# CFD Analysis of wind Turbine Blade Suitable for Wind Tree, Modular Tree and Bush Tree.

[1]Alfaz. A. Mutwalli, [2] Nilesh Desai

[1]Student, Department of Mechanical Engineering, Sanjay Ghodawat University.

[2] Professor, Department of Mechanical Engineering, Sanjay Ghodawat University.

---***---

**Abstract -**Wind power and renewable energy is in greater demand in present and also in future upcoming years. This is due to environmental impact. As we all know wind is freely available in nature those can give rise to electricity. As in present era we get electricity bill in a very high cost. So to overcome these challenge wind tree, modular tree and bush tree can be installed residential as well as commercial area. Horizontal axis it require more space so it is quite difficult to install. So the best technique is to install is vertical axis wind turbine. At the time of installation of vertical axis wind turbine we don't know which turbine will be suitable and power output, because electricity generation depends on wind speed and those winds are captured by wind turbine blade. those wind turbine are attached to wind tree, modular tree and bush tree in areas such as roads, hospital, college, residential areas, industrial area, etc. the paper addresses the wind turbine blade suitable for wind tree, modular tree and bush tree by computational fluid dynamics (CFD) using ANSYS fluent.

***Key Words*****:** Wind Turbine blade, wind tree, modular tree, bush tree, Computational fluid dynamics.

## 1. INTRODUCTION

Due to increase in pollution, fossil fuel and rise in electricity bill charges, there is huge demand for renewable resource and alternate method. Pollution affects both in rural and urban areas. Poor people who are unable sustain the electricity bill are suffering the most. This also effects overall economic cost.

Wind turbine is one which do not cause hazard to the environment. it is the lowest global warming potential per unit of electrical power generation as compared to the other sources. As wind is freely available in nature various research are being conducted so that wind power generation gets a better boost in the society. There are two types of wind turbine that re horizontal axis wind turbine and vertical axis wind turbine but in such cases Horizontal axis wind turbine require larger area of installation. So it is not suitable in residential area. So was mainly focus on vertical wind turbine blade. That can be installed on wind tree, modular tree and bush tree.

The concept behind using wind tree, modular tree and bush tree is that currently various cities faces energy crisis and looking for enhancing renewable energy source and finding the alternative techniques, so that their electricity bill charges can be reduced and power output can be available freely. The wind tree, modular tree and bush tree captures all types of wind from general breezes to massive gusts whether turbulent or laminar in both urban and rural areas. The difference between all the types are wind tree requires more number of wind turbine blades; modular tree requires less as compared to wind tree and bush tree requires less as compared to modular tree. By installing those trees as per requirement we will get more power output by using wind energy with less installation area that can be installed in residential area too. Another asset is that we will be having less weight, smaller in size and can use wind energy i.e. freely available in nature. We also find difficulties whether which turbine blade we should use so that it captures maximum amount of wind energy .so we have carried the simulation by CFD using ANSYS Fluent so that we can get an idea about the power generation. Also by using this technique we boost the renewable energy source and reduce the use of non renewable source.

## 2. METHODOLOGY

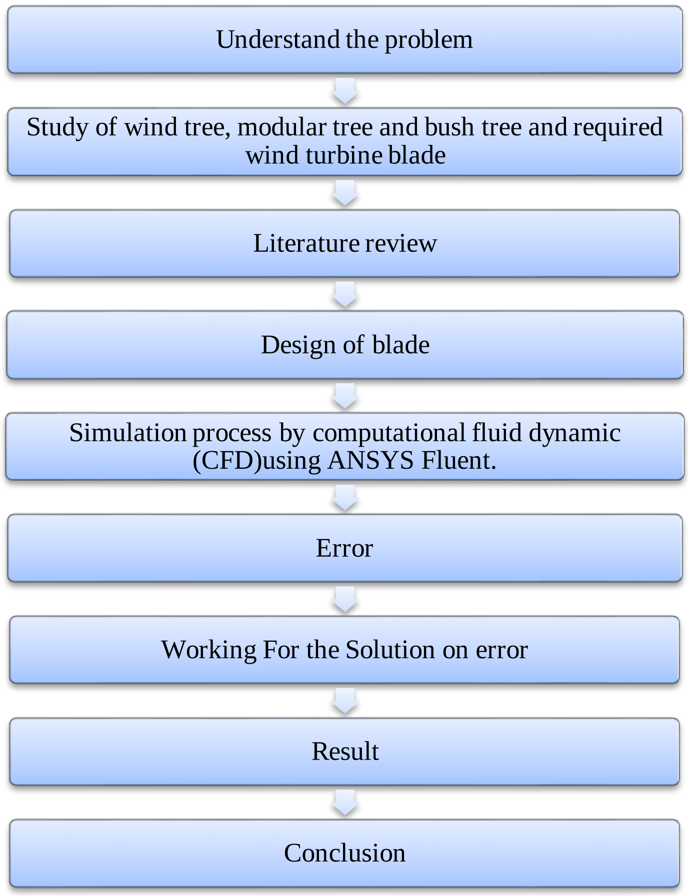


**Figure1:** Methodolgy of CFD Analysisising ANSYS fluent of wind turbine blade

Figure1 indicates the methodology of CFD Analysis using ANSYS Fluent of wind turbine blade.

- We understood the problem, environmentalhaard, increse in electricity bill charges issues.
- Studied Wind tree, Modulartree and Bush tree which are goodin appearance and require less space forinstallation sothat it can also be installed inresidential purpose.
- We studied various researchpaperrelatedtowind turbine of CFD Analysis using ANSYS Fluent. We aslo studied the problem or disadvantages those research paper are facing and made solution on it.
- We designed geometry drawing which is provided by design modeller by ANSYS.
- After designing the balde, we simulated those blades to rule by pressure,velocity and airflow.
- But during during the running we find some error and simultaneouly we worked on it.
- Then we simulated again hence, we found no error.
- We get the result which are analyed by pressure,velocity and airflow.

## 3. RESULTS

### 3.1) PART 1

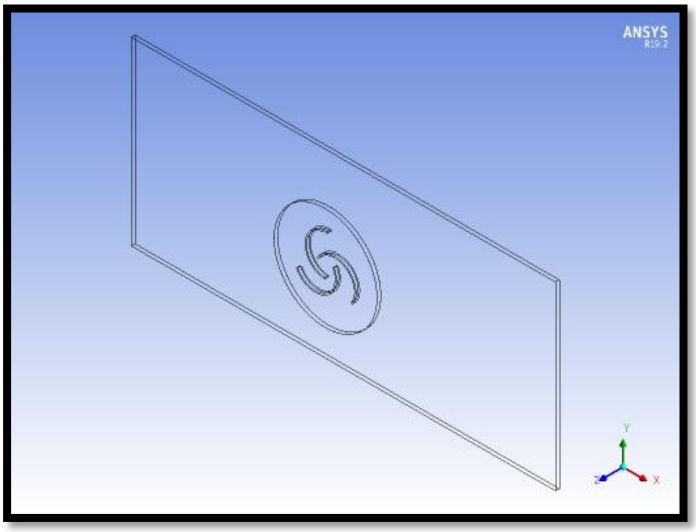


**Figure2:** Isometric View

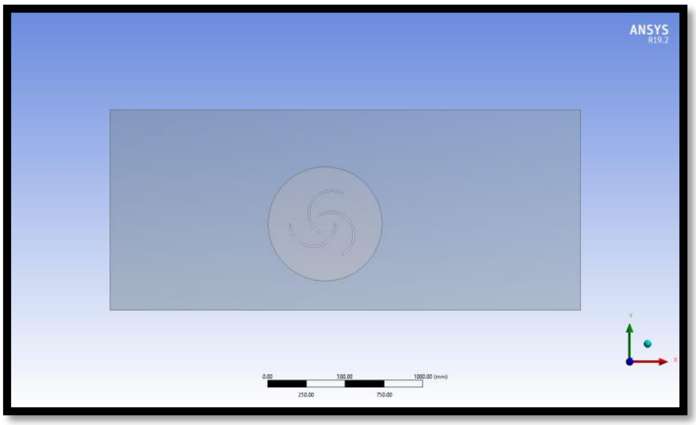


**Figure3:** Top View of 2 dimension wind turbine blade

**Figure 3** indicates the 2 dimension top view of wind turbine blade after meshing the geometry then we go for the meshing process.

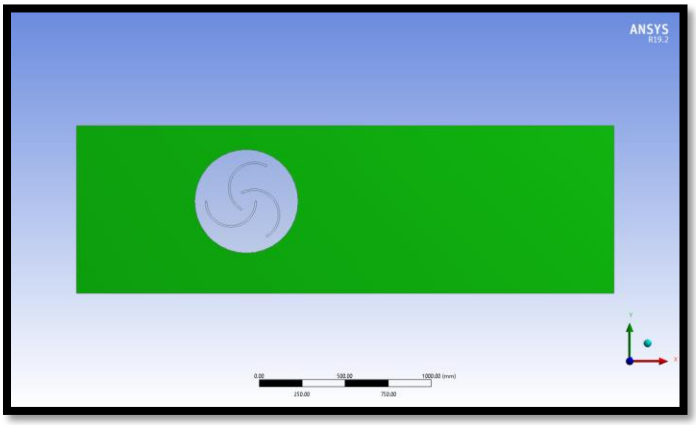

**Figure4:** Meshing part Exterior

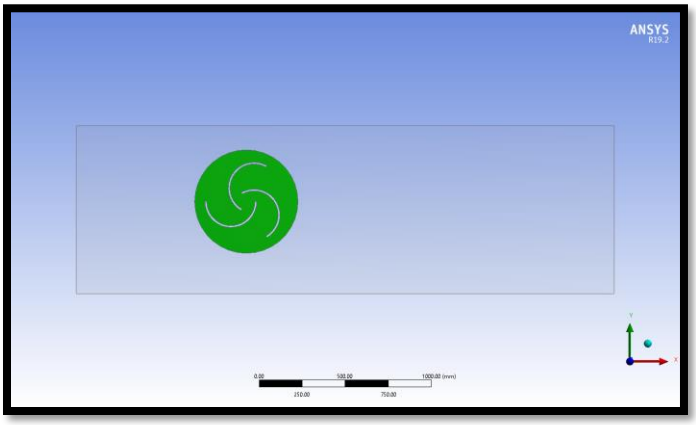

**Figure5:** Meshing part sketch Interior

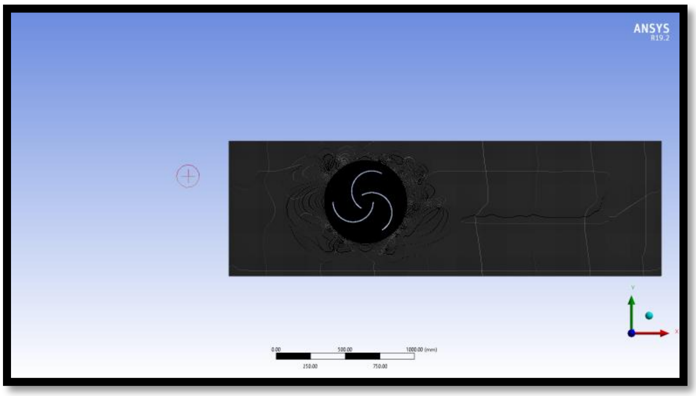

**Figure6**: Meshing of the geometry.

**Figure6.** Shows the meshing of the geometry. After meshing we go to setup to find the results.

**Table 1:** Mesh Information

| Domain | Nodes | Elements | Wedges |
|---|---|---|---|
| exterior | 18096 | 17549 | 17549 |
| interior | 9124 | 8618 | 8618 |
| All Domains | 27220 | 26167 | 26167 |

**Table 2:** Mesh Statistics

| Domain | Minimum Face Angle | Maximum Face Angle | Maximum Edge Length Ratio | Maximum Element Volume Ratio | Connectivity Range | |
|---|---|---|---|---|---|---|
| exterior | 15.0005 [ degree ] | 117.709 [ degree ] | 3.80669 | 6.63888 | 1 | 10 |
| Interior | 30.1579 [ degree ] | 99.3271 [ degree ] | 4.75858 | 3.11948 | 2 | 8 |
| All Domains | 15.0005 [ degree ] | 117.709 [ degree ] | 4.75858 | 6.63888 | 1 | 10 |

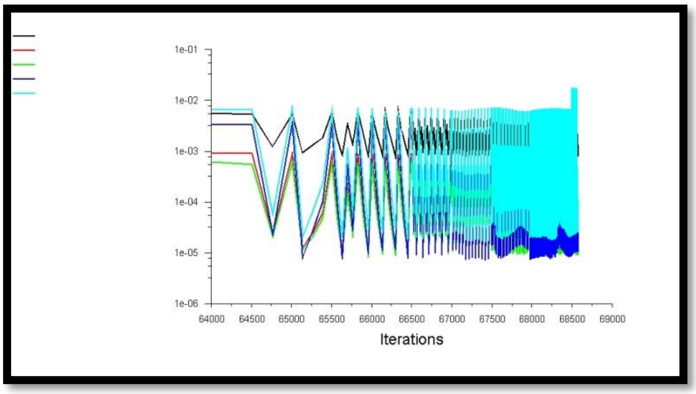


**Figure7:** Iteration we performed.

**Figure 7** shows the iteration which we performed after running the calculation. After iteration gets complete, we get to see the velocity and pressure contours.

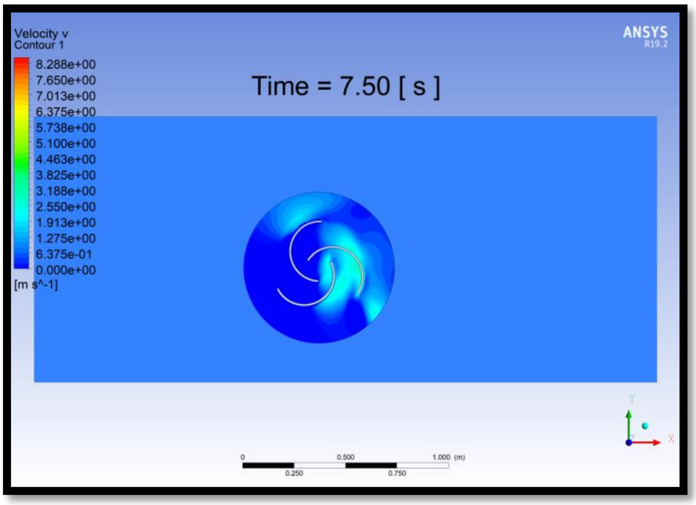


**Figure8**: Velocity contours

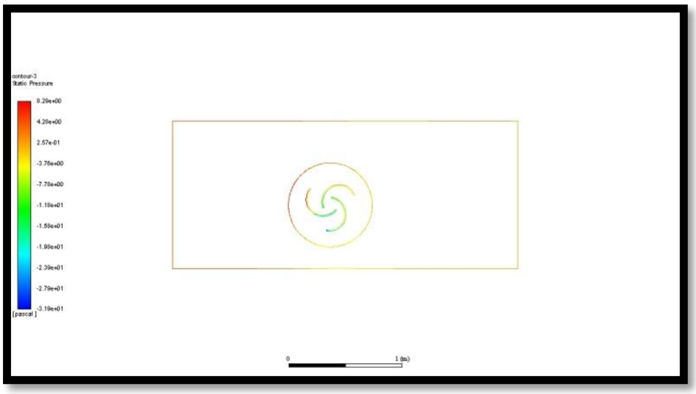

**Figure9:** Pressure contour

In the above simulation process we got to know that,

Considering **Figure8**

| | |
|---|---|
| Input speed | 2.5m/s |
| Turbine rotates at minimum speed of | 0 m/s to 8.5 m/s. |
| Turbine minimum pressure | −31 pa |
| Turbine maximum pressure | 82 pa |
| Pressure on each side of turbine | Not same |

| Upper Blade | Lower Blade |
|---|---|
| High pressure | low pressure |
| Direct contact wind so wind pressure is more | No Direct contact wind so wind pressure is less. |
| Comparing 8 and 9: Hence, is more velocity of fluid leads to less fluid pressure | |

### 3.2 PART 2

### H Rotor Darrieus Wind Turbine

| Average of Facet Values | |
|---|---|
| Velocity Magnitude (m/s) | |
| Outlet 1 | 100 |
| Outlet 2 | 157.89723 |
| wall-rotating body | 0.27340809 |
| Net | 1.2484478 |

| Average of Facet Values | |
|---|---|
| Tangential Velocity (m/s) | |
| Outlet 1 | -0.25845662 |
| Outlet 2 | -5.5996394 |
| wall-rotating body | 0.27340809 |
| Net | 0.24933901 |

| Mass Flow Rate (kg/s) | |
|---|---|
| Outlet 1 | 330.75 |
| Outlet 2 | -330.75 |
| Net | 2.3245814e-06 |

| Mass-Weighted Average | |
|---|---|
| Velocity Magnitude (m/s) | |
| Rotating body | 0.27569444 |
| Net | 0.27569444 |

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 06 Issue: 04 | April - 2022 Impact Factor: 7.185 ISSN: 2582-3930

| Mass-Weighted Average | |
|---|---|
| Tangential Velocity (m/s) | |
| Rotating body | 0.27432722 |
| Net | 0.27432722 |

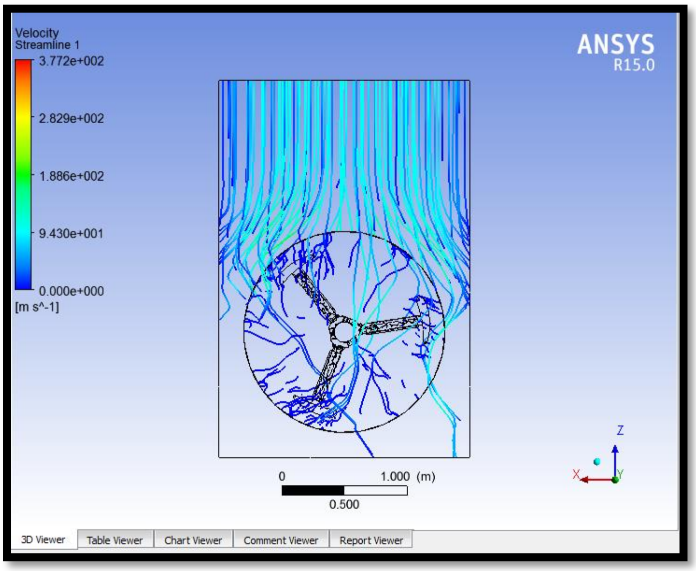


**Figure10.** Velocity streamlines 1

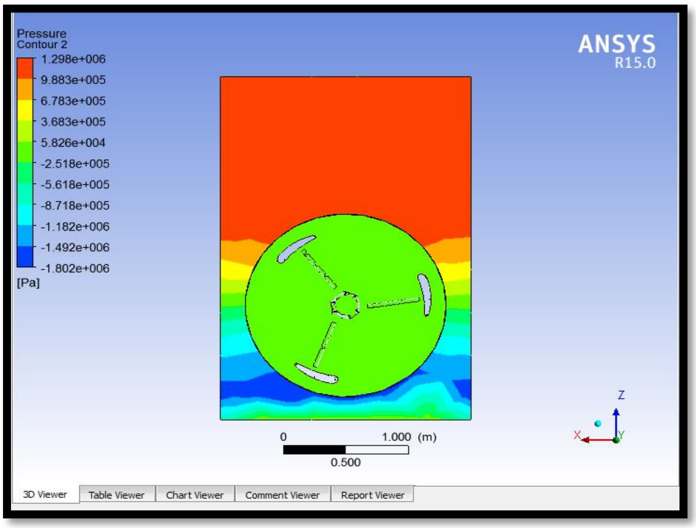


**Figure11.** Pressure contour 2

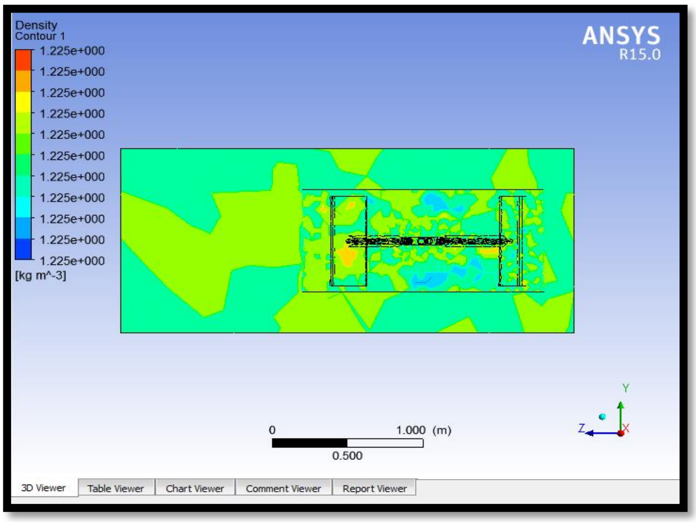


**Figure12.** Density contour 1

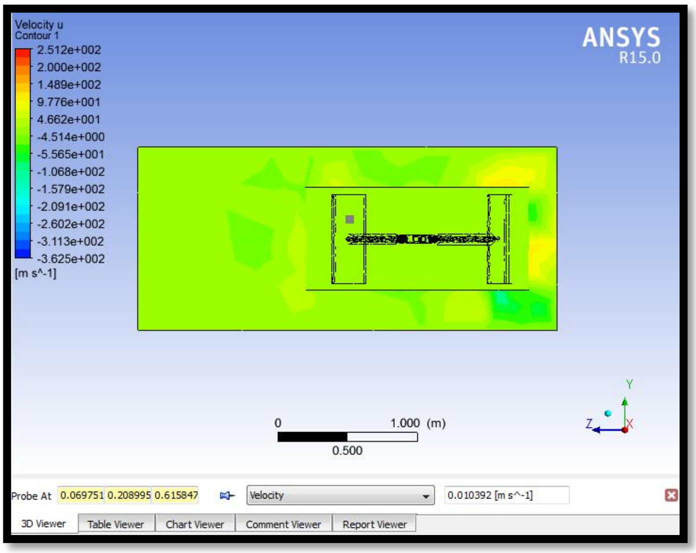


**Figure13.** Velocity contour 1

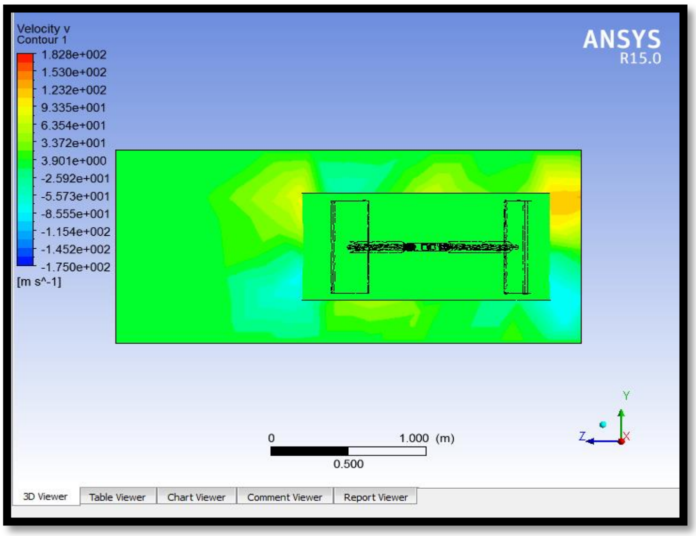


**Figure14.** Velocity contour 1

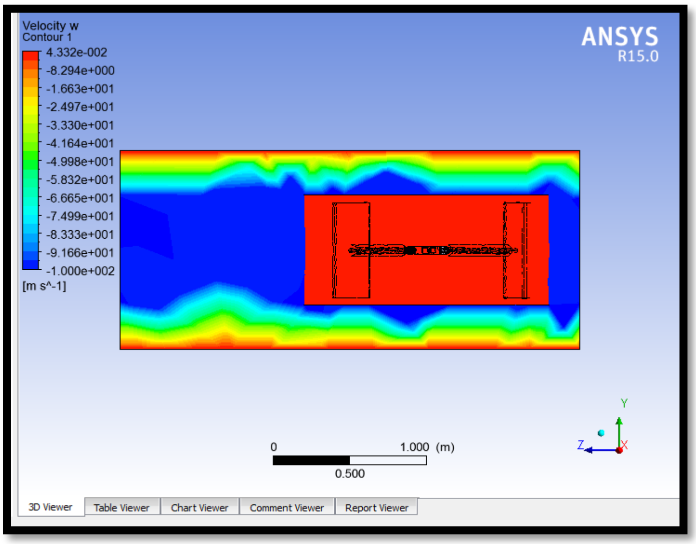


**Figure15.** Velocity contour 1

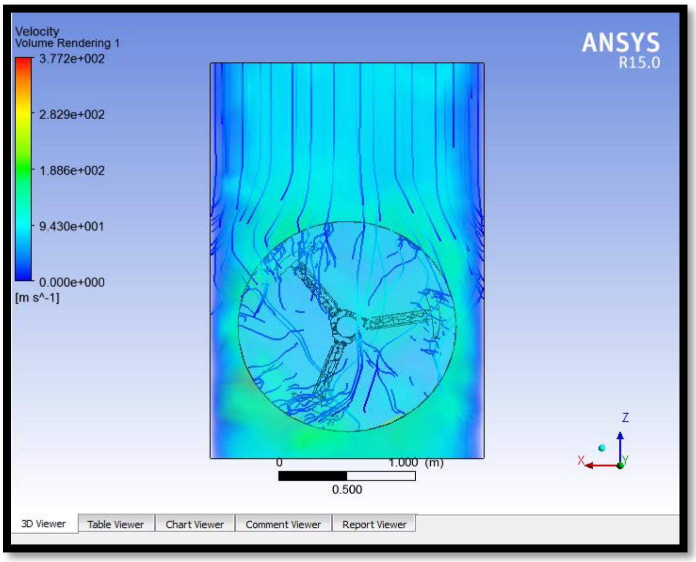


**Figure16.** Velocity Volume Rendering 1

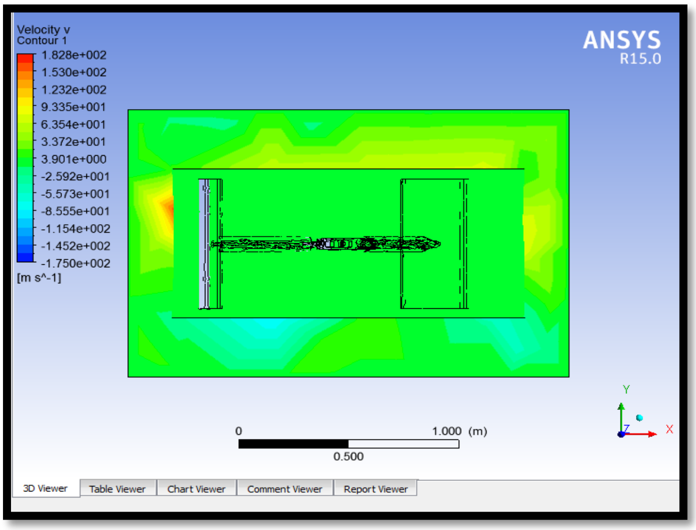


**Figure17.** Velocity contour 1

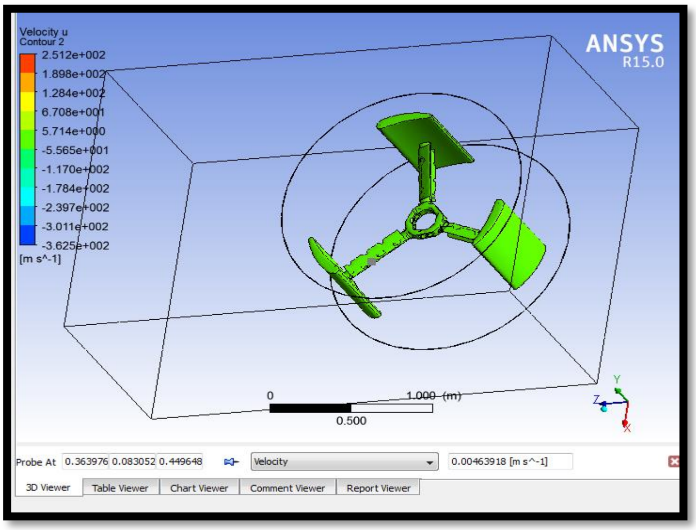


**Figure18.** Velocity contour 2

## 3. CONCLUSIONS

Manufacturers as well as costumer will face much issue during the installation process of wind turbine as there is lack of renewable energy source in use. We don't know which turbine will be suitable for particular wind velocity. So it is mandatory to first know that particular wind turbine is best for wind velocity.

By using CFD Analysis using ANSYS fluent we got the result and an idea that wind turbine blades can be installed on a wind tree, modular tree and bush tree to get maximum result and power output rather than before manufacturing the turbine blades first without knowing wind velocity, installation area and simulation on Computational fluent dynamic (CFD) Analysis. So by using these techniques we can conclude the blades which are required to install on wind tree, modular tree and bush tree

## ACKNOWLEDGEMENT

I also express my deepest respect and gratitude to our guide Mr. N. Desai for his valuable guidance, inspiration and encouragement. His keen and indefatigable in this project work helped me to reach to an irreproachable destination.

## REFERENCES

1. M. R. Castelli, A. Englaro and E. Benini, "The Darrieus wind turbine: Proposal for a new performance prediction model based on CFD," 2011.
2. J. Mccosker, "Design and Optimization of a Small Wind Turbine", 2012.
3. M. H. Ali, "Experimental Comparison Study for Savonius Wind Turbine of Two & three Blades at Low Wind Speed", 2013.
4. M. S. H. a. S. K. Afaq, "Design and analysis of a straight bladed vertical axis wind turbine blade using analytical and numerical techniques, 2013.
5. K. Suffer, "Modeling and Numerical Simulation of a Vertical Axis Wind Turbine Having Cavity Vanes", 2014
6. A. M. Chowdhury, H. Akimoto and Y. Hara, "Comparative CFD analysis of Vertical Axis Wind Turbine in upright and tilted configuration," 2015.
7. A. Alaimo, "3D CFD Analysis of a Vertical Axis Wind Turbine," 2015.
8. A.Afif, "CFD Analysis of vertical axis wind turbine using ansys Fluent", 2020.