



Computation Fluid Dynamics of Wind Turbine Using S809 Airfoil and Ansys CFX

Himanshu Khandelwal

Assistant Professor

Modi Institute of Technology Kota

Abstract : Computational Fluid Dynamics (CFD) is the analysis of fluid flows using numerical solution methods. Using CFD, you are able to analyze complex problems involving fluid-fluid, fluid-solid or fluid-gas interaction. Engineering fields where CFD analyses are frequently used are for example aerodynamics and hydrodynamics, where quantities such as lift and drag or field properties as pressures and velocities are obtained. Fluid dynamics is involved with physical laws in the form of partial differential equations. Sophisticated CFD solvers transform these laws into algebraical equations and are able to efficiently solve these equations numerically. The aerodynamic design of optimum rotor blades from a known airfoil type means determining the geometric parameters (such as chord length and twist angle distribution along the blade span) for a certain tip-speed ratio at which the power coefficient of the rotor is maximum. For this reason firstly the change of the power coefficient of the rotor with respect to tip-speed ratio should be figured out in order to determine the design tip-speed ratio λ_d where the rotor has a maximum power coefficient. The blade design parameters will then be according to this design tip-speed ratio. Examining the plots between relative wind angle and local tip-speed ratio for a wide range of glide ratios gives us a unique relationship when the maximum elemental power coefficient is considered. And this relationship can be found to be nearly independent of glide ratio and tip-loss factor.

IndexTerms - Computation Fluid Dynamics , Wind Turbine , S809 Airfoil , Ansys CFX.

I. INTRODUCTION

Wind power is one of the fastest-growing renewable energy technologies. Usage is on the rise worldwide, in part because costs are falling. Global installed wind-generation capacity onshore and offshore has increased by a factor of almost 75 in the past two decades, jumping from 7.5 gigawatts (GW) in 1997 to some 564 GW by 2018, according to IRENA's latest data. Production of wind electricity doubled between 2009 and 2013, and in 2016 wind energy accounted for 16% of the electricity generated by renewables. Many parts of the world have strong wind speeds, but the best locations for generating wind power are sometimes remote ones. Offshore wind power offers tremendous potential.[1]

Wind turbines first emerged more than a century ago. Following the invention of the electric generator in the 1830s, engineers started attempting to harness wind energy to produce electricity. Wind power generation took place in the United Kingdom and the United States in 1887 and 1888, but modern wind power is considered to have been first developed in Denmark, where horizontal-axis wind turbines were built in 1891 and a 22.8-metre wind turbine began operation in 1897. [1]

Wind is used to produce electricity using the kinetic energy created by air in motion. This is transformed into electrical energy using wind turbines or wind energy conversion systems. Wind first hits a turbine's blades, causing them to rotate and turn the turbine connected to them. That changes the kinetic energy to rotational energy, by moving a shaft which is connected to a generator, and thereby producing electrical energy through electromagnetism.

The amount of power that can be harvested from wind depends on the size of the turbine and the length of its blades. The output is proportional to the dimensions of the rotor and to the cube of the wind speed. Theoretically, when wind speed doubles, wind power potential increases by a factor of eight. [2]

Wind-turbine capacity has increased over time. In 1985, typical turbines had a rated capacity of 0.05 megawatts (MW) and a rotor diameter of 15 metres. Today's new wind power projects have turbine capacities of about 2 MW onshore and 3–5 MW offshore. [2]

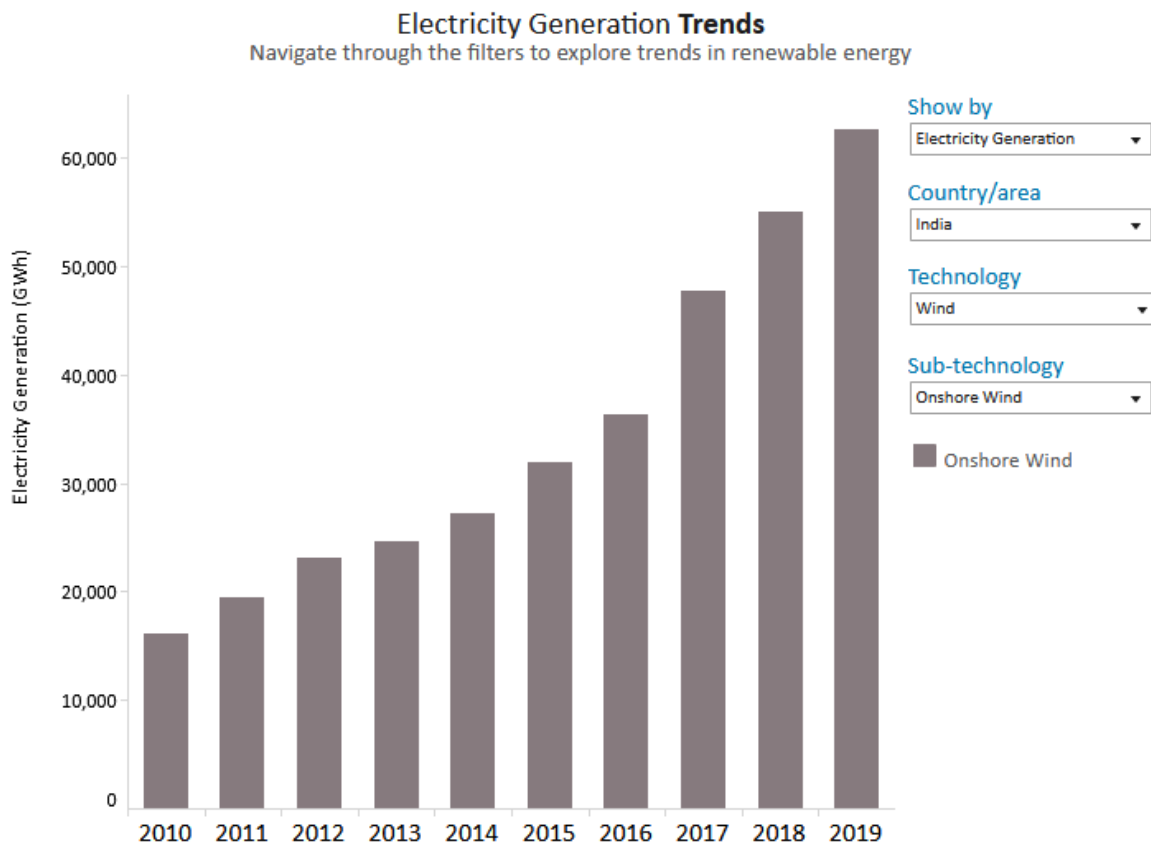


Fig 1.1 Wind Energy Electricity Generation Data

Commercially available wind turbines have reached 8 MW capacity, with rotor diameters of up to 164 metres. The average capacity of wind turbines increased from 1.6 MW in 2009 to 2 MW in 2014. [2]

1.2 Wind Turbines

Wind turbines, like windmills, are mounted on a tower to capture the most energy. At 100 feet (30 meters) or more aboveground, they can take advantage of the faster and less turbulent wind. Turbines catch the wind's energy with their propeller-like blades. Usually, two or three blades are mounted on a shaft to form a rotor. [3]

A blade acts much like an airplane wing. When the wind blows, a pocket of low-pressure air forms on the downwind side of the blade. The low-pressure air pocket then pulls the blade toward it, causing the rotor to turn. This is called lift. The force of the lift is actually much stronger than the wind's force against the front side of the blade, which is called drag. The combination of lift and drag causes the rotor to spin like a propeller, and the turning shaft spins a generator to make electricity. [3]

1.2.1 Land-Based Wind Energy

Wind turbines can be used as stand-alone applications, or they can be connected to a utility power grid or even combined with a photovoltaic (solar cell) system. For utility-scale (megawatt-sized) sources of wind energy, a large number of wind turbines are usually built close together to form a wind plant, also referred to as a wind farm. Several electricity providers today use wind plants to supply power to their customers. [4]

Stand-alone wind turbines are typically used for water pumping or communications. However, homeowners, farmers, and ranchers in windy areas can also use wind turbines as a way to cut their electric bills.



Fig 1.2 Land- Based Wind Turbines

Page size: A4 size only

II. LITERATURE SURVEY

S. S. Miriyala, R. Banerjee and K. Mitra, 2020 [4] In contemporary practices, Computational Fluid Dynamics (CFD) based tools are increasingly applied to build high fidelity First Principles based Models (FPMs) for designing tactical missile systems. However, optimization, sensitivity analysis and uncertainty quantification using such models still remain to be extremely tedious and, hence, are performed offline. Artificial Neural Networks (ANNs) are known to be efficient machine learning based models capable of modelling large amount of nonlinearities in the data. However, heuristics involved in their modelling prevent their application as surrogate models to computationally intensive FPMs. In this work, a novel algorithm, aimed at simultaneous optimal estimation of architecture (number of hidden layers and nodes in each layer), training sample size and activation function in ANNs is proposed. The proposed algorithm classifies as a generic multi-objective evolutionary neural architecture search strategy to design ANNs. It is solved using the population based evolutionary optimization algorithm called Nondominated Sorting Genetic Algorithm-II. The high fidelity data to train the ANNs are obtained from CFD model for simulating supersonic flow over a tactical missile body in ANSYS Fluent®. The obtained ANNs with a test set accuracy of around 99% are then used for quantifying uncertainties, due to order of discretization, flux and Spalart-Allmaras turbulence model in CFD simulations, on the coefficients of lift, drag and rolling moment using the method of analysis of variance (ANOVA). The applicability of optimally designed MLP surrogates for performing ANOVA based study presents novel insights into the epistemic uncertainties arising due to the CFD model which can help in realistic designs of tactical missiles.

R. Havryliv, I. Kostiv and V. Maystruk, 2020 [5] The effects of the mesh size and time step size on CFD modeling results have been studied. The numerical modeling results present the flow distribution of velocity fields inside the industrial scale vessel of 30 m³ equipped with a side-mounted agitator. For the turbulent fluid flow modeling the “realizable k-ε” model and Sliding Mesh approach implemented in the ANSYS Fluent software were used for transient simulation. The optimal installation values for transient modeling parameters for the mixing process realized in Ansys Fluent software were proposed.

S. Aslan *et al.*, 2020 [6] This paper proposes a novel method to noninvasively measure the peak systolic pressure difference (PSPD) across coarctation of the aorta for diagnosing the severity of coarctation. Traditional non-invasive estimates of pressure drop from the ultrasound can underestimate the severity and invasive measurements by cardiac catheterization can carry risks for patients. To address the issues, we employ computational fluid dynamics (CFD) computation to accurately predict the PSPD across a coarctation based on cardiac magnetic resonance (CMR) imaging data and cuff pressure measurements from one arm. The boundary conditions of a patient-specific aorta model are specified at the inlet of the ascending aorta by using the time-dependent blood velocity, and the outlets of descending aorta and supra aortic branches by using a 3-element Windkessel model. To estimate the parameters of the Windkessel model, steady flow simulations were performed using the time-averaged flow rates in the ascending aorta, descending aorta, and two of the three supra aortic branches. The mean cuff pressure from one arm was specified at the outlet of one of the supra aortic branches. The CFD predicted PSPDs of 5 patients (n=5) were compared with the invasively measured pressure drops obtained by catheterization. The PSPDs were accurately predicted (mean $\mu=0.3\text{mmHg}$,

standard deviation $\sigma = 4.3\text{mmHg}$) in coarctation of the aorta using completely non-invasive flow and cuff pressure data. The results of our study indicate that the proposed method could potentially replace invasive measurements for estimating the severity of coarctations. Clinical relevance-Peak systolic pressure drop is an indicator of the severity of coarctation of the aorta. It can be predicted without any additional risks to patients using non-invasive cuff pressure and flow data from CMR.

C. Du, I. Firmansyah and Y. Yamaguchi, 2020 [7] Lattice Gas Cellular Automata (LGCA) simulations are typical High-Performance Computing (HPC) applications commonly used to simulate fluid flows. Due to the computational locality and discretization of LGCA, these simulations can achieve high performance by using parallel computing devices like GPUs or multi-core CPUs. Nevertheless, many studies also have shown that state-of-the-art Field Programmable Gate Arrays (FPGAs) have enormous parallel computing potential and power-efficient for high-performance computations. In this paper, we present an FPGA-based fluid simulation architecture design for the LGCA method. Our design exploits both temporal and spatial parallelism inside the LGCA algorithm to scale up the performance on FPGA. We also propose an application-specific cache structure to overcome the memory bandwidth bottleneck. Furthermore, our development process is based on the High-Level Synthesis (HLS) approach that increases productivity. Experimental results on a Xilinx Vcu 1525 FPGA show that our design is able to achieve 17130.2 Million Lattice Updates Per Second (MLUPS).

A. O. Ayeleso, M. T. E. Kahn and A. K. Raji, 2016 [8] This paper presents a three-dimensional (3-D) simulation of incompressible viscous flow of an electrically conducting ionised gas inside a non conducting rectangular duct. The understanding of the flow of such fluid along the duct domain is very important in the design of a direct energy conversion system such as magnetohydrodynamics, MHD, power generation. The operating fluid considered in the present study is a propane gas, with burning temperature of about 1977°C . This gas, with electrical conductivity of 100 s/m start through an inlet velocity of 1 m/s along the axial direction and is transversed by a uniform magnetic field in the radial direction. The uniform magnetic flux densities, B , values selected are 0 T , 1.12 T , 1.32 T and 1.41 T . Subsequently, numerical simulations are performed by means of the Comsol Multiphysics Commercial Software, which is used to solve the governing 3-D fluid dynamics and electromagnetic field partial differential equations iteratively (Maxwell's and Reynolds-Average Navier-Stokes). The x-component velocity field of the propane gas fluid is found to decrease when the intensities of the applied magnetic field are increased. In addition, the fluid flow pressure is low at the duct entrance and gradually increases at the center of the duct due to the intensity of the applied magnetic fields. The computation analysis performed in this work present a useful starting point to study more complex problems, especially with the potential use of MHD power generator for industrial applications.

P. Berg *et al*, 2016 [9] Computational Fluid Dynamics enables the investigation of patient-specific hemodynamics for rupture predictions and treatment support of intracranial aneurysms. However, due to numerous simplifications to decrease the computations effort, clinical applicability is limited until now. To overcome this situation a clinical research software prototype was tested that can be easily operated by attending physicians. In order to evaluate the accuracy of this prototype, four patient-specific intracranial aneurysms were investigated using four different spatial resolutions. The results demonstrate that physicians were able to generate hemodynamic predictions within several minutes at low spatial resolution. However, depending on the parameter of interest and the desired accuracy, higher resolutions are required, which will lead to an increase of computational times that still look very attractive towards clinical usability. The study shows that the next step towards an applicable individualized therapy for patients harboring intracranial aneurysms can be done. However, further in vivo validations are required to guarantee realistic predictions in future studies.

H. Xin-Cheng and A. -I. Xiao, 2014 [10] With the rapid development and popularization of CFD technology, requirements on data amount and computing speed have also increased a lot. The computing ability of single CPU is unable to meet the requirement. Therefore, iteration and parallel computation are the most important methods for CFD problems with large grid scales. This paper introduces the most popular iteration methods, classical iteration methods and their parallel computation strategies. On the basis of it, the parallel computing simulation environment is also constructed. It comprehensively applies symmetric multi-processing and distributed technologies and Windows XP 64 bit system. Research on CFD has focus on the parallel computation, which can meet the practical demand for computer capacity and computing time.

III. PROPOSED WORK

3.1 CFD Modelling Around a Wind Turbine

Wind turbines were first used to produce electricity in the twentieth century, in response to accelerated economic growth and increased power consumption. Nowadays wind energy is highly desirable for being a non-polluting and theoretically inexhaustible mode of power generation. Another relevant factor is the strong ecological appeal present in the wind energy generation, often listed as a requirement in current projects.

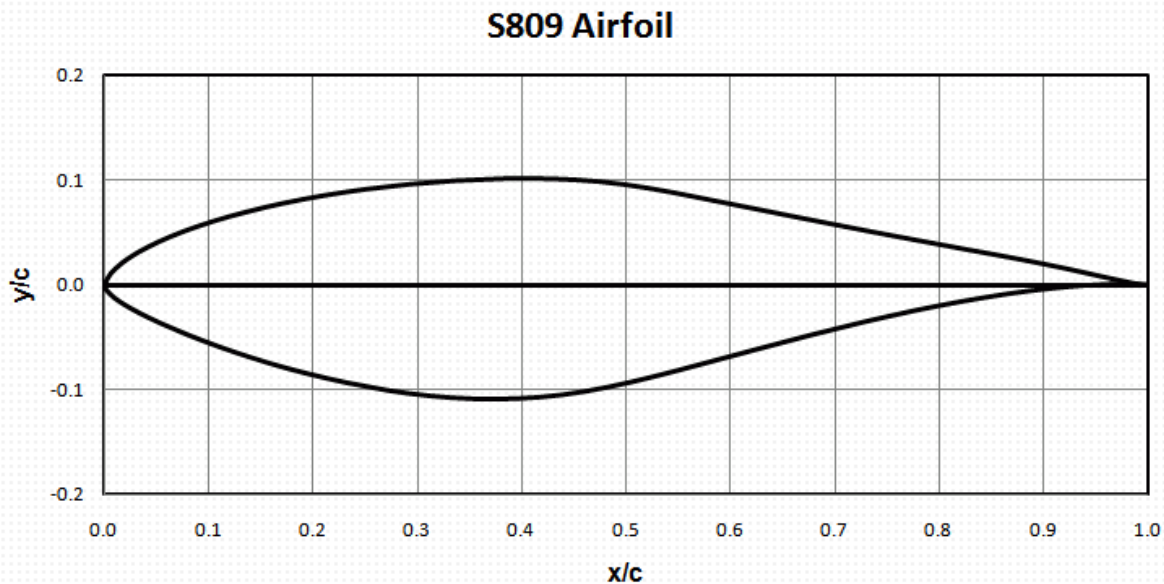


Fig 3.9 S809 Airfoil

The S809 airfoil was designed specially for **horizontal-axis-wind-turbine (HAWT)** applications. The thickness ratio of the airfoil is 21%. The asymmetric profile of the S809 airfoil, with a convex upper surface, a rounded edge called leading edge, and a sharp edge referred to as the trailing edge. The asymmetry stems from the shape of the lower surface of the S809 airfoil.

Asymmetric S809 airfoils are optimized to generate higher levels of lift when the lower surface of the airfoil is positioned closer to the direction of airflow.

Using the Spalart-Allmaras turbulent viscosity, the dimensionless lift, drag and pitching moment coefficients were calculated for wind-turbine blade at different angles of attack. These CFD model values were then validated using published calibrated lift and drag coefficients.

These data were exploited in order to select the aerofoil with best aerodynamic performance for basis of a 3-D model. Thereafter a three-dimensional CFD model of small horizontal axis wind-turbine was produced. The numerical solution was carried out by simultaneously solving the three-dimensional continuity, momentum and the Navier-Stokes equations in a rotating reference frame.

3.2 Navier–Stokes equations

In physics, the Navier–Stokes equations are certain partial differential equations which describe the motion of viscous fluid substances, named after French engineer and physicist Claude-Louis Navier and Anglo-Irish physicist and mathematician George Gabriel Stokes.

The Navier–Stokes equations mathematically express conservation of momentum and conservation of mass for Newtonian fluids. They are sometimes accompanied by an equation of state relating pressure, temperature and density. They arise from applying Isaac Newton's second law to fluid motion, together with the assumption that the stress in the fluid is the sum of a diffusing viscous term (proportional to the gradient of velocity) and a pressure term—hence describing viscous flow. The difference between them and the closely related Euler equations is that Navier–Stokes equations take viscosity into account while the Euler equations model only inviscid flow. As a result, the Navier–Stokes are a parabolic equation and therefore have better analytic properties, at the expense of having less mathematical structure (e.g. they are never completely integrable).

IV. IMPLEMENTATION

4.1 ANSYS CFX

Ansys computational fluid dynamics (CFD) products are for engineers who need to make better, faster decisions. Our CFD simulation products have been validated and are highly regarded for their superior computing power and accurate results. Reduce development time and efforts while improving your product's performance and safety. Intuitive, yet extremely powerful, our computational fluid dynamics software's accelerate product development. Ansys CFD products give you the possibility to make incredible progress through innovation as pressures to optimize products ratchet up and margins for error narrow quickly. Comprised of the industry's most accurate and trusted solvers, you will have confidence in your results. Whether you want to maximize the efficiency of an internal combustion engine or perform an in-flight icing simulation, Ansys has the tool for you. Maximize your time and increase productivity with modern, user-friendly Ansys CFD products.

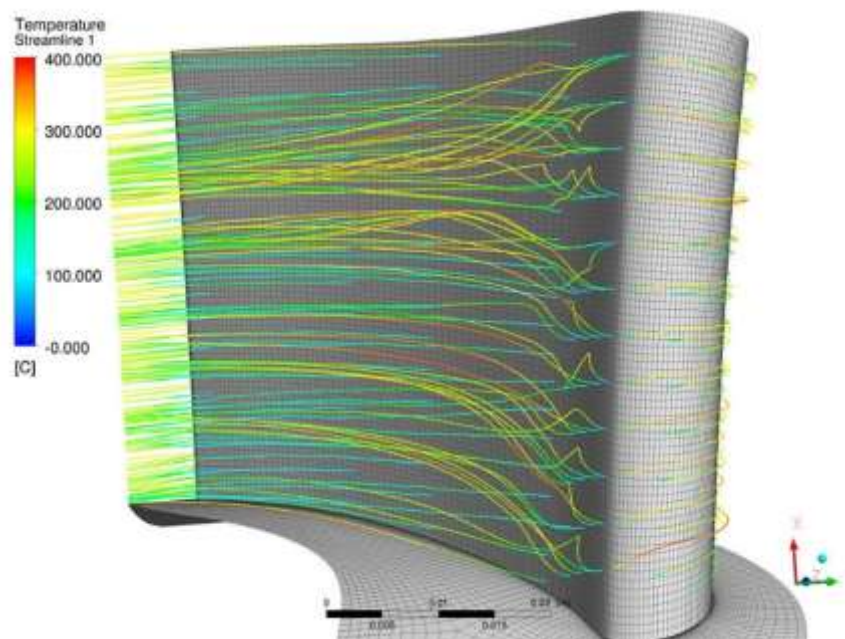


Fig 4.1 ANSYS CFX

Known for its extreme robustness, CFX is the gold standard CFD software when it comes to turbomachinery applications. Both solver and models are wrapped in a modern, intuitive, and flexible GUI, with extensive capabilities for customization and automation using session files, scripting and a powerful expression language. Highly scalable high-performance computing will help speed up simulations including pumps, fans, compressors and turbines. This tool quickly solves turbomachinery problems and provides engineers with more time to focus on their design.

New manufacturing methods have opened the door to more effective turbine cooling channel geometries. The new geometries will lead to better performance and efficiency but are more complex. To achieve highly accurate results, a team of Purdue engineers turned to Ansys CFX.

The engineers executed key computations with little turnaround time. With the extra time, they could perform more in-depth comparisons and run further simulations to fully optimize their product. Ansys CFX gave them the tools to land on a multi-mesh “weave” design. The new turbine design runs with much cooler blades and has been implemented at the Air Force Research Laboratory.

4.2 Proposed Work

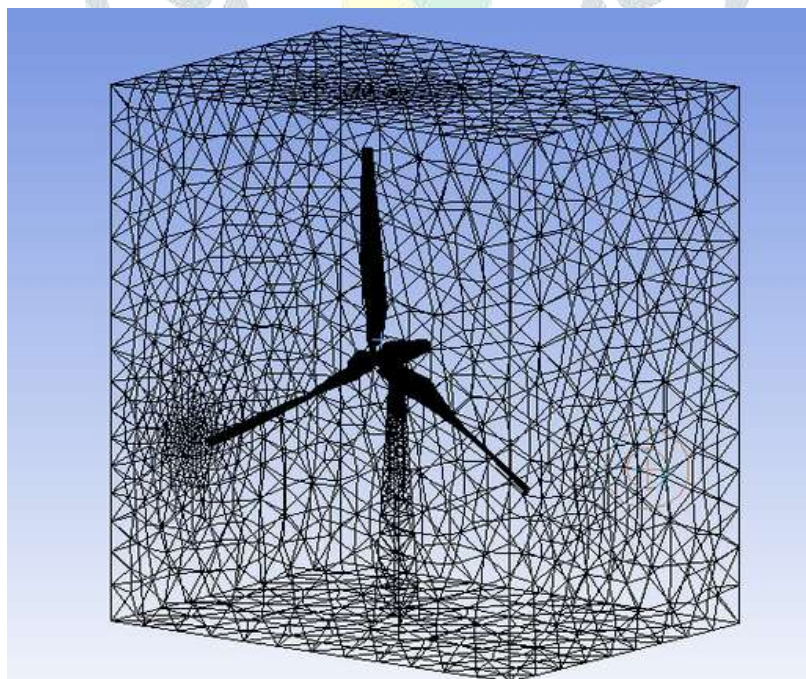


Fig 4.2 Geometry Used for Analysis

The wind Turbine mesh can be read into Design modeller and can be meshed using the automatic mesh option.

Step 1

Chose rename and call the surface inflow.

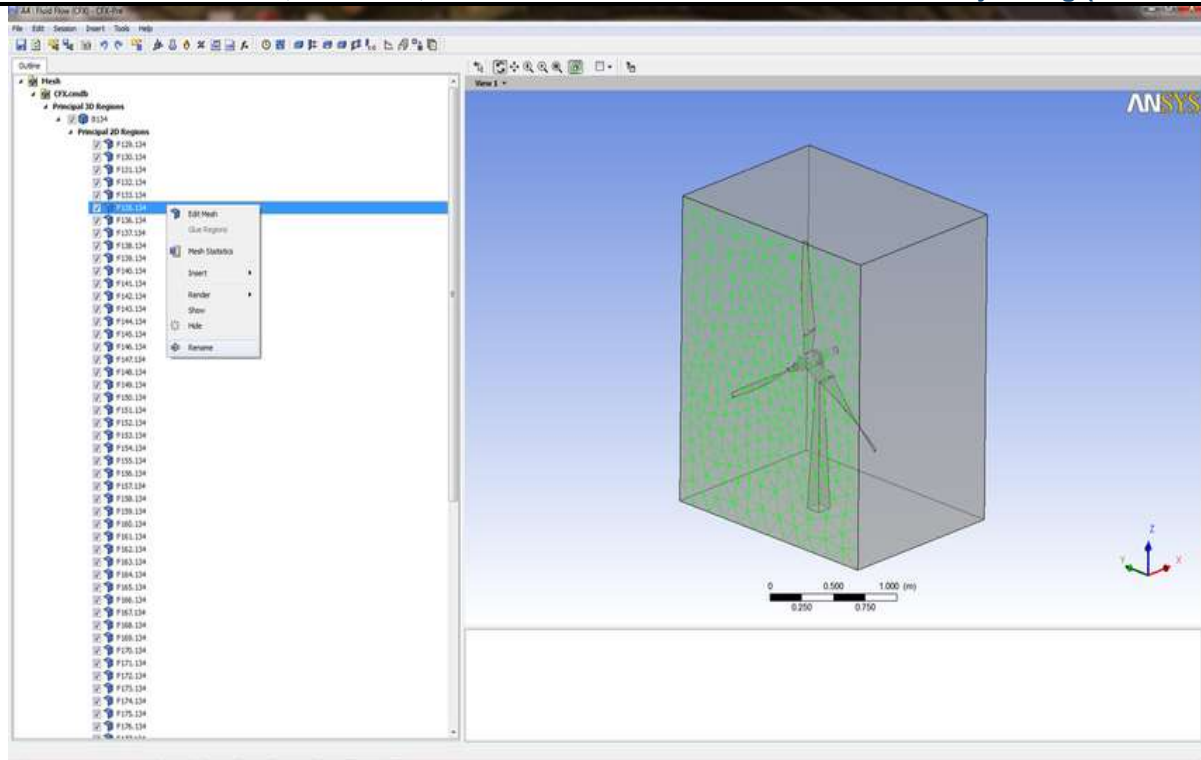


Fig 4.3 Surface Inflow Step 1

Step 2 Call the highlighted surface grass. This boundary will be applied a wall boundary condition.

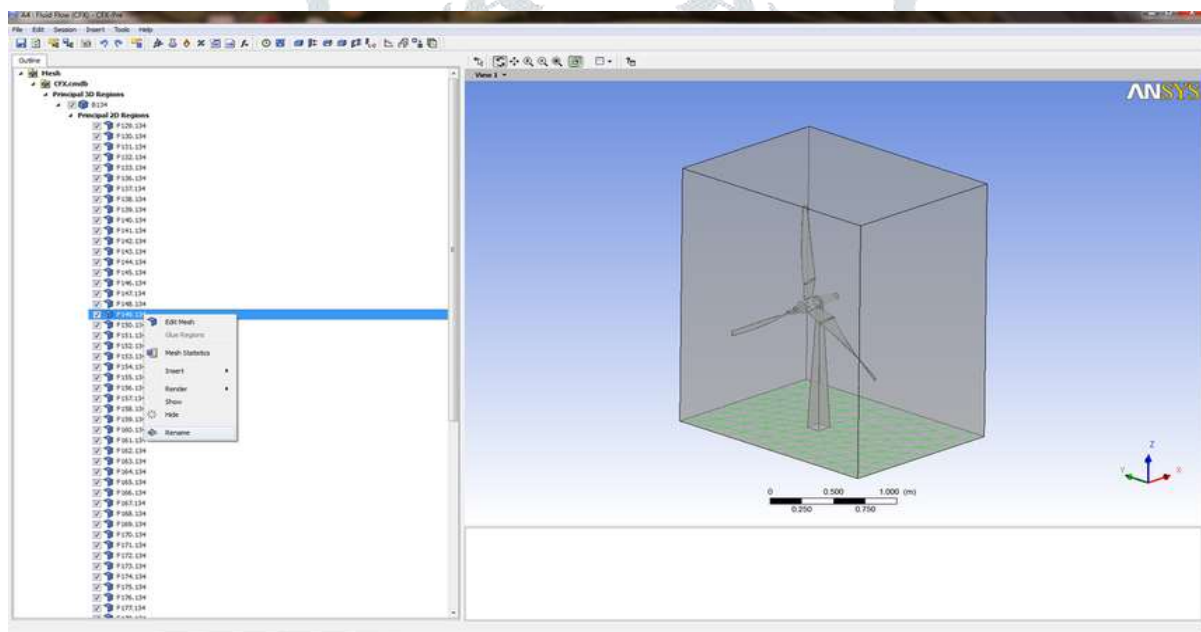


Fig 4.4 Surface Grass Renaming

V. CONCLUSION

Sophisticated CFD solvers transform these laws into algebraical equations and are able to efficiently solve these equations numerically. The aerodynamic design of optimum rotor blades from a known airfoil type means determining the geometric parameters (such as chord length and twist angle distribution along the blade span) for a certain tip-speed ratio at which the power coefficient of the rotor is maximum. For this reason firstly the change of the power coefficient of the rotor with respect to tip-speed ratio should be figured out in order to determine the design tip-speed ratio λ_d where the rotor has a maximum power coefficient. The blade design parameters will then be according to this design tip-speed ratio. Examining the plots between relative wind angle and local tip-speed ratio for a wide range of glide ratios gives us a unique relationship when the maximum elemental power coefficient is considered. And this relationship can be found to be nearly independent of glide ratio and tip-loss factor.

REFERENCES

1. M. Zhao and J. Zou, "Research of Computation about Magnetic Fluid under Magnetic Fluid," *2011 First International Conference on Instrumentation, Measurement, Computer, Communication and Control*, 2011, pp. 381-383, doi: 10.1109/IMCCC.2011.102.

2. A. Topkara, E. Ökeer, B. Hakyemez, E. Ö. Isik and B. Olcay, "Computational fluid dynamics simulations based on time-of-flight magnetic resonance angiography," *2014 18th National Biomedical Engineering Meeting*, 2014, pp. 1-5, doi: 10.1109/BIYOMUT.2014.7026390.
3. M. Dawei, L. Yufeng and X. Yongming, "Analysis of Fluid Field and Temperature Field of MW Wind Turbine Based on Fluent," *2011 Fourth International Conference on Intelligent Computation Technology and Automation*, 2011, pp. 59-62, doi: 10.1109/ICICTA.2011.22.
4. S. S. Miriyala, R. Banerjee and K. Mitra, "Uncertainty quantification using Auto-tuned Surrogates of CFD model Simulating Supersonic flow over tactical missile body," *2020 IEEE Symposium Series on Computational Intelligence (SSCI)*, 2020, pp. 2885-2892, doi: 10.1109/SSCI47803.2020.9308325.
5. R. Havryliv, I. Kostiv and V. Maystruk, "Using The Computational Fluid Dynamic Software To Mixing Process Modeling In The Industrial Scale Vessel With Side-Mounted Agitator," *2020 10th International Conference on Advanced Computer Information Technologies (ACIT)*, 2020, pp. 192-195, doi: 10.1109/ACIT49673.2020.9208986.
6. S. Aslan *et al.*, "Non-invasive Prediction of Peak Systolic Pressure Drop across Coarctation of Aorta using Computational Fluid Dynamics," *2020 42nd Annual International Conference of the IEEE Engineering in Medicine & Biology Society (EMBC)*, 2020, pp. 2295-2298, doi: 10.1109/EMBC44109.2020.9176461.
7. C. Du, I. Firmansyah and Y. Yamaguchi, "High-Performance Computation of LGCA Fluid Dynamics on an FPGA-Based Platform," *2020 5th International Conference on Computer and Communication Systems (ICCCS)*, 2020, pp. 520-525, doi: 10.1109/ICCCS49078.2020.9118557.
8. A. O. Ayeleso, M. T. E. Kahn and A. K. Raji, "Computational fluid domain simulation of MHD flow of an ionised gas inside a rectangular duct (august 2016)," *2016 International Conference on the Industrial and Commercial Use of Energy (ICUE)*, 2016, pp. 84-91.
9. P. Berg *et al.*, "Bringing hemodynamic simulations closer to the clinics: A CFD prototype study for intracranial aneurysms," *2016 38th Annual International Conference of the IEEE Engineering in Medicine and Biology Society (EMBC)*, 2016, pp. 3302-3305, doi: 10.1109/EMBC.2016.7591434.
10. H. Xin-Cheng and A. -I. Xiao, "Iteration and Parallel Computation on Computational Fluid Dynamics," *2014 7th International Conference on Intelligent Computation Technology and Automation*, 2014, pp. 318-321, doi: 10.1109/ICICTA.2014.84.