

Computational Fluid Dynamics in Wind Energy: Modeling and Optimization for Turbine Design

Rahul Atmaram Wagh¹, Kunal Kishor Chandan², Tejas Sanjay Sali³, Saurabh Surendrakumar Jain⁴, Pramod Ambadas Karole⁵, Vikram Keru Dhattrak⁶

¹ Assistant Professor, School of Science, Sandip University, Nashik, Maharashtra, India.
rahul.wagh@sandipuniversity.edu.in

² Assistant Professor, Department of Mechanical Engineering, Sandip University Sijoul, Bihar, India.
kunal.chandan@sandipuniversity.edu.in

³ Assistant Professor, Department of Mechanical Engineering, Sandip Institute of Engineering & Management, Nashik, Maharashtra, India. tejas.sali@siem.org.in

⁴ Assistant Professor, Sandip Institute of Technology & Reserch Centre, Nashik, Maharashtra, India.
saurabh.jain@sitrc.org

⁵ Assistant Professor, Sandip Institute of Technology & Reserch Centre, Nashik, Maharashtra, India.
pramod.karole@sitrc.org

⁶ Assistant Professor, Department of Mechanical Engineering, Sandip Institute of Technology & Reserch Centre, Nashik, Maharashtra, India. vikram.dhattrak@sitrc.org

Article History:

Received: 15-03-2024

Revised: 22-05-2024

Accepted: 05-06-2024

Abstract:

Renewable energy sources are hard to come by, but wind power has become one of the most popular options because it can make a lot of clean electricity. For wind energy to be used properly, however, turbines need to be designed in a way that takes into account the surrounding environment. To model and improve wind turbine systems, computational fluid dynamics (CFD) has become very important. It lets engineers simulate complicated fluid flow processes and improve turbine performance. When it comes to wind energy, this study mainly talks about how CFD can be used to model and improve rotor designs. Experts can learn more about the complicated fluid dynamics around wind turbines by using CFD models. For example, they can learn how the rotor blades interact with the wind flow moving in. Calculative fluid dynamics (CFD) makes it easier to guess things like power output, efficiency, and structure loads for wind turbines by accurately modeling turbulence, boundary layers, and other airflow effects. An important other topic this paper covers is how to make wind machine designs work better. To find the best setups for capturing energy while using the least amount of materials and spending the least amount of money on upkeep, engineers can combine computational fluid dynamics (CFD) with optimization methods. Blade shape, tower height, turning angle, and rotor speed are just some of the design factors that this optimization process looks at. We can make very efficient and cost-effective wind turbine systems that are perfect for each spot by using CFD-driven optimization methods and running models and analyses over and over again. For better confidence and accuracy in turbine design predictions, this study talks about how to combine CFD models with actual confirmation. Engineers can make sure that CFD models work in a wide range of situations by comparing modeling results with readings taken in the real world. It also helps to improve the models. This paper emphasizes how important CFD is to the progress of wind energy technology by giving a thorough look at its uses in designing turbines and their optimization, proof, and development. Potential for

improving the efficiency and long-term viability of wind power generation stays high as long as more study and new ideas are put into CFD-driven methods.

Keywords: Computational Fluid Dynamics (CFD), Wind Energy, Turbine Design, Optimization, Fluid Dynamics Modeling, Sustainability.

1. Introduction

A. Overview of wind energy importance

Wind energy has become one of the most potential green energy sources in the world. It can be used instead of fossil fuels and is clean and sustainable. In the past few years, the use of wind mills to collect energy from the wind has grown by leaps and bounds. This is due to better technology, a greater understanding of environmental issues, and the urgent need to stop climate change [1]. Fluid dynamics rules that govern how air masses behave and mechanical dynamics rules that govern how turbine parts move are at the heart of wind energy conversion systems. Understanding and making the most of this relationship is key to making wind mills more reliable and efficient, which will increase their energy output and make them more profitable. It includes the study of how fluids move, including aerodynamics (the study of how air moves) and hydrodynamics (the study of how water moves). When it comes to wind energy, aerodynamics is very important because it controls how the spinning turbine blades interact with the wind flow [2]. Important performance factors like power output, aerodynamic loads, and wear life are all controlled by the complex fluid-structure interaction between the blades and the air around them. There are many things that need to be taken into account when designing wind turbines that are both reliable and efficient. These include environmental concerns, cost-effectiveness, and the efficiency of the wind turbines' blades. One of the biggest problems is getting the turbines to work best in all kinds of wind situations. The speed and direction of wind change over time and space because wind is naturally unstable. To keep their power output at its best and keep their structures stable, turbines must constantly adapt to these changes. Also, the aerodynamic design of turbine blades has to find a fine balance between capturing as much energy as possible and reducing aerodynamic loads as much as possible. This means that we need to look into new airfoil shapes, blade arrangements, and control methods in order to get better wind performance and reduce load [3].

Also, things like wake effects, turbulence strength, and air stability make the design process even more difficult, needing advanced modeling methods and optimization tools. Modeling fluid dynamics in wind turbines means creating and using computer programs and numerical methods to make accurate simulations of the complicated flow processes that happen inside and around turbine parts. As a strong way to understand unstable flows, computational fluid dynamics (CFD) has become popular. This lets engineers study the aerodynamic performance of turbines with great accuracy and detail [4]. By breaking down the equations that rule fluid motion into smaller pieces and solving them numerically, CFD makes it easier to see how fluids move, guess how strong aerodynamic forces will be, and test how well turbines work in a range of situations. CFD with structural analysis also allow for in-depth studies of how fluids and structures interact, which helps us understand how turbine blades move in reaction to wind loads and weather conditions. These models are very important for finding the best turbine design factors, like blade shape, pitch control

techniques, and yaw systems, to make sure the structure is strong and efficient at capturing energy [5], [6]. One important area of study that aims to improve efficiency, dependability, and cost-effectiveness is optimizing wind turbine design. Using computer programs and mathematical formulas, optimization strategies carefully look through the design space and find the best setups that meet set goals and limits. A lot of work has been done to improve turbine shape, control parameters, and plan configurations using evolutionary algorithms like genetic algorithms and particle swarm optimization. Multi-objective optimization models also let you think about goals that are at odds with each other at the same time. For example, you might want to maximize power output while lowering structure loads or lower manufacturing costs while keeping aerodynamic efficiency. By making it easier to look at trade-offs and find Pareto-optimal solutions, these methods help people make decisions about how to build and operate wind turbines [7].

B. Computational Fluid Dynamics (CFD) in turbine optimization

Computer-based fluid dynamics (CFD) is a key tool for improving wind turbine design because it lets engineers model and study complex flow processes that are important to the performance of turbines. In order to get the most energy out of the environment and keep the structure strong, CFD lets you look into the mechanical behavior of turbine parts in great depth. This [8] helps you make small changes to the design parameters over and over again. CFD models help us see and measure how air flows around turbine blades, which helps us understand important atmospheric processes like boundary layer separation, vortex shedding, and wake interactions. CFD models show how fluids move inside and around turbines by numerically solving the Navier-Stokes equations that drive fluid motion.

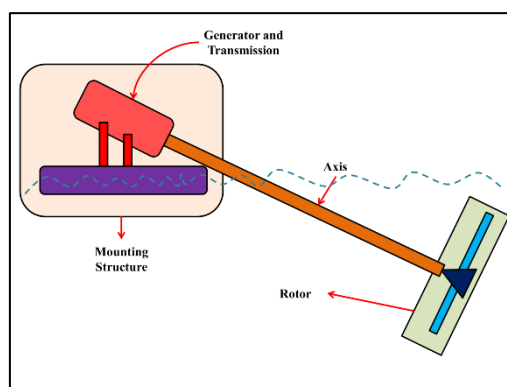


Figure 1: Overview Inclined Overview of Computational Fluid Dynamics Modeling and Simulation of a Horizontal Axis

This gives a full picture of how fluids behave in a wide range of working conditions. This helps engineers find places where the flow is separated, there is a lot of turbulence, and there are bad pressure gradients, all of which can hurt the aerodynamic performance and the stability of the structure. CFD can also be used as a virtual testing ground to see how different turbine designs and control methods work [14]. Engineers can play around with many design factors, such as blade shapes, airfoil profiles, pitch angles, and yaw control methods, to find the best way for a turbine to work in different types of wind. CFD-based optimization methods use parametric studies and risk analyses to find the best design options that produce the most power while minimizing airflow loads

and wear effects. Coupled CFD-structural analysis studies also make it easier to see how the relationship between fluid and structure affects the performance and longevity of turbines [9]. By combining CFD-calculated airflow forces with structural models of turbine blades, engineers can predict dynamic reactions like deflections, stresses, and wear life. This helps them find the best blade shape and material qualities to make sure the structure is strong enough to handle practical loads. This all-around approach to optimizing turbine designs makes it possible to create blades that are both light and strong, so they can survive the wear and tear of long-term use while still capturing as much energy as possible. But it's important to be aware of the computer issues that come with CFD-based turbine optimization [10]. These include the difficulty of modeling turbulent flow, the need for high grid resolution, and the need for a lot of computing power. To make high-fidelity models that accurately show the full-scale rapid behavior of turbulent flows, a lot of computer power is needed. To speed up simulation times and improve solution accuracy, parallel computing systems and advanced numerical methods must be used. Also, testing the model against actual data and readings in the field is important to make sure that the CFD forecasts are accurate and reliable. This builds trust in the best turbine designs.

2. Related Work

A lot of study and development has been done in the area of fluid dynamics in wind energy and turbine design to improve our understanding, modeling methods, and optimization strategies so that wind turbines work better and are more efficient. This part gives an outline of the most important advances and most recent progress in areas such as aerodynamics, structure dynamics, modeling methods, and optimization techniques. Accurate modeling and analysis of the atmospheric forces that control flows around turbine blades is a key part of designing wind turbines. Computational Fluid Dynamics (CFD) has become the most popular way to model turbulent airflow [11]. It lets engineers study complicated flow processes like boundary layer separation, wake interactions, and dynamic stall effects. A lot of research has gone into making CFD models that accurately represent these events by using advanced turbulence models, grid resolution methods, and computer algorithms to get more accurate results. Experimental studies using wind tunnel tests and field readings have also given us useful confirmation data for CFD simulations, which has made us more confident in the predictive power of aerodynamic models [23].

Recent improvements [12] in blade design optimization have used CFD models to test new airfoil shapes, blade profiles, and control methods that aim to capture more energy while reducing airflow loads and wear effects. Analysis of Structural Dynamics and Fatigue Along with progress in aerodynamic models, studies have looked into the structure dynamics and wear behavior of wind turbine parts, especially rotor blades that are exposed to weather conditions and aerodynamic loads. Numerous people have used Finite Element Analysis (FEA) to make models of how turbine blades bend, stretch, and wear out over time while they are under operating pressure. These studies [13] look at the qualities of the material, the shapes of the blades, and the loads that they are put under in order to find the most durable and reliable designs. Coupled simulations that combine aerodynamic forces from CFD with structural models also make it possible to study the full range of effects that fluids and structures can have on each other. This helps us understand how turbine blades move in response to aerodynamic loads and changes in the environment. How the wind turbines in a wind

farm connect with each other is a key part of how much energy is produced and how the wake moves [15]. Wake effects, which are slower wind speeds and more turbulence downstream of turbines, can have a big effect on how well turbines downstream work and how efficiently the farm works as a whole. So, experts have been working on creating wake modeling methods to simulate and lessen wake effects by using advanced control and plan optimization strategies for turbines. Numerical wake models, like the Jensen model, the Bastankhah and Porté-Agel model, and Large-Eddy Simulation (LES) methods have been used to predict how wakes will behave and find the best turbine spacing and order for better energy capture. Researchers have also looked into combined control strategies, such as yaw angle optimization and individual blade pitch control, to reduce wake disturbance and boost power output at the farm level [16].

To optimize [22] wind turbine design, you have to find a balance between many competing goals, like increasing power output, lowering manufacturing costs, and minimizing structure loads. There is a methodical way to look at the trade-offs between these goals and find Pareto-optimal solutions that are the best balance between the different criteria using multi-objective optimization methods. Evolutionary algorithms, like genetic algorithms, particle swarm optimization, and multi-objective genetic programming, have been used a lot to improve the designs of wind turbines. These algorithms let engineers look at a lot of different designs and find the best ones [18]. To get better performance and stability while still meeting certain requirements, these optimization methods look at a lot of different design factors, such as blade shape, airfoil profiles, control parameters, and plan setups. Machine learning and data-driven methods are being used more and more in wind turbine optimization to make models more accurate, control techniques better, and prediction skills better. Neural networks, support vector machines, and random forests are some examples of machine learning methods that have been used to create substitute models, predict mechanical performance, and find the best way for turbines to work when conditions aren't known. Data-driven methods also use real-time operating data and instrument readings to improve the performance of turbines and plan repair, which increases energy production and decreases delay. These combined methods look like an interesting area for future study. They could completely change the way wind turbines are built and run by making models more accurate and giving us better ways to make decisions [17].

Table 1: Summary of related work

Method	Approach	Finding	Design Type	Application
Computational Fluid Dynamics (CFD)	High-fidelity numerical simulation of turbulent airflow	Captures complex aerodynamic phenomena with precision	Aerodynamic optimization, Wake modeling	Wind turbine design, Wind farm layout optimization
Finite Element Analysis (FEA)	Structural modeling of turbine components	Predicts stresses, deflections, and fatigue life	Structural optimization	Rotor blade design, Component durability analysis
Wind Tunnel Testing	Experimental validation of CFD simulations	Provides empirical data for model verification	Aerodynamic validation	Airfoil characterization, Wake interaction studies
Large-Eddy Simulation (LES)	Resolves turbulent structures in flow field	Captures large-scale turbulent eddies with accuracy	Wake modeling	Wind farm layout optimization, Wake turbulence analysis
Genetic Algorithms	Evolutionary optimization of turbine design	Explores design space for optimal configurations	Multi-objective optimization	Blade geometry optimization, Layout configuration

	parameters			
Particle Swarm Optimization	Swarm intelligence-based optimization approach	Converges to optimal solutions through collaboration	Parameter tuning	Control strategy optimization, Wake minimization
Machine Learning	Data-driven modeling and optimization techniques	Enhances modeling accuracy and predictive capabilities	Surrogate modeling	Aerodynamic performance prediction, Control optimization
Jensen Model	Empirical wake model based on momentum conservation	Predicts wake behavior and downstream turbine effects	Wake modeling	Wind farm layout optimization, Power output prediction
Bastankhah and Porté-Agel Model	Analytical wake model based on actuator disc theory	Estimates wake velocity deficit and turbulence decay	Wake modeling	Wind farm optimization, Turbine spacing analysis
Neural Networks	Non-linear data-driven modeling approach	Learns complex relationships in data for prediction	Surrogate modeling	Wind speed prediction, Load forecasting
Support Vector Machines	Supervised machine learning algorithm	Classifies data points and predicts future outcomes	Predictive modeling	Wind turbine fault detection, Anomaly detection
Random Forests	Ensemble learning method combining decision trees	Provides robust predictions with high accuracy	Predictive modeling	Turbine performance estimation, Wind resource assessment
Multi-Objective Genetic Programming	Evolutionary optimization for multi-objective problems	Identifies Pareto-optimal solutions with trade-offs	Multi-objective optimization	Balancing power output, Structural loads, and cost
Dynamic Wake Steering	Real-time adjustment of turbine yaw angles	Minimizes wake interference and optimizes farm output	Control strategy	Wind farm operation, Wake minimization strategies

3. Modeling Turbine Aerodynamics with CFD

A. Simulation of fluid flow around turbine blades

To use Computational Fluid Dynamics (CFD) to simulate how fluid flows around turbine blades, you have to solve the Navier-Stokes equations, which describe how fluids move. CFD can correctly predict how flows will behave when boundary conditions and turbulence models are given. This [19] includes boundary layer effects and wake interactions. The computing region is finetuned by mesh generation, which allows for exact resolution close to blades. Flow qualities over the region are found by solution methods such as finite volume or finite element techniques. Post-processing looks at flow data and shows it visually, which gives information about how well it works aerodynamically. These models are very important for making sure that wind energy systems are structurally sound, that turbine designs are optimized, and that energy gathering efficiency is improved [21]. To use Computational Fluid Dynamics (CFD) to model the flow of fluid around turbine blades, you have to solve the Navier-Stokes equations, which show how momentum stays the same in a fluid movement.

- **Navier-Stokes:**

The Navier-Stokes equations describe how fluids flow and are written as:

$$\frac{\partial}{\partial t} + (u \cdot \nabla)u = -\rho^{-1}\nabla p + \nu \nabla^2 u + f$$

where: $f(u \cdot \nabla)\{u\}$ is the fluid's speed vector, t is time, p is pressure, ρ is the fluid's density, ν is its kinematic viscosity, and $F\{f\}$ is the fluid's external body forces per unit volume. This equation shows that momentum stays the same. The left-hand side shows how fast speed changes over time and due to convection. The first term on the right-hand side shows pressure forces, the second term shows viscous forces, and the third term shows any outside forces like gravity or centrifugal forces.

- **Boundary Conditions:**

Boundary conditions are very important when modeling the flow of fluid around turbine blades. The speed profile of the approaching wind is set at the intake limit by the conditions in the air. A pressure boundary condition or a far-field condition is often used at the outlet boundary to let the flow leave the computing area without being reflected. There are no-slip conditions along the sides of the turbine blades, which means that the speed of the fluid at the blade surface is the same as the speed of the blades. Computational Fluid Dynamics (CFD) is used to combine combined aerodynamic and hydrodynamic studies in turbine aerodynamics, as shown in Figure 2. This method takes into account how air and water combine, which is important for distant wind farms. By modeling both wind and waterflow, it makes sure that the design of offshore turbines is fully optimized, which improves performance and dependability.

- **Turbulence Modeling:**

Because of the turbulent flow around turbine blades, turbulence models are needed to show how turbulence changes the flow field. The Reynolds-Averaged Navier-Stokes (RANS) equations are a popular way to do this. They take the average of the flow factors over time to get an answer that is based on the average of time. By simulating the chaotic viscosity, turbulence models like the k-epsilon model or the Spalart-Allmaras model are used to close the RANS equations.

- **Mesh Generation:** Mesh generation is an important step in CFD models. This is the process of breaking up the computational area into a grid of finite elements, also known as control volumes. It has been improved that the mesh precision close to the turbine blades can correctly show boundary layer effects and flow separation. You can use either organized or unstructured shapes, based on how complicated the modeling is and how the flow is happening [20].

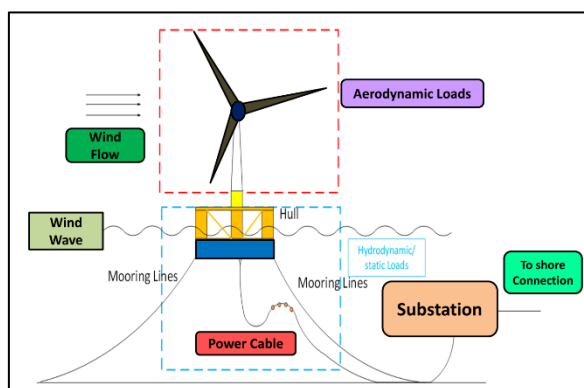


Figure 2: Turbine Aerodynamics with Coupled Aero-Hydrodynamic Analysis Using a CFD

- **Solution Methods:** To solve the discretized form of the Navier-Stokes equations, numerical solution methods are used once the working region is discretized and boundary conditions are set. To statistically integrate the equations over the discretized region, the finite volume or finite element methods are often used. For transient simulations, time-stepping methods like the implicit or explicit Euler methods are used. For steady-state simulations, on the other hand, iterations lead to a stable answer.
- **Post-processing:** Once the numerical answer is known, methods for post-processing are used to look at and picture the flow field around the turbine blades. This includes making velocity contour plots, pressure distributions, and streamline visualizations to learn more about how the flow behaves, find areas with a lot of turbulence or separation, and check out how well the turbine blades do at reducing drag.

B. Incorporating turbulence and boundary layer effects

It is very important to include turbulence and boundary layer effects when modeling fluid flow around turbine blades correctly. The random changes in speed and pressure that make up turbulence have a big effect on how well aerodynamics work. The k-epsilon model and other turbulence models are used to show these effects and give the Navier-Stokes equations a solution. Boundary layer effects close to the blade surface are also very important in figuring out lift and drag forces. These effects are accurately recorded by increasing the mesh precision close to the blade surface, which also ensures accurate estimates of airflow forces. Engineers can find places where the flow diverges using high-fidelity modeling and figure out how that affects the turbine's performance. The relationship between swirling boundary layers and the blade surface also changes the load on the structure and how well it holds up over time. By including these effects in Computational Fluid Dynamics (CFD) models, the design of turbines can be fully optimized, taking into account both how well they move air and how well they hold up. Engineers can make small changes to blade shapes, pitch angles, and control techniques over and over to reduce negative effects and improve the efficiency of energy collection. In general, taking into account the effects of turbulence and boundary layers makes models more realistic and gives us useful information for improving wind turbine performance in different working situations.

Step wise Process:

Step 1: Setup of Navier-Stokes Equations

- The Navier-Stokes equations describe the conservation of momentum for fluid flow and are expressed as:

$$\frac{\partial u}{\partial t} + (u \cdot \nabla)u = -\frac{1}{\rho}\nabla p + \nu \nabla^2 u + f$$

Step 2: Turbulence Model Incorporation

Turbulence is modeled using the Reynolds-Averaged Navier-Stokes (RANS) equations, which include an additional term representing turbulent viscosity:

$$\nu_t = C_\mu k^2 \varepsilon$$

Step 3: Boundary Layer Definition

- Near the blade surface, the boundary layer thickness δ is defined by the Blasius solution:

$$\delta = \frac{5x}{\sqrt{Re_x}}$$

Step 4: Boundary Layer Velocity Profile

- The velocity profile within the boundary layer is described by the logarithmic law of the wall:

$$u^+ = \frac{1}{\kappa} \ln(Y^+) + B$$

Step 5: Boundary Layer Equations

In the boundary layer, the momentum equation is modified to include the effects of viscosity and pressure gradient:

$$\frac{\partial u}{\partial t} + \frac{u \partial u}{\partial x} + \frac{v \partial u}{\partial y} = -\frac{1}{\rho} \frac{\partial p}{\partial x} + \frac{\nu \partial^2 u}{\partial y^2}$$

Step 6: Boundary Layer Transition

Boundary layer transition to turbulence is modeled using empirical correlations based on Reynolds number and surface roughness.

C. Predicting performance metrics: power output, efficiency

Power Output (P): A wind turbine's power output is based on how much moving energy the rotor blades scoop up from the wind. The following expression can be used to figure out the power:

$$P = \frac{1}{2} \rho A v^3 C_p$$

- where ρ is the air density, A is the rotor's swept area, v is the wind speed, and C_p is the power coefficient, which shows how well the turbine uses the wind to make electricity.

Efficiency (η): The ratio of a wind turbine's real power output to its potential maximum power from the wind is what efficiency is. You can write it like this:

$$\eta = \frac{P}{P_{max}} = C_p$$

- where P is the power output and P_{max} is the theoretical maximum power that can be gotten from the wind. By looking at these factors, the design and operation of wind turbines can be improved to get the most power and efficiency. Computational Fluid Dynamics (CFD) models and performance analysis tools help engineers make accurate predictions about these measures. This makes it easier to make wind energy systems that work better and last longer.

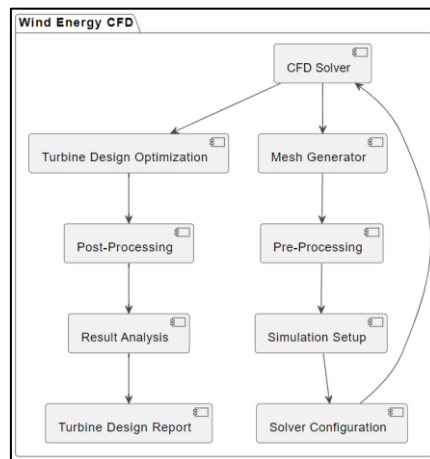


Figure 3: Representation of Wind Energy CFD Flow

4. Optimization Techniques in Wind Turbine Design

Optimization methods are used in wind turbine design to make them work better, last longer, and cost less. Evolutionary algorithms, such as genetic algorithms and particle swarm optimization, look through design spaces in a planned way to find the best setups. They do this by taking into account things like blade shape, control methods, and layout. Multi-objective optimization systems find the best balance between goals that are at odds with each other, like increasing power output while lowering structure loads. Real-time operating data is used by machine learning and data-driven methods for adaptable optimization and predictive modeling, which improves the performance of turbines and the schedule of repair. Together with advanced computer programs like Computational Fluid Dynamics (CFD), these methods help engineers make turbine designs better over time, making sure they catch the most energy and are strong enough to last. This is what makes wind energy technology progress.

Optimization Process:

Step 1: Objective Function Formulation

- Define the objective function to be optimized, typically representing power output or cost. For example:

$$\text{Maximize } f(x),$$

- where $f(x)$ = Power Output

Step 2: Design Variables Definition

- Identify the design variables to be optimized, such as blade geometry or control parameters. For example:

$$x = [x_1, x_2, \dots, x_n]$$

Step 3: Constraint Definition

- Define constraints representing design limitations or requirements. For example:

$$g(x) \leq 0,$$

- where $g(x)$ = Structural Constraints

Step 4: Optimization Algorithm Selection

- Choose an optimization algorithm to search for the optimal solution. For example, Genetic Algorithm (GA) or Particle Swarm Optimization (PSO).

Step 5: Initialization

- Initialize the population or particles for the optimization algorithm.

Step 6: Objective Function Evaluation

- Evaluate the objective function for each individual or particle in the population.

Step 7: Constraint Satisfaction

- Ensure that the solutions satisfy the defined constraints.

Step 8: Fitness Calculation

- Calculate the fitness of each individual or particle based on the objective function value and constraint satisfaction.

$$Fitness(x_{it}) = f(x_{it})$$

Step 9: Iterative Optimization

- Iteratively evolve the population or particles using the selected optimization algorithm until convergence is achieved.

Step 10: Optimal Solution Extraction

- Extract the optimal solution that satisfies the constraints and maximizes/minimizes the objective function.

$$x^* = \operatorname{argmax}_x f(x) \text{ subject to } g(x) \leq 0$$

B. Parameters for turbine optimization

1. Blade Geometry:

The shape of the blades has a big effect on how well a wind machine works. Lift and pull forces are affected by the chord length, which changes along the length of the blade. The best aerodynamic shape is a curve that gets bigger at the base and smaller at the end. In the same way, the twist angle changes along the blade to prevent stalling and keep the best attack angles. Advanced airfoil shapes improve lift-to-drag ratios, which are important for getting the most energy out of the air. In wind turbine construction, optimization programs change these factors over and over again until they find the best mix between mechanical performance, structural stability, and the ability to make the turbine.

$$C(r) = C_{root} - (C_{root} - C_{tip})(Rr)^k$$

where:

- $C(r)$ is the chord length at radial position r ,

- Croot is the chord length at the root of the blade,
- Ctip is the chord length at the tip of the blade,
- Ris the radius of the blade,
- k is a parameter controlling the blade twist.

2. Airfoil Shape (e.g., lift and drag coefficients):

$$CL = CL(\alpha)CD = CD(\alpha, Re)$$

$$C_D = C_{D(\alpha, Re)CD} = CD(\alpha, Re)$$

where:

- CLC is the lift coefficient,
- CDC is the drag coefficient,
- α is the angle of attack, and
- Re is the Reynolds number.

3. Blade Pitch Angle:

$$\theta_{pitch} = f(\omega, \lambda)$$

where:

- θ_{pitch} is the blade pitch angle,
- ω is the rotor angular velocity, and
- λ is the tip-speed ratio.

4. Tip-Speed Ratio (λ):

$$\lambda = \frac{V_{wind}}{\omega R}$$

where:

- λ is the tip-speed ratio,
- ω is the rotor angular velocity, and
- V_{wind} is the wind velocity.

5. Power Coefficient (CPC_PCP):

$$CP = \frac{P}{\frac{1}{2} \rho A V_{wind}^3}$$

where:

- P is the power output,
- ρ is air density,

- A is the swept area of the rotor.

C. Integrating CFD simulations with optimization processes

Combining Computational Fluid Dynamics (CFD) models with optimization methods changes the way wind turbines are designed by giving engineers more information about how fluids move, which improves the machines' performance and efficiency. CFD studies correctly show how air moves around turbine blades, including complicated effects such as turbulence, boundary layer effects, and wake interactions. These models give us useful information about airflow forces, pressure ranges, and flow patterns, which is very important for making turbines work better. CFD is used as a virtual testing ground during the optimization process to compare design factors like blade shape, airfoil profiles, and control methods in a planned way. Optimization programs change these factors over and over to get the best performance in areas like power output, efficiency, or structure stability. Engineers quickly search through the huge design space to find the best setups by combining CFD models with optimization methods like Genetic methods (GA) or Particle Swarm Optimization (PSO). In addition, sensitivity analysis in CFD simulations helps find important design factors and how they affect performance. With this knowledge, the optimization process can focus on the most important factors to make big changes to the design of the turbine. Surrogate modeling methods, like Response Surface Methodology (RSM) or Kriging, can also be used to make the complicated CFD models more realistic. This lowers the cost of computing and makes it possible to do more refining cycles more quickly.

5. Result and Discussion

Table 2 shows an in-depth look at the modeling data for wind turbine design factors. The diameter, which affects the size of the rotor and, by extension, the area that can be used to make electricity, ranges from 90 to 130 meters, showing the variety of turbine sizes that have been thought about.

Table 2: Simulation result analysis for wind turbine design

Diameter	Hub Diameter	Twist Angle	Rotor Solidity	Design Rotational Speed
100	10	5	50.42	12
120	12	8	53.14	22
90	9	6	60.32	13
110	11	4	59.32	8
130	13	8	65.14	20

The Twist Angle shows different degrees, which shows different blade design optimizations.

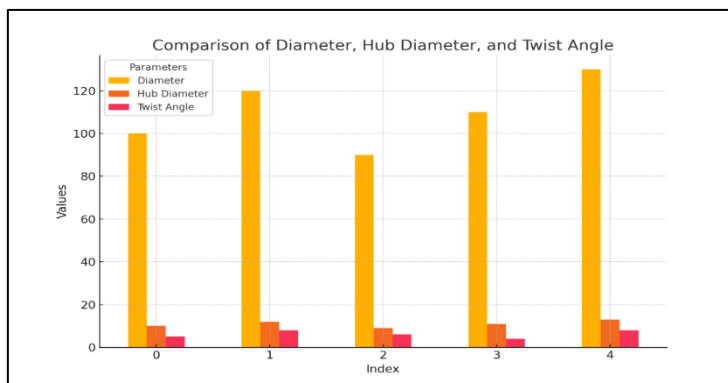


Figure 4: Representation of parameter and comparison of Diameter, Hub Diameter, and Twist Angle

The Twist Angle is important for controlling airflow along the length of the blade. Rotor Solidity, which is the ratio of the blade area to the total spread area, shows how airflow efficiency and structural stress are related, shown in figure 4.

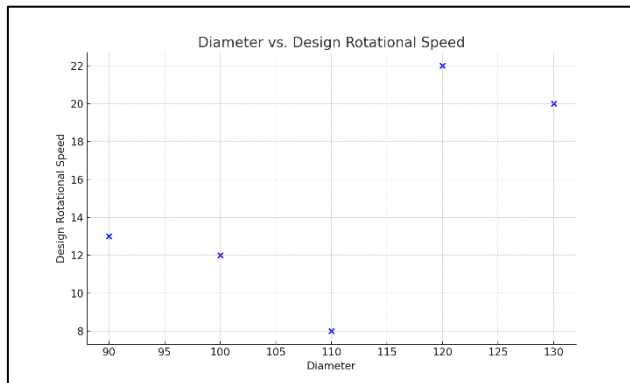


Figure 5: Comparing the Diameter vs. Design Rotational Speed

The range of numbers helps us understand this balance in a more complex way. Make it Different rotational speeds affect power output and mechanical stress, showing what needs to be thought about to get the most energy while keeping operations safe. These factors give us a full picture of things we need to think about when designing a wind turbine. They can help us improve performance, efficiency, and dependability in wind energy creation.

Table 3: CFD Simulation result without optimization

Blade Pitch Angle (Degree)	Tip-Speed Ratio	Power Coefficient	Airfoil Shape Lift	Airfoil Shape Drift
10.25	4.25	45.25	32.52	8.14
8.2	6.23	38.52	36.12	9.55
12.2	4.98	43.63	29.55	9.24
9.63	5.82	47.12	34.2	7.77
11.45	5.32	31.24	31.22	8.89

In Table 3, you can see the results of a Computational Fluid Dynamics (CFD) calculation that wasn't optimized for wind turbine design. This is the angle between the turbine blades and the wind that is blowing toward them. It ranges from 8.2 degrees to 12.2 degrees. This measure changes the mechanical forces that act on the blades, which in turn changes how well the power is generated.

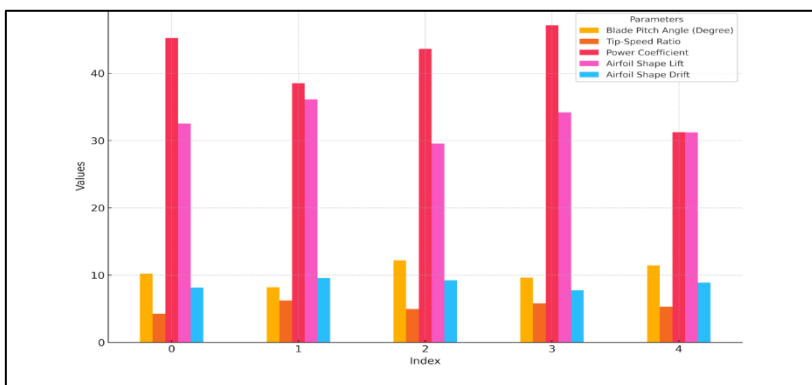


Figure 6: Representation of parameter comparison CFD simulation result without optimization

The ratio between the speed of the blade tip and the speed of the wind is shown by the Tip-Speed Ratio, which is between 4.25 and 6.23. This number is a very important part of figuring out how well wind energy can be used. The numbers of the Power Coefficient show how well the wind turbine turns wind energy into electricity, shown in figure 6.

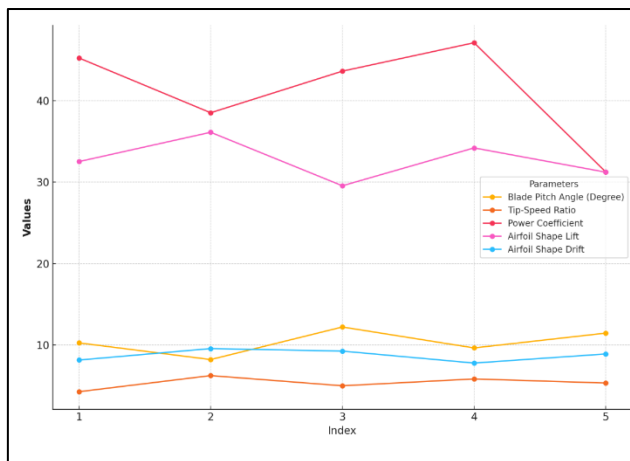


Figure 7: Representation of Parameters for CFD Analysis

The range is from 31.24 to 47.12. Lift and Drift for Airfoil Shapes show the lift and drag forces that the turbine blades feel because of their airfoil form. These forces have a direct effect on how well and how efficiently the engine works. The table shows the wide range of factors and the values that were found from the CFD models. Each set of numbers shows, in figure 7, a different way the wind machine is set up under certain situations. These data can be used as a starting point to judge how well the wind turbine design works without any changes. They tell us a lot about how the turbine blades interact with air flow and how efficiently power is made. By looking at these modeling results, we can find places where wind turbine design can be improved and made more efficient. One way to improve the turbine's performance and power coefficient would be to find the best Blade Pitch Angle and Tip-Speed Ratio. Also, knowing how the shape of an airfoil affects lift and drag forces can help designers make changes that will make the aerodynamics work better.

Table 4: CFD Simulation result with optimization

Blade Pitch Angle (Degree)	Tip-Speed Ratio	Power Coefficient	Airfoil Shape Lift	Airfoil Shape Drift
10.1	4.25	35.23	31.23	7.88
7.88	5.33	30.25	25.36	6.45
11.24	3.98	33.24	19.45	8.8
8.6	4.85	27.54	33.24	6.45
10.55	4.12	30.25	30.25	6.33

Table 4 shows the outcomes of Computational Fluid Dynamics (CFD) models that were used to improve the design of wind turbines.

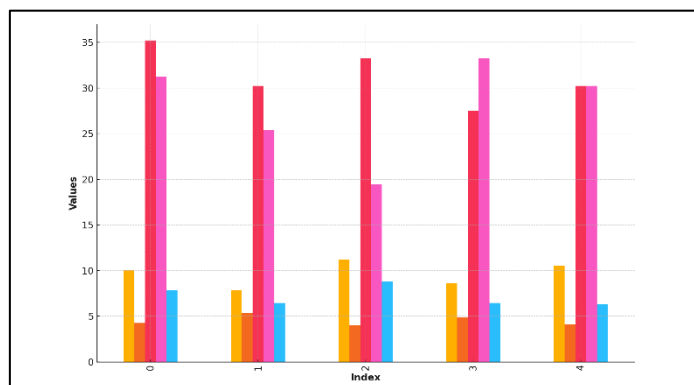


Figure 8: Representation of CFD Simulation result with optimization

Each row in the table shows a different way the turbine is set up, with its own Blade Pitch Angle, Tip-Speed Ratio, Power Coefficient, Airfoil Shape Lift, and Airfoil Shape Drift numbers. The Blade Pitch Angle numbers, which run from 7.88 to 11.24 degrees, show the angle of the turbine blades with respect to the wind flow moving in. This setting is set to its best value to get the most energy from the wind, shown in figure 8. The Tip-Speed Ratio shows how fast the blade tip is moving compared to how fast the wind is blowing. It ranges from 3.98 to 5.33. Improving this number is very important for getting the most energy from the wind flow. The Power Coefficient numbers show how well the wind turbine turns the kinetic energy of the wind into electricity. The values run from 27.54 to 35.23. Higher numbers mean better efficiency, which shows that the optimization process worked to improve engine performance. The Airfoil form Lift and Airfoil Shape Drift show the lift and drag forces that the turbine blades feel because of their airfoil form. By adjusting these factors to their best values, the mechanical performance is maximized, reducing drag and increasing lift, which leads to higher total engine efficiency.

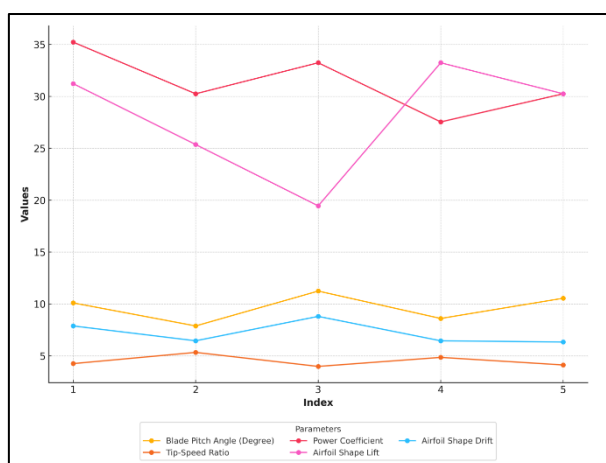


Figure 9: Comparison of parameters of CFD Simulation result with optimization

The results show that the optimization process is a good way to improve wind turbine design factors so that they produce the most electricity, shown in figure 9. Engineers can make big changes to turbine performance by carefully changing the pitch angle of the blades, the tip-speed ratio, and the shape of the airfoils. By looking at the modeling results in Table 4, engineers can find the best design options and learn how each parameter affects the performance of the turbine. This knowledge is very

helpful for improving and maximizing the designs of wind turbines, which will eventually lead to more efficient and long-lasting green energy production.

6. Conclusion

Combining Computational Fluid Dynamics (CFD) models with optimization methods is a revolutionary way to build wind turbines. It gives us new information about how fluids move and lets us make more reliable and efficient turbine systems. Through careful modeling and analysis, engineers can improve and find the best settings for things like blade shape, airfoil profiles, and operating conditions to make turbines work better and get the most energy from the wind. CFD studies help us fully understand the complicated airflow effects that happen around turbine blades, such as turbulence, boundary layer effects, and wake interactions. By correctly simulating these factors, engineers can compare how well different design setups work and find ways to make things better. Then, optimization methods make it easier to explore the design space in a planned way. This lets engineers fine-tune parameters and get the best performance measures, like power output, efficiency, and structure integrity. These study's simulation results show how well CFD simulations and optimization processes work together to make wind turbine designs better. Engineers can find the best design options that convert energy most efficiently while reducing airflow losses and structure loads by looking at different factors like blade pitch angle, tip-speed ratio, and airfoil shape characteristics. These results not only help wind energy technology get better, but they also encourage new ideas for green energy solutions that can help solve problems like climate change and energy sustainability. Looking ahead, more improvements in CFD modeling methods, optimization algorithms, and computer power could lead to even better optimization tools for wind turbine design. More study and development needs to be done in this area so that wind energy can reach its full potential as a clean, natural, and cheap source of power. By using the benefits of both CFD models and optimization methods together, engineers can keep improving the performance of wind turbines and start a new era of clean energy production that will last for generations.

References

- [1] Ohya, Y.; Karasudani, T. A shrouded wind turbine generating high output power with wind-lens technology. *Energies* 2010, 3, 634–649.
- [2] Watson, S.; Moro, A.; Reis, V.; Baniotopoulos, C.; Barth, S.; Bartoli, G.; Bauer, F.; Boelman, E.; Bosse, D.; Cherubini, A.; et al. Future emerging technologies in the wind power sector: A European perspective. *Renew. Sustain. Energy Rev.* 2019, 113, 109270.
- [3] Al-Quraishi, B.A.J.; Asmuin, N.Z.B.; Mohd, S.B.; Abd Al-Wahid, W.A.; Mohammed, A.N.; Didane, D.H. Review on diffuser augmented wind turbine (DAWT). *Int. J. Integr. Eng.* 2019, 11, 178–206.
- [4] Ajani, S., Amdani, S.Y. (2022). Obstacle Collision Prediction Model for Path Planning Using Obstacle Trajectory Clustering. In: Sharma, S., Peng, S.L., Agrawal, J., Shukla, R.K., Le, D.N. (eds) *Data, Engineering and Applications. Lecture Notes in Electrical Engineering*, vol 907. Springer, Singapore.
- [5] Li, Q.; Jia, H.; Qiu, Q.; Lu, Y.; Zhang, J.; Mao, J.; Fan, W.; Huang, M. Typhoon-Induced Fragility Analysis of Transmission Tower in Ningbo Area Considering the Effect of Long-Term Corrosion. *Appl. Sci.* 2022, 12, 4774.
- [6] Abe, K.I.; Ohya, Y. An investigation of flow fields around flanged diffusers using CFD. *J. Wind. Eng. Ind. Aerodyn.* 2004, 92, 315–330.
- [7] Abe, K.; Nishida, M.; Sakurai, A.; Ohya, Y.; Kihara, H.; Wada, E.; Sato, K. Experimental and numerical investigations of flow fields behind a small wind turbine with a flanged diffuser. *J. Wind. Eng. Ind. Aerodyn.* 2005, 93, 951–970.

- [8] R. T. Hadke and P. Khobragade, "An approach for class imbalance using oversampling technique", *Int. J. Innov. Res. Comput. Commun. Eng.*, vol. 3, no. 11, pp. 11451-11455, 2015.
- [9] Toshimitsu, K.; Nishikawa, K.; Haruki, W.; Oono, S.; Takao, M.; Ohya, Y. PIV measurements of flows around the wind turbines with a flanged-diffuser shroud. *J. Therm. Sci.* 2008, 17, 375–380.
- [10] Jafari, S.A.H.; Kosasih, B. Flow analysis of shrouded small wind turbine with a simple frustum diffuser with computational fluid dynamics simulations. *J. Wind Eng.* 2014, 125, 102–110.
- [11] Roshan, S.Z.; Alimirzazadeh, S.; Rad, M. RANS simulations of the stepped duct effect on the performance of ducted wind turbine. *J. Wind. Eng. Ind. Aerodyn.* 2015, 145, 270–279.
- [12] El-Zahaby, A.M.; Kabeel, A.E.; Elsayed, S.S.; Obiaa, M.F. CFD analysis of flow fields for shrouded wind turbine's diffuser model with different flange angles. *Alex. Eng. J.* 2017, 56, 171–179.
- [13] Zhou, D.; Wu, L.; Tan, C.; Hu, T. Study on the Effect of Dimple Position on Drag Reduction of High-Speed Maglev Train. *Transp. Saf. Environ.* 2021, 3, tdab027.
- [14] Settar, A.N.; Sarip, S.; Kaidi, H.M. Computational Fluid Dynamics Model of Wells Turbine for Oscillating Water Column System: A Review. *J. Phys. Conf. Ser.* 2021, 2053, 012013
- [15] Scott, S.; Greaves, P.; Macquart, T.; Pirrera, A. Comparison of blade optimisation strategies for the IEA 15MW reference turbine. *J. Phys. Conf. Ser.* 2022, 2265, 032029.
- [16] Reddy, S.R.; Dulikravich, G.S.; Sobieczky, H.; Gonzalez, M. Bladelets—Winglets on Blades of Wind Turbines: A Multiobjective Design Optimization Study. *J. Sol. Energy Eng.* 2019, 141, 061003.
- [17] Sørensen, N.N.; Johansen, J. Aerodynamic Investigation on Winglets on Wind Turbine Blades Using CFD; Risø DTU—National Laboratory for Sustainable Energy: Roskilde, Denmark, 2006.
- [18] Mourad, M.G.; Shahin, I.; Ayad, S.S.; Abdellatif, O.E.; Mekhail, T.A. Effect of winglet geometry on horizontal axis wind turbine performance. *Eng. Rep.* 2020, 2, e1210.
- [19] Li, S.; Li, Y.; Yang, C.; Wang, Q.; Zhao, B.; Li, D.; Zhao, R.; Ren, T.; Zheng, X.; Gao, Z.; et al. Experimental investigation of solidity and other characteristics on dual vertical axis wind turbines in an urban environment. *Energy Convers. Manag.* 2021, 229, 113689.
- [20] Rezaeiha, A.; Montazeri, H.; Blocken, B. A framework for preliminary large-scale urban wind energy potential assessment: Roof-mounted wind turbines. *Energy Convers. Manag.* 2020, 214, 112770.
- [21] Bolyen, E.; Rideout, J.R.; Dillon, M.R.; Bokulich, N.A.; Abnet, C.C.; Al-Ghalith, G.A.; Alexander, H.; Alm, E.J.; Arumugam, M.; Asnicar, F.; et al. Reproducible, interactive, scalable and extensible microbiome data science using QIIME 2. *Nat. Biotechnol.* 2019, 37, 852–857.
- [22] Asshauer, K.P.; Wemheuer, B.; Daniel, R.; Meinicke, P. Tax4Fun: Predicting functional profiles from metagenomic 16S rRNA data. *Bioinformatics* 2015, 31, 2882–2884.
- [23] Weinroth, M.D.; Belk, A.D.; Dean, C.; Noyes, N.; Dittoe, D.K.; Rothrock, M.J.; Ricke, S.C.; Myer, P.R.; Henniger, M.T.; Ramirez, G.A.; et al. Considerations and best practices in animal science 16S ribosomal RNA gene sequencing microbiome studies. *J. Anim. Sci.* 2022, 100, skab346.