefficient of lift and drag.

RPM	Velocity Ratio (α)	Coefficient of Drag(C_d)	Coefficient of Lift(C_l)
25	0.010	0.925	-0.008
500	0.200	0.925	-0.008
900	0.360	0.835	-0.016
1020	0.408	0.766	-0.018
1115	0.460	0.693	-0.006
1240	0.496	0.632	0.002
1300	0.520	0.614	0.014
1400	0.560	0.602	0.122
1500	0.600	0.605	0.230
1600	0.64	0.605	0.326
1700	0.68	0.618	0.495
1780	0.712	0.611	0.537
1800	0.720	0.614	0.548
1900	0.760	0.616	0.650
2000	0.800	0.618	0.710
2080	0.832	0.622	0.706
2100	0.840	0.622	0.811
2200	0.880	0.622	0.873
2220	0.888	0.640	0.942
2300	0.920	0.628	0.967
2420	0.968	0.614	1.040
2500	1.00	0.604	1.089
2600	1.040	0.593	1.194
2620	1.048	0.589	1.060
2700	1.080	0.578	1.284

GRAPHS

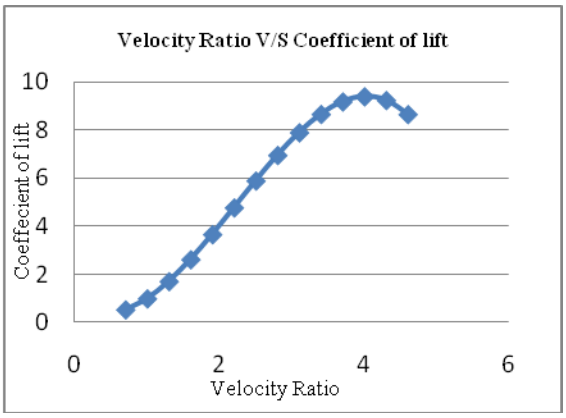


Fig.3 curve of coefficient of lift

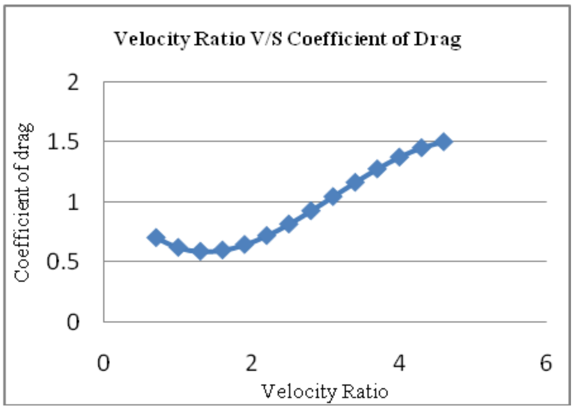


Fig.4 curve of coefficient of Drag

IV. INTRODUCTION TO CFD

Computational Fluid Dynamics (CFD) is a division of fluid mechanics that uses numerical analysis and complex fluid equations to solve and analyze problems that involve fluid flows. The calculations required to simulate the interaction fluid with surfaces at certain boundary conditions are performed using a computer.

CFD analysis helps in optimizing the design cycles. In addition improvement in equipments are built and installed in a minimum downtime. The main benefit of CFD is that prototyping can be done in a much faster way. It also provides us with the better details. In this paper the CFD software is used to study the behaviour of fluid flow around the rotating cylinder.CFD finds its applications in Aerospace , Automobile , Biomedical , Chemical processing , Hydraulics , Marine , Oil & Gas , Power Generation , Sports , Pollutant monitoring and many more .

4.SETTING BOUNDARY CONDITION

a. Meshing

Specific type of triangular fine meshing was given to the surrounding area and Inflation type of meshing was given in a circular pattern around 2-D cylinder, to get a smooth image of the fluid flow [7].

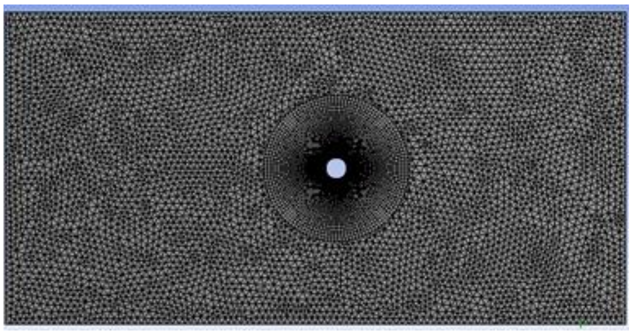


Fig .5 shows Meshing around 2-D Cylinder

5.SIMULATION

In the CFD simulation, the boundary condition are specified in such a way that at inlet a uniform velocity is prescribed & on the surface of the cylinder a non-slip wall is arranged, also the curvature of cylinder is provided with moving walls. Various other aspects & boundary conditions were well defined to get accurate results .The simulation here computed as purely laminar boundary layer flow.

Simulation (a) for velocity contours on steady cylinder

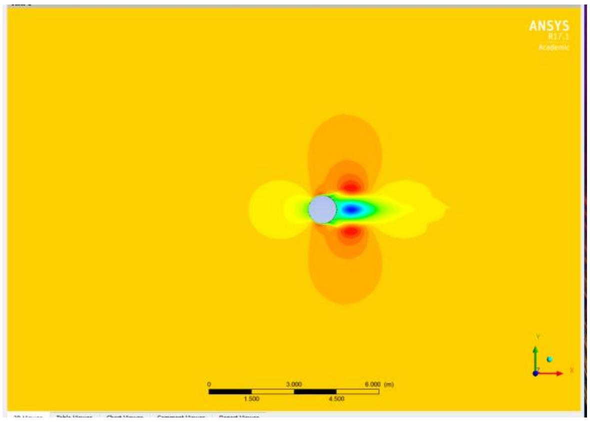


Fig.6 represents Stationary cylinder with fluid inlet speed 7m/s

Simulation (b) for pressure contours on steady state cylinder.

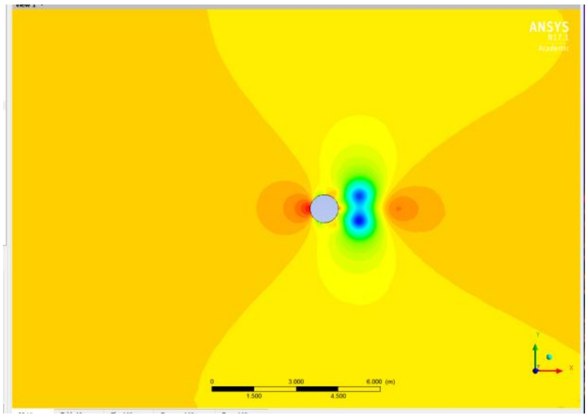


Fig.7 represents Stationary cylinder with fluid inlet speed 7m/s

Simulation (c) for velocity contours on rotating cylinder.

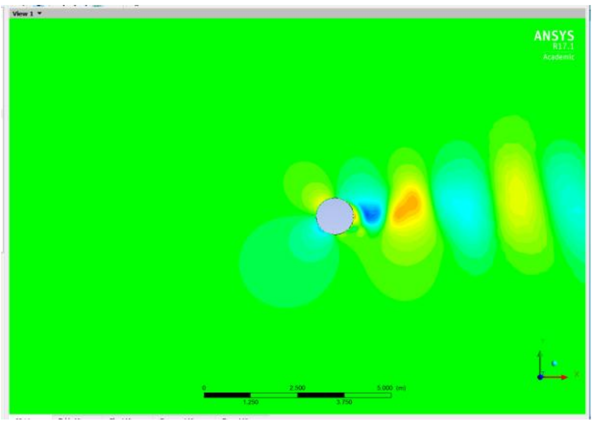


Fig.8 represents Rotating cylinder with fluid at inlet

Simulation (d) for pressure contours on rotating cylinder

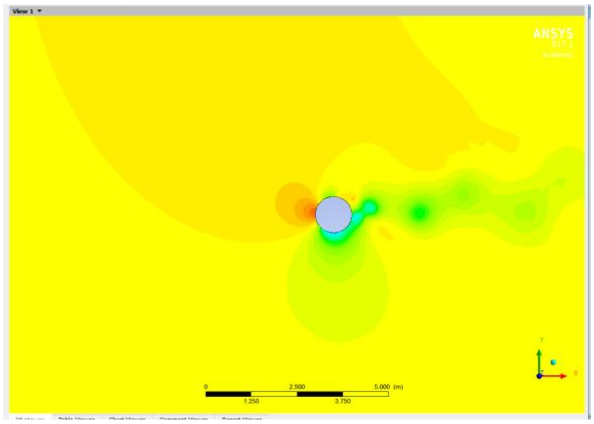


Fig.9 represents Rotating cylinder with fluid at inlet

Simulation (e) for Stream Lined Simulation of rotating cylinder

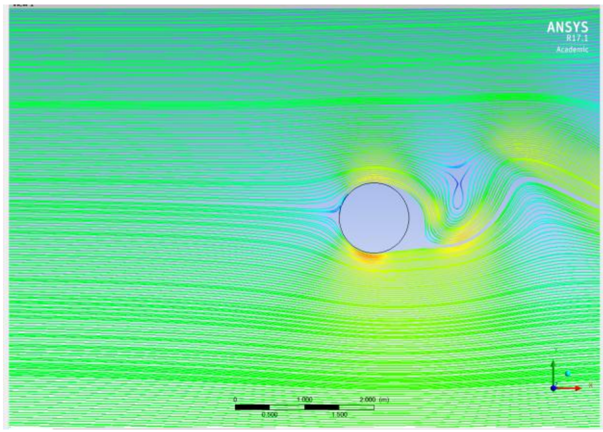


Fig.10 represents rotating cylinder at wind speed 15m/s

In the above simulation the pressure, velocity Contour and the streamline projection of the cylinder is shown is Fig.6,7,8,9,10. Does the effect of Magnus forces can be seen in simulations. The steady cylinder shows a symmetric simulation image, where as in rotating cylinder simulation phenomenon known as vortex shredding was observed .

Simulation (f) to get coefficient of lift & drag on the cylinder.

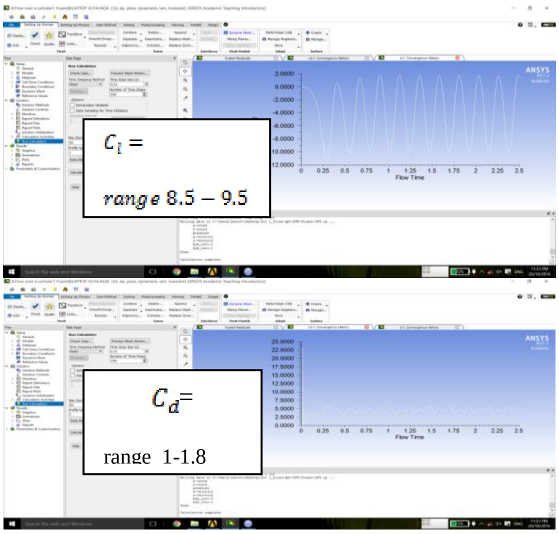


Fig.11 has velocity ratio α=3.727

V. CONCLUSION

In this paper, the analysis results are based on computational fluid dynamics (CFD). The behavioral study of fluid flow around rotating cylinder and steady cylinder is depicted and are represented on simulated figures. Also the coefficient of lift and drag is determined against velocity ratio (α) using CFD. The analysis of the experiments reported, do not give any solutions but contribute to the discussion o the possible benefits of rotor powered ships.

REFERENCES

1. Tannja Suominem, "A thesis submitted for degree of Bachelor of Marine Technology", Rotor pilot project on M/S Eastraden of bore fleet", pp.9-30.

2. Williams Ake, Liljenberg Hans, "Revival of flettner rotor-Beneficial or not for merchant vessels, fishing boats and recreational craft?", pp. 125-127.

3. Zhao Jie, Qingming Hou, Hongzhe Jin, Jihong Yan, Yanhe Zhu, Ge Li, "Discussion on improving Magnus effect of cylinder based on CFD", International conference on mechatronics and automation, pp.539-543.

4. Elliott G. Reid, "tests of rotating cylinders", national advisory committee for aeronautics, NACA-TN-209.

5. http://en.wikipedia.org/wiki/Magnus_effect.

6. Pearson D. R, "The use of flettner rotors in efficient ship design" , BMT Defence services Ltd, UK, pp. 4-6.

7. Middendorf John, "CFD modeling of wind tunnel flow over a rotating cylinder", pp.1-26.