International Journal of LNCT
Volume-2 Issue-8 October-2018
ISSN: (Online):2456-9895

# CFD Analysis of Wind Flow Over Wind Turbine

**Rajendra kumar Dewangan[1] , Prof. Sachin kumar Nikam[2]**

**M.Tech Scholar, Departmental Of Mechanical Engineering , LNCT Bhopal, M.P. India[1]**
**Department of Mechanical Engineering , LNCT Bhopal, M.P. India[2]**

**Abstract-** One of the most abundant sources of renewable energy is wind. Today, a considerable amount of resources are being utilized for research on harnessing the wind energy efficiently. Out of all the factors responsible for efficient energy production, the aerodynamics of flow around the wind turbine blades play an important role. This work aims to undertake aerodynamic analysis of a Horizontal Axis Wind Turbine. A steady state, incompressible flow solver for multiple reference frames, MRF Simple-Foam is modified and used for performing simulations of flow over National Renewable Energy Laboratory Phase VI wind turbine rotor. The code is first tested on a locally modeled wind turbine blade and is then validated by using the actual NREL rotor. The flow behavior is studied and a comparison of results from the simulations and the experimental wind tunnel data is presented. The ability of Computational Fluid Dynamics (CFD) techniques in simulating wind flow over entire wind turbine assembly is also displayed by carrying out moving mesh simulations of a full wind turbine.

**Keywords-** CFD Simulation, Coefficient of pressure , Comparison, Experiment Measurement, Wind Turbine, Hybrid Model.

## 1. Introduction

The energy resources are decreasing at a high rate all over the world which is a cause of concern for mankind. Consumption of these energy resources (especially the fossil fuels) is the reason for fast change in global climate. As per EIA (The Energy Information Administration), burning of fossil fuels contribute to 86% of the total production of energy. Millions of years are required for the formation of fossil fuels, but these are being consumed at a much faster rate, because of the increase in energy needs of the world. Such high rate consumption of fossil fuels will lead to a time when these resources will no more exist. Checking out few mathematical figures might help in understanding the problem. According to a prediction in a report from U.S. Department of Energy, the age of oil reserve is around 43 years and of gas is 167 years and coal would last up to 417 years only. To overcome the dependency on these non-renewable and exhaustible resources of energy, attention must be given to renewable sources of energy. Also burning of fossil fuels releases very harmful gases in the atmosphere and is the main cause of air pollution. According to an estimate around 10.65 billion tons of CO adds up in the atmospheric air, due to burning of fossil fuels. Hence it is necessary to opt for better, clean and renewable sources of energy.

Wind is one of the renewable sources of energy and therefore many researches are being continuously done to effectively harness its power. It is a clean and a renewable source of energy. Wind energy is abundant and is widely distributed in nature. Harnessing wind energy does not lead to the production or emission of greenhouse gases. Owing to these reasons, wind energy is widely preferred as an alternative to the fossil fuels.With the help of wind turbines, kinetic energy of wind gets converted into mechanical energy which is further converted into electrical energy. Till date, the capacity of largest wind turbines is up to 6 MW.There are various environmental challenges in for the operation of wind turbines. Some of these challenges are high variation in angle of attack, turbulence in the surrounding atmosphere and wakes from adjacent turbines, tower wakes etc. All these factors must be considered while designing a wind turbine.It is important to predict or estimate aerodynamics associated with wind turbines. Setting up a wind tunnel to perform full scale experiments would be a costly approach. CFD or Computational Fluid Dynamics would be a better and economic approach. To analyze the performance of wind turbines, use of CFD is increasing since the last decade. Day by day, technology is increasing and so the computation power of computers, which is beneficial in the case of CFD. Today, use of CFD in supercomputers is able to create the simulation of flow of wind around a wind turbine. There are different CFD codes for different turbulence models and the specific one can be used for the given conditions. Though the modern approach of simulation made the task easier and cheaper but it is still a challenge to simulate wind flow around wind turbines, due to its complicated aerodynamics.The flow of air at the hub of turbine is different from the flow of air at the tip of the blades. Also the flow of air varies along the length of the blades of the turbine. Therefore the aerodynamics of the surface of rotor of any wind turbine is very different in nature and is rarely seen in any other field of aerodynamics.All the factors discussed above were the source of motivation to perform this research. The intention of this work is to study, analyze and understand the aerodynamics (unsteady) of the wind turbines. The purpose is achieved by performing the CFD simulations of the flow of wind around a wind turbine.

# 2. CFD Analysis of Wind Flow over Wind Turbine

To obtain a solution in a CFD solver, basic steps must be followed. The two most important stepsinvolved are “Preprocessing” and “Post processing”. Generation of the grid or mesh and conversion of this grid into a CFD compatible format is the part of preprocessing, while visualizing the results though obtained is done in the post processing step, which is the final step. Preprocessing is generally the most time consuming and tedious job. Software related to post processing is coupled to the CFD solver. Sections given below explain these steps used in this work.

## 2.1 Mesh Generation

Grid or mesh generation is the first step performed to start the preprocessing steps. It is one of the most important steps as it works as the foundation of the solution, therefore it should be done as accurately as possible. The quality of mesh is responsible for the accuracy in the results.Most widely used grids are of two type viz. structured grid and unstructured grid. A regular repeating element is the characteristics of structured grid. Structured grids in 2D are represented by quadrilateral elements while hexahedral elements represent such grids in 3D. As the pattern is made up of a regular repeating element, the information regarding the connectivity of the elements is implicitly stored. Every cell of the grid is addressed directly by the indices (i,j) and (i,j,k) in 2D and 3D respectively, which saves a lot of computational efforts in calculations.

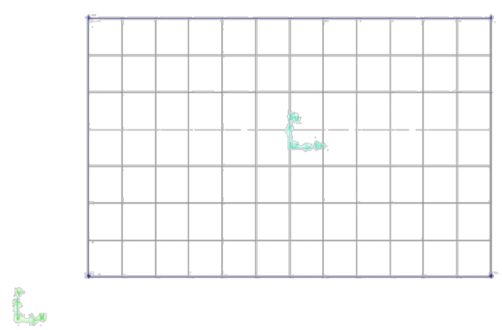

**Fig 2.1 a typical 2d structured grid consisting of rectangular elements.**

## 2.2 Domain Decomposition

Another major step of preprocessing is the decomposition of domain. Computations are expensive because of large calculations. This expense can be reduced to a limit by decomposing a single big mesh into a group of number of smaller meshes. Then these smaller meshes are given for calculations separately to different machines or CPUs, which makes it possible to perform their parallel calculations, which saves a lot of time.

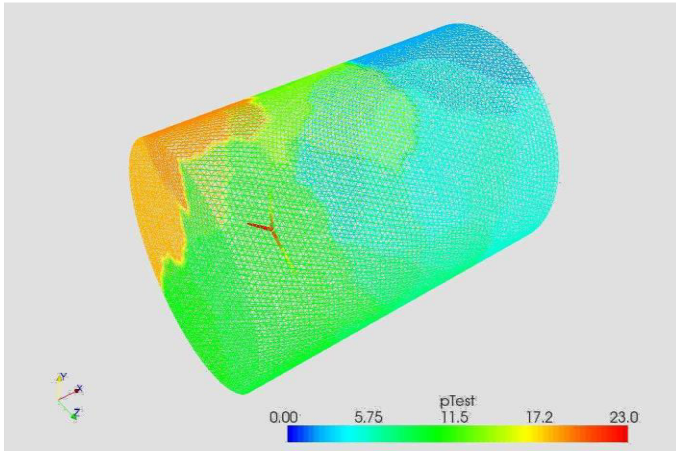


**Fig 2.2 Mesh around wind turbine rotor is decomposed into 24 smaller meshes using a METIS partitioning method. Coloured by processor Id**

## 2.3 Post processing

The software used for this step, in this study was the para View. This motive of this step by quadrilateral elements while hexahedral elements represent such grids in 3D. As the pattern is made FOAM's built in post processing module named "para Foam" made it accessible to use Para View. The results obtained from the foam were imported in the Para view by using para foam.

**Table 2.1 List of some post-processing utilities used in the Simulations.**

| Utility | Operation |
|---|---|
| vorticity | Calculates the vorticity magnitude as well as components. |
| Calculate Torque | Calculates torque and power at the specified patch. |
| Patch Integrate | Integrates a field (p, U, k, etc) over the specified patch. |
| Q | Calculates the second invariant of the velocity gradient tensor. |

## 2.4 Boundary and Initial Conditions and Turbulence Model

### 2.4.1 Boundary and Initial Conditions

To solve the problem in CFD solver, the next step is to apply the boundary conditions for the patches of the domain. Domain patches applied on the boundaries is considered as boundary conditions for the patches of the domain. Basically there are two divisions in the conditions, which are the Neumann boundary conditions and the Dirichlet conditions. Neumann boundary conditions are used for defining the gradients at the patch of the boundary while Dirichley are used for defining a dependent variable by a patch value. Hence the two types of boundary conditions are described in the work viz. fixed gradient type or the fixed value type.

Turbo Dynamic MF Foam and MRF Simple Foam involve the implementation of various boundary conditions. The various boundary conditions used in this study are listed in the table 2.2 is to visualize and analyze the solutions as obtained from the FOAM solver.

**Table 2.2 Boundary conditions used and their Operations.**

| **Boundary Condition** | **Operation** |
|---|---|
| Fixed Value | Fixes the value of a particular Patch field to the value specified by the user. |
| Zero Gradient | Normal gradient of the patch field is 0. |
| slip | For a vector field, the normal component is set to 0 while tangential is set to zero Gradient. For a scalar field, it is same as zero-Gradient. |
| cyclic (periodic) | Sets the periodic boundary condition across the patches. |
| ggi | interpolates flow variables between the moving and the static mesh patches. |

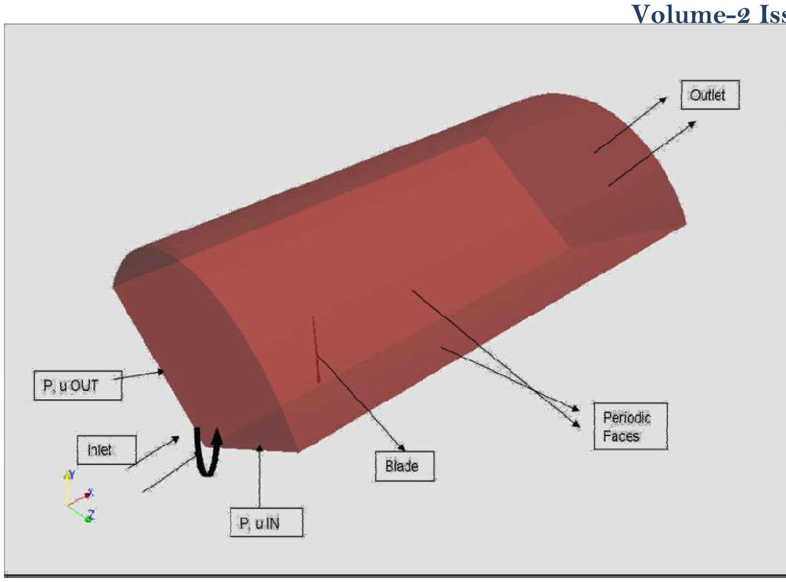


**Fig 2.3 Schematic representation of periodic boundary conditions.**

## 2.4.2 Turbulence Model

To solve the turbulent eddy viscosity model used was the Spalart Allmaras single equation model for the turbulence. In this model, only one differential equation is used for the eddy viscosity $\nu_T$. The eddy viscosity for this model is given as:

$$\nu_T = \tilde{\nu} f_{v1}$$

where $f_{v1} = \frac{X^3}{X^3 + C_{v1}^3}$ and $X = \frac{\tilde{\nu}}{\nu}$.

Here, $\nu$ is the molecular viscosity.

$\tilde{\nu}$ follows the following transport equation

$$\frac{D\tilde{\nu}}{Dt} = c_{b1}(1 - f_{t2})S\tilde{\nu} + \frac{1}{\sigma}\left[\nabla.((\nu + \tilde{\nu})\nabla\tilde{\nu}) + c_{b2}(\nabla\nu)^2\right] - \left[c_{w1}f_w - \frac{c_{b1}}{k}f_{t2}\right]\left[\frac{\tilde{\nu}}{d}\right]^2 + f_{t1}\nabla(u)^2 \qquad (2.1)$$

Equation 2.1 is already used in Open FOAM. For the detailed explanation of the turbulence model and development of the equation given above including the use of it, please refer to Spalart -Allmaras (1994).

The turbulence model of the Spalart Allmaras is comparatively a simple and single equation model which works good for solving the turbulence problems for simple boundary layers. It works by solving the modeled transport equation of the eddy viscosity (kinematic). The design of this model was specifically for analyzing the aerodynamics of in any system. Because of the use of single or one equation model (which is solved for only one variable) reduces the effort required for the computation. The implementation of the Spalart-Allmaras model is dependent on the boundary condition specified for the $\tilde{\nu}$, therefore the boundary conditions applied for $\tilde{\nu}$ in our simulation for various patches are:

**Table 2.3 Boundary Conditions for $\tilde{\nu}$**

| Patch | BC |
|---|---|
| inlet | Fixed value (0.001) |
| outlet | Zero Gradient |
| side walls | Zero Gradient |
| rotor | Zero Gradient |
| periodic face | Fixed value (0.001) |

# 3. Results and Discussion

The graph representing the comparison of the theoretical values with the experimental values is then finally plotted, which was helpful in the validation of the approach and in the validation of the code. As the simulation for setup include the complete assembly of the wind turbine like the tower, yaw, ground, rotor etc, the simulation generated the actual flow of the wind representing the actual behavior of the flow. The first comparison will be made in power output of the rotor. Figure 3.1 shows this comparison. Various observations can be made from the figure 3.1. It can be observed that CFD values somehow match with the experimental results, though the results were not accurate. The reason for this missing accuracy might be the limitations of the turbulence model used in the current work, which obvious as flow of wind over blade has many complexities like flow separation, stalling etc.

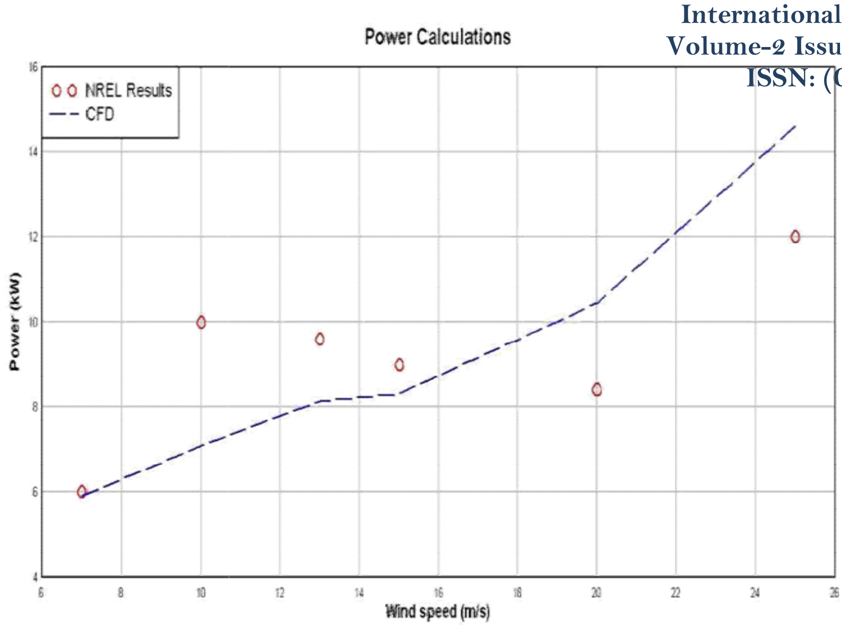


**Fig 3.1 Comparison of power values between the CFD Values and the NREL experimental values.**

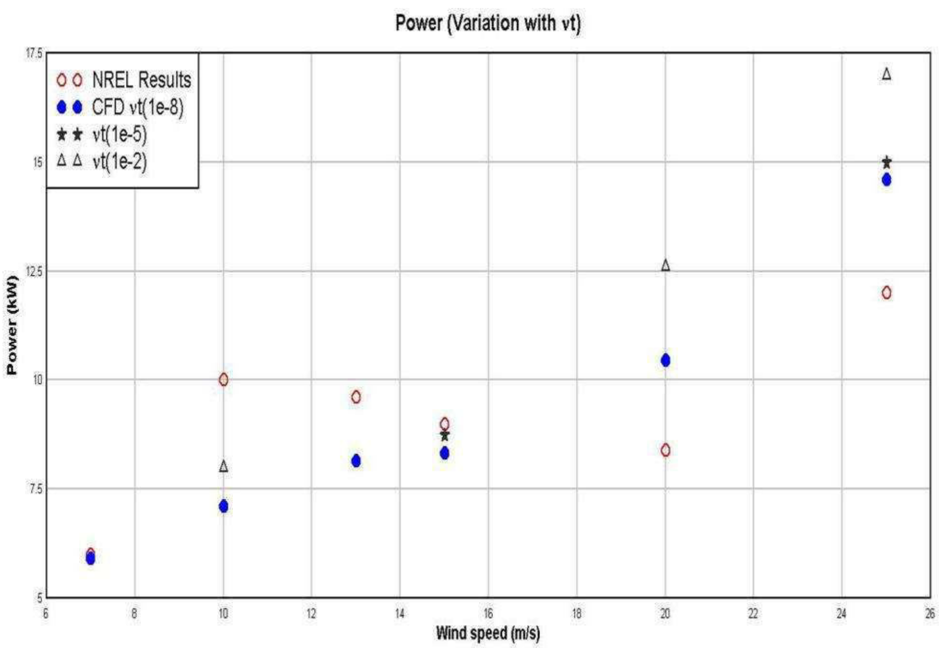


**Fig 3.2 Variation of the rotor power with the variation of inlet eddy viscosity (changing turbulence parameters).**

As the analysis of the power output was completed, the validation was supported by the analyzing the variation of the power distribution over the blade. It is followed by the calculation of the coefficient of pressure over both (upper and lower) surface of the blade, as given in the equation below.

$$Cp = \frac{P - P_\infty}{0.5^{*}\rho(U_\infty^2 + (r^2\Omega^2)}$$

Here

P= value of pressure on the blade along its chord at any point.

$U_\infty$= Velocity of the flow at the inlet

Ω = Angular speed of the blade

The coefficient of the pressure was calculated for different sections of the blade. The values of the coefficient of pressure though obtained by the simulation were compared with those values which are available from the experimental results.

Graph given the three figures can be observed for the comparison of the results, in the case of computation of the coefficient of pressure and it can be seen that match between the simulation results and the experimental results was not successful. Results corresponding to the upper surface of the blade does not match though of the lower surface matches well. The reason for unsuccessful matching of the upper surface values might the high value of separation of flow which was not solved by the solver Effectively & Efficiently.

Figure 3.1 better explains the reason for these plots. Tip of the blade has an important role in the power output from the rotor and thus the corresponding section is an important section to be analyzed. If we observe the graph of the tip section for the coefficient of the pressure, we can observe that the values obtained by simulation are higher than the experimental values.

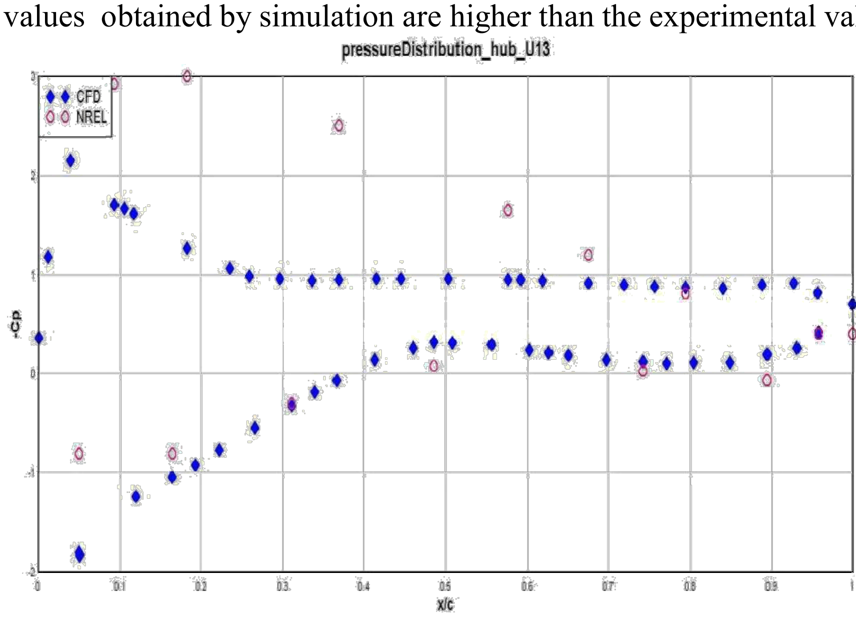


**Fig 3.3 Coefficient of pressure at 30%R blade location for 13 m/s inlet wind speed**

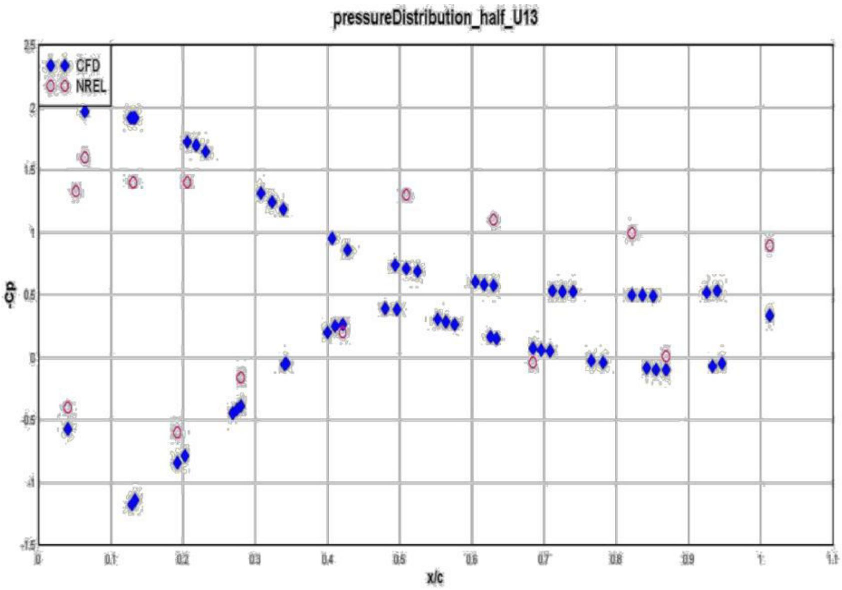


**Fig 3.4 Co-effcient of pressure at 50%R blade location for 13 m/s inlet wind speed.**

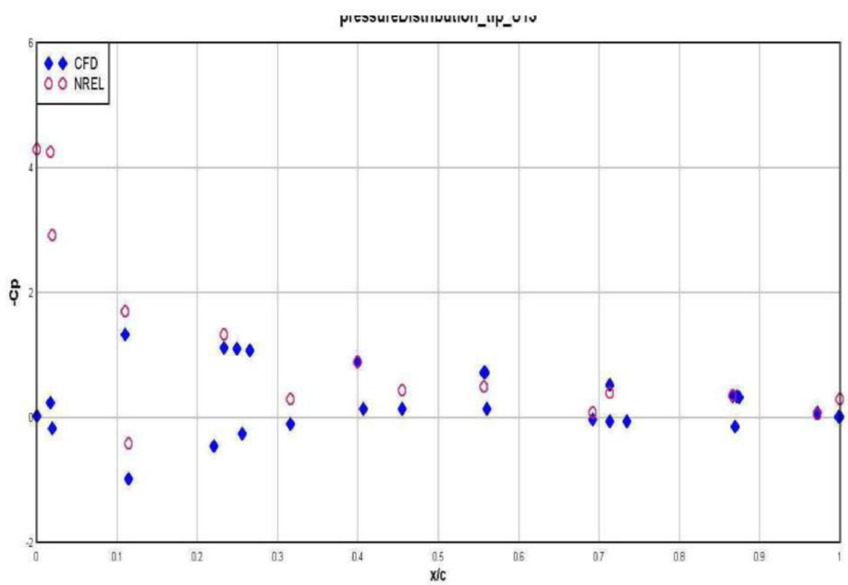


**Fig 3.5 Co-effcient of pressure at 95%R blade location for 13 m/s inlet wind speed.**

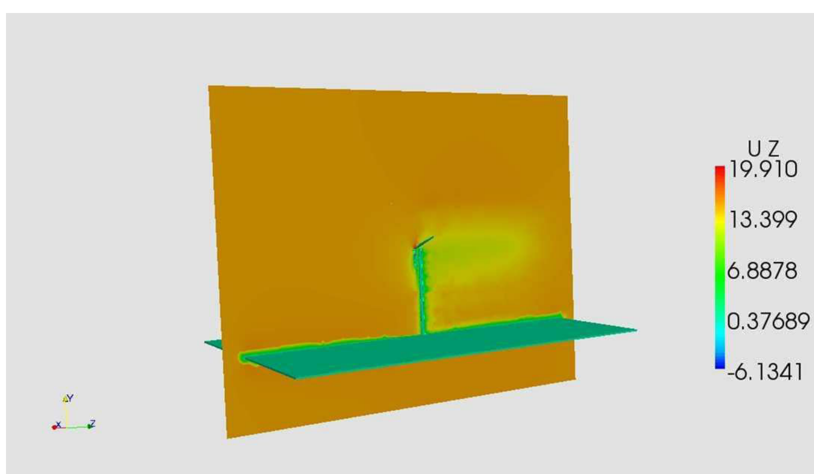


**Fig 3.6 . Stream wise velocity in the computational domain. U∞ = 15m/s**

In the figure 3.6 a section through the domain is shown for the display of the velocity of the wind inside the domain. It is easy to observe and mark the wake. Considerable amount of decrease in the speed of the wind past the tower clearly give the strong significance of the generation of the wake and due to which the power output of the turbine located at the downstream will get highly affected. In the figure 3.7 vorticity is plotted on an iso surface, which are clearly observable. Wake generated was symmetrical about the tower while the vorticity in the flow of the wind diffuses in the direction of the flow i.e. the downstream.

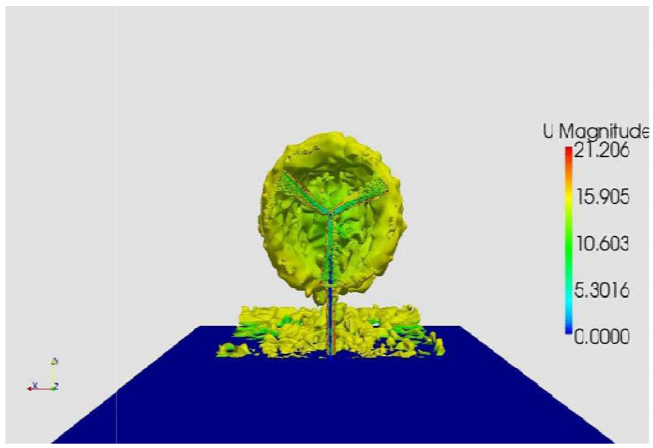


**Fig 3.7 Iso surface plot of stream wise vorticity coloured by velocity magnitude representing the wake of the rotor blades . U∞ = 15m/s**

# 4. Conclusion and Future work

This work was performed to create a code for the analysis of wind turbine assembly and validate it while establishing a understanding of the flow behavior of the wind around the wind turbine. The code was created in MRF Simple Foam.The simulation performed on the blade of wind turbine of NREL Phase VI revealed that the code was not able to generate successful match with the experimental values, but there was a trend between the simulation values and the experimental values, within a range of comparison. The difference in the geometry of airfoil sections of the two blades (simulation blade and experimental blade) might be the reason for this difference in results. The simulation of the blade of NREL wind turbine motor was performed by creating the geometry of the similar blade in GAMBIT by using the table of S809 airfoil and this blade was subjected to different Magnitudes of the speed of the wind for making analyses of the different cases. RANS equations were used for the conservation of momentum and continuity while Spalart Allmaras model was used for dealing with the turbulence in the simulation.The parameters like output power, coefficient of pressure etc. were obtained and compared with the experimental values and the values matched with slight difference. Finally this work was able to analyze the complexities involved in the aerodynamic behavior of the flow of the wind through a wind turbine rotor, and the knowledge regarding the simulation, use of GGI, use of Open Foam solver and CFD was gained.Though the steady state results were obtained with some accuracy but the transient analysis of the turbulent model of the wind turbine in the turbo Dynamic Foam solver was not accurate and therefore can be worked upon in future to improve its accuracy, especially for solving larger problem size involving high aerodynamics and turbulence.The algorithm of the GGI mesh interface can be improved for increasing the accuracy while decreasing the time and computational efforts.This framework of CFD can be used further for making the simulations of the models of the turbine and phenomenon like wake, flow separation etc associated with it by others.

## 5. References

1. Patrick Irungu Muiruri and Oboetswe Seraga Motsamai for Three dimensional CFD Simulation of a Wind turbine blade Section.

2. Myungsung Lee and Seung Ho Lee for a A Numerical simulation of flow field in a wind Farm on Complex Terrain.

3. A.A.Memon and H.U.Abbasi for Modelling and simulation of flow around wind turbine blade by using Actuator Surface Model.

4. Antonio Segalini for Linearized simulation of flow over wind Farms and Complex Terrains.

5. Mr. Monir Chandrala and Prof Abhishek Choubey for Aerodynamic analysis of Horizontal Axis Wind Turbine blade.

6. P. Benard and A. Vire for large Eddy Simulation of wind turbine Wakes including Geometrical Effects.